

SIEMENS

SINUMERIK 840D/840Di/810D HMI Advanced

Operator's Guide

Valid for

<i>Control</i>	<i>Software version</i>
SINUMERIK 840D	6
SINUMERIK 840DE (export version)	6
SINUMERIK 840D (powerline)	6
SINUMERIK 840DE (powerline export version)	6
SINUMERIK 840Di	2
SINUMERIK 840DiE (export version)	2
SINUMERIK 810D	3
SINUMERIK 810DE (export version)	3
SINUMERIK 810D (powerline)	6
SINUMERIK 810DE (powerline export version)	6
HMI Advanced	6

11.02 Edition

Introduction	1
Operator Components/ Operating Sequences	2
Operating Example	3
Machine	4
Parameters	5
Program	6
Services	7
Diagnosis	8
Start-Up	9
Maintenance	10
Appendix	A

SINUMERIK® Documentation

Printing history

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

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

Status code in the "Remarks" column:

- A** New documentation.
- B** Unrevised reprint with new Order No.
- C** Revised edition with new status.

If factual changes have been made on the page since the last edition, this is indicated by a new edition coding in the header on that page.

Edition	Order No.	Remarks
02.01	6FC5298-6AF00-0BP0	A
11.02	6FC5298-6AF00-0BP2	C

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

Edition	Order No.	Remarks
11.02	6FC5298-6CA00-0BG3	C

Trademarks

SIMATIC®, **SIMATIC HMI®**, **SIMATIC NET®**, **SIROTEC®**, **SINUMERIK®** and **SIMODRIVE®** are registered trademarks of Siemens AG. The other designations in this publication may also be trademarks, the use of which by third parties may constitute copyright violation.

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

This publication was produced with WinWord V 8.0 and Designer V 7.0. The reproduction, transmission or use of this document or its contents is not permitted without express written authority. Therefore we cannot guarantee that they are completely identical. Offenders will be liable for damages. All rights, including rights created by patent grant or registration of a utility model or design, are reserved.

© Siemens AG 1995–2002. All rights reserved

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

We have checked that the contents of this document correspond to the hardware and software described. Nonetheless, differences might exist and therefore we cannot guarantee that they are completely identical. The information contained in this document is, however, reviewed regularly and any necessary changes will be included in the next edition. We welcome suggestions for improvement.

Subject to change without prior notice.

Introduction	1-21
1.1 The product SINUMERIK 840D/810D.....	1-22
1.2 Handling instructions.....	1-24
1.3 Switching the control ON and OFF	1-25
Operator Components/Operating Sequences	2-27
2.1 Operator panel fronts.....	2-29
2.1.1 Operator panel front OP 010.....	2-29
2.1.2 Operator panel front OP 010S	2-30
2.1.3 Operator panel front OP 010C.....	2-30
2.1.4 Operator panel front OP 012.....	2-31
2.1.5 Operator panel front OP 015.....	2-31
2.1.6 Full standard keyboard	2-32
2.2 Operator panel front keys	2-34
2.3 Machine control panels.....	2-39
2.4 Keys of the machine control panel.....	2-40
2.4.1 EMERGENCY STOP	2-40
2.4.2 Operating modes and machine functions	2-40
2.4.3 Feed control	2-42
2.4.4 Spindle control	2-44
2.4.5 Keyswitch.....	2-45
2.4.6 Program control	2-47
2.5 Screen layout	2-49
2.5.1 Overview.....	2-49
2.5.2 Global machine status display	2-50
2.5.3 Program control display	2-55
2.6 General operating sequences.....	2-57
2.6.1 Program overview and program selection	2-57
2.6.2 Changing the menu window.....	2-58
2.6.3 Selecting a directory/file.....	2-59
2.6.4 Editing entries/values	2-60
2.6.5 Confirming/canceling an input.....	2-62
2.6.6 Editing a parts program in the ASCII editor	2-62
2.6.7 Switch channel over.....	2-69
2.6.8 m:n communication links	2-70
2.6.9 Pocket calculator function.....	2-74
2.7 Help.....	2-75
2.7.1 Editor Help	2-77
2.7.2 Short help for program commands	2-78

2.7.3	Extended help for programming commands.....	2-82
2.8	Job list.....	2-83
2.8.1	Description of syntax for job lists.....	2-85
2.8.2	Example of a job list for two-channel 1:1 links.....	2-89
2.8.3	Example of a job list for multi-channel m:n links.....	2-90
2.8.4	Sequence of operations "Executing job list".....	2-91
2.8.5	Rename workpieces with job lists.....	2-93
2.8.6	Copy workpieces with job lists.....	2-94
2.8.7	Archiving workpieces with job lists with m:n.....	2-94
Operating Example		3-95
	Typical operating sequence.....	3-95
"Machine" Operating Area		4-97
4.1	Data structure of the numerical control.....	4-99
4.1.1	Operating modes and machine functions.....	4-100
4.1.2	Operating mode group and channels.....	4-102
4.1.3	Status display with symbols for all channels (SW 6.2 or later).....	4-103
4.1.4	Select operating mode, change operating mode.....	4-104
4.2	General functions and displays.....	4-107
4.2.1	Start/stop/cancel/continue parts programs.....	4-107
4.2.2	Display program level.....	4-108
4.2.3	Switching between the machine/workpiece coordinate system (Machine-MCS/Work- WCS).....	4-109
4.2.4	Displaying axis feedrates.....	4-111
4.2.5	Displaying G functions and transformations.....	4-111
4.2.6	Displaying auxiliary functions.....	4-112
4.2.7	Displaying spindles.....	4-113
4.2.8	Handwheel.....	4-114
4.2.9	Status of synchronized actions.....	4-115
4.2.10	Preset.....	4-117
4.2.11	Set actual value.....	4-119
4.2.12	Inch/metric switchover.....	4-121
4.3	Reference point approach.....	4-123
4.4	Jog mode.....	4-126
4.4.1	Function and basic display.....	4-126
4.4.2	Traversing axes.....	4-129
4.4.3	Inc: Incremental dimension.....	4-130
4.4.4	Repos (repositioning).....	4-131
4.4.5	SI (Safety Integrated): User confirmation.....	4-132
4.4.6	Scratching (SW 6 or later).....	4-134
4.4.7	Scratching (SW 5 or earlier).....	4-137

4.4.8	Display of system frames.....	4-140
4.5	MDI mode	4-142
4.5.1	Function and basic display.....	4-142
4.5.2	Save program, file function	4-145
4.5.3	Teach In	4-146
4.6	Automatic mode.....	4-149
4.6.1	Function and basic display.....	4-149
4.6.2	Program overview	4-151
4.6.3	Loading and unloading the workpiece/parts program	4-152
4.6.4	Log: Loading list of the programs.....	4-153
4.6.5	Executing programs from the hard disk.....	4-154
4.6.6	Access to external network drive	4-155
4.6.7	Program editing.....	4-157
4.6.8	Block search/setting the search destination	4-159
4.6.9	Fast external block search.....	4-162
4.6.10	Start of search in Program test mode, multi-channel	4-166
4.6.11	Overstore	4-169
4.6.12	Program control	4-171
4.6.13	DRF offset.....	4-175
"Parameters" Operating Area		5-177
5.1	Tool data	5-179
5.1.1	Structure of tool compensation	5-179
5.1.2	Tool types and tool parameters	5-179
5.2	Tool offset	5-196
5.2.1	Function and basic display of tool offset.....	5-196
5.2.2	New tool	5-198
5.2.3	Display tool.....	5-200
5.2.4	Delete tool.....	5-201
5.2.5	New tool edge	5-201
5.2.6	Delete tool edge.....	5-202
5.2.7	Determine tool offsets.....	5-203
5.2.8	Immediate activation of tool offset	5-204
5.3	Tool management.....	5-205
5.3.1	Basic functions of tool management.....	5-207
5.3.2	Display/change tool data.....	5-215
5.3.3	Changing the meaning/display of wear data for tools.....	5-219
5.3.4	Grinding data extension in SW 6.2 or later	5-221
5.3.5	Loading	5-224
5.3.6	Unloading.....	5-229
5.3.7	Relocation	5-230
5.3.8	Tool master data in tool catalog.....	5-232

5.3.9	Tool offset data in the tool cabinet	5-235
5.3.10	Job processing for tools	5-238
5.4	R variables (arithmetic variables).....	5-246
5.4.1	Function.....	5-246
5.4.2	Editing/deleting/finding R variables	5-246
5.5	Setting data	5-248
5.5.1	Working area limitation	5-248
5.5.2	Jog data.....	5-249
5.5.3	Spindle data	5-250
5.5.4	Dry run feedrate for DRY mode	5-251
5.5.5	Starting angle for thread cutting	5-252
5.5.6	Other types of setting data	5-253
5.5.7	Protection zones.....	5-254
5.5.8	Electronic gear (SW 6.3 and higher).....	5-255
5.6	Work offset.....	5-256
5.6.1	Function.....	5-256
5.6.2	Changing the settable work offset (G54 ...).....	5-258
5.6.3	Global work offset/frame	5-259
5.6.4	Display active settable work offsets	5-262
5.6.5	Display active programmable work offsets.....	5-263
5.6.6	Display active external work offsets	5-264
5.6.7	Display the sum of the active work offsets.....	5-265
5.6.8	Activate work offset and basic frame immediately	5-265
5.6.9	Actual value display: settable work offset system, SZS	5-266
5.7	User data/user variables (GUD, PUD, LUD).....	5-267
5.7.1	General information.....	5-267
5.7.2	Changing/finding user data/user variables.....	5-268
5.8	Displaying system variables	5-270
5.8.1	Processing/creating variable views	5-271
5.8.2	Managing variable views	5-272
5.8.3	Logging system variables.....	5-273
"Program" Operating Area		6-277
6.1	Program types.....	6-279
6.1.1	Part program	6-279
6.1.2	Subroutine.....	6-279
6.1.3	Workpiece.....	6-279
6.1.4	Cycles.....	6-279
6.2	Storing programs.....	6-280
6.2.1	HMI.....	6-280
6.3	Program basic display.....	6-280

6.4	Editing programs.....	6-282
6.4.1	Text editor.....	6-282
6.4.2	Undo and redo in the editor (SW 6.3 and higher).....	6-283
6.4.3	Selective program protection: *RO*.....	6-284
6.5	Free contour programming.....	6-285
6.5.1	General.....	6-285
6.5.2	Programming a contour.....	6-286
6.5.3	Contour elements.....	6-290
6.5.4	Graphic representation of the contour.....	6-291
6.5.5	Input screen forms for parameterizing the contour elements.....	6-292
6.5.6	Specifying contour elements in polar coordinates, Closing the contour.....	6-295
6.5.7	Undercuts in turning technology.....	6-300
6.5.8	Help.....	6-303
6.5.9	Parameter description of straight line/circle contour elements.....	6-304
6.5.10	Examples of user-defined contour programming.....	6-305
6.5.11	Cycle support.....	6-308
6.6	Program simulation.....	6-309
6.6.1	Simulation user interface.....	6-310
6.6.2	Simulation settings.....	6-321
6.6.3	Downtime settings.....	6-325
6.6.4	Display and colors.....	6-326
6.6.5	Section by section simulation (SW 5.2).....	6-327
6.6.6	Simulation for orientable toolholder.....	6-328
6.7	Simulation with external network drive.....	6-329
6.8	Program management.....	6-330
6.8.1	Overview.....	6-330
6.8.2	NC file types and directories.....	6-331
6.8.3	New workpiece/parts program.....	6-332
6.8.4	Storing setup data.....	6-335
6.8.5	Selecting a program for execution.....	6-336
6.8.6	Loading/Unloading a program.....	6-339
6.8.7	Program management.....	6-340
6.8.8	Copying/inserting.....	6-341
6.8.9	Deleting.....	6-344
6.8.10	Renaming.....	6-345
6.8.11	Enabling.....	6-347
6.8.12	Log.....	6-348
6.9	Accessing an external network drive/computer.....	6-349
	"Services" Operating Area	7-351
7.1	Function.....	7-352

7.2	Directory structure.....	7-352
7.2.1	NC active data.....	7-352
7.2.2	Hard disk.....	7-353
7.2.3	Directories.....	7-356
7.2.4	Data selection.....	7-357
7.3	Formats for saving and importing data.....	7-360
7.3.1	Punched tape format.....	7-361
7.3.2	PC format.....	7-365
7.4	RS-232 interface parameters.....	7-366
7.4.1	Interface parameters.....	7-369
7.5	Operator interface.....	7-371
7.5.1	Services basic display.....	7-371
7.5.2	Setting the RS-232 interface.....	7-374
7.5.3	Reading in data.....	7-379
7.5.4	Reading out data.....	7-381
7.5.5	Log.....	7-382
7.5.6	Importing/exporting ISO programs.....	7-384
7.6	Manage data.....	7-389
7.6.1	Standardized program and data management (SW 6.3 and higher).....	7-389
7.6.2	Creating a new file/directory.....	7-390
7.6.3	Loading/unloading.....	7-390
7.6.4	Copying/inserting.....	7-391
7.6.5	Deleting.....	7-393
7.6.6	Changing the properties of a file/drive/archive.....	7-394
7.6.7	Defining and activating user data (GUD).....	7-396
7.7	Start-up functions.....	7-398
7.7.1	Series start-up.....	7-398
7.7.2	Restoring the original state via NC card.....	7-401
7.7.3	Software update.....	7-402
	"Diagnosis" Operating Area	8-405
8.1	Diagnosis basic display.....	8-406
8.2	Alarms/messages/alarm log.....	8-408
8.3	Service display.....	8-410
8.3.1	Service overview (SW 6 and higher).....	8-410
8.3.2	Service axis.....	8-412
8.3.3	Service drive.....	8-413
8.3.4	Service Safety Integrated.....	8-414
8.3.5	Configuration data.....	8-418
8.3.6	Communication error log.....	8-419

8.3.7	Action log	8-419
8.3.8	Version.....	8-420
8.3.9	Displaying the version screen for cycles (SW 6.3 and higher)	8-420
8.3.10	Exporting cycle versions (SW 6.3 and higher).....	8-423
8.3.11	Displaying loadable compile cycles (SW 6.3 and higher)	8-423
8.4	PLC status	8-425
8.4.1	General	8-425
8.4.2	Changing/deleting values.....	8-426
8.5	Selecting/creating operand forms for PLC status	8-428
8.5.1	File functions.....	8-429
8.6	Display NC system resources.....	8-430
"Start-Up" Operating Area		9-431
9.1	Machine data	9-436
9.1.1	Display options: Masking filter	9-438
9.2	User views	9-439
9.3	NC.....	9-441
9.4	PLC.....	9-441
9.4.1	PLC status	9-442
9.4.2	Setting the date/time.....	9-442
9.4.3	File functions.....	9-442
9.5	Drives/Servo	9-443
9.6	HMI	9-443
9.6.1	Changing the HMI interface	9-443
9.6.2	System settings.....	9-446
9.7	Tool management.....	9-450
Maintenance		10-451
10.1	Operating data	10-452
10.2	Cleaning.....	10-453
Appendix		A-455
A	Abbreviations	A-455
B	Terms.....	A-463
C	References	A-483
D	Index.....	A-499

Preface



Note

This HMI Advanced Operator's Guide describes the user interfaces and the operating sequences for the main operations on systems with software version 6 or later with Sinumerik 840D and software version 4 or later with Sinumerik 810D.

The MMC description applies to software versions up to and including 5 with Sinumerik 840D and including 3 with Sinumerik 810D.

The term "MMC" still appears (with explanation) in this description as a softkey designation. However, it stands here for HMI Advanced with PCU 50.

Organization of documentation

SINUMERIK documentation is organized on 3 separate levels:

- General Documentation
- User Documentation
- Manufacturer/Service Documentation

Target group

This Manual is intended for machine users. This publication provides detailed information that the user requires to operate the SINUMERIK 840D or SINUMERIK 810D

Standard functions

This Operator's Guide describes the functionality afforded by standard functions. Modifications and additions implemented by the machine manufacturer are documented by the machine manufacturer.

More detailed information about other publications relating to SINUMERIK 840D, 810D or SINUMERIK FM-NC and publications that apply to all SINUMERIK controls (e.g. Universal Interface, Measuring Cycles...) can be obtained from your local Siemens branch office.

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

Validity

Catalog NC 60 is the definitive document as regards the validity of functions

/BU/ Ordering Information, Catalog NC 60.

Export version

The following functions are not available in the export version:

Function	810DE	840DE
5-axis machining package	–	–
Handling transformation package (5 axes)	–	–
Multi-axis interpolation (> 4 axes)	–	–
Helical interpolation 2D+6	–	–
Synchronized actions, stage 2	–	0 ¹⁾
Measurements, stage 2	–	0 ¹⁾
Adaptive control	0 ¹⁾	0 ¹⁾
Continuous dressing	0 ¹⁾	0 ¹⁾
Utilization of compile cycles (OEM)	–	–
Sag compensation, multi-dimensional	–	0 ¹⁾

– Function not available

1) Restricted functionality

Structure of descriptions

All functions and operating options have been described according to the same internal structure as far as this is suitable and practicable. The various levels of information have been structured so that you can find the information you are looking for quickly.

1. Function

This theoretical section is primarily intended as learning material for the NC beginner. It provides important information that will help you to understand the principle of operating functions.

You should work through the manual at least once to get an idea of the scope of the functions and capability of your SINUMERIK control.

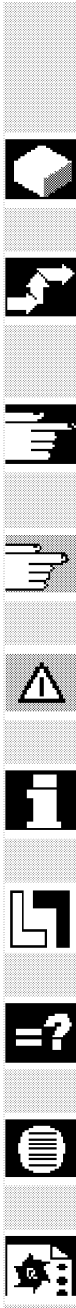
2. Sequence of operations

This section provides a clear diagrammatic description of the sequence of key inputs required. If entries have to be made at individual stages of the sequence or if you require additional information, you will find this next to the key illustrations.

3. Additional notes

For safety reasons, some of the functions are protected from access by unauthorized persons. The machine manufacturer can influence or modify the described functions. Please follow the instructions of the machine manufacturer.





Explanation of symbols

Function

Sequence of operations

Additional notes

Cross-references to other documentation or sections

Danger notes

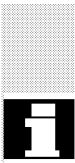
Additional notes or background information

Ordering data option

Explanation

Description of syntax

Programming examples

**Notes**

The following special symbols and keywords have been used in this documentation:

This symbol appears in this documentation whenever it is necessary to draw your attention to an important item of information.



In this document, you will find the symbol depicted with a reference to an ordering code. Please note that the function described can operate only if the specified option is installed in the control.

Warnings

The following warnings with varying degrees of severity appear in this document.

**Danger**

Indicates an imminently hazardous situation which, if not avoided, **will** result in death or serious injury or in substantial property damage.

**Warning**

Indicates a potentially hazardous situation which, if not avoided, **could** result in death or serious injury or in substantial property damage.

**Caution**

Used with the safety alert symbol indicates a potentially hazardous situation which, if not avoided, **may** result in minor or moderate injury or in property damage.

Caution

Used without safety alert symbol indicates a potentially hazardous situation which, if not avoided, **may** result in property damage.

Notice

Used without the safety alert symbol indicates a potential situation which, if not avoided, **may** result in an undesirable result or state.

**References**

This symbol appears whenever specific information can be found in other literature.

A complete list of available literature is included in the Appendix of this Operator's Guide.

Principle

Your SIEMENS 840D or 810D has been designed and constructed according to state-of-the-art technology and approved safety regulations and standards.

Additional equipment

The applications of SIEMENS controls can be expanded by adding special additional devices, equipment and expansions supplied by SIEMENS.

Personnel

Only **appropriately trained, authorized and reliable personnel** may be allowed to operate this equipment. No-one without the necessary training must be allowed to operate the control, even temporarily.

The **responsibilities** of the personnel employed to set up, operate and maintain the equipment must be clearly **defined** and **supervised**.

Procedure

Before the control is started up, the personnel who will work on the control system must become thoroughly acquainted with the Operator's Guide. It is also the duty of the equipment operator to **constantly monitor** the overall technical condition of the control (outwardly apparent defects or damage as well as changes in operating performance).

Service

Repairs must be carried out by personnel who are **specially trained and qualified** in the relevant technical subject according to the information supplied in the service and maintenance guide. All relevant safety regulations must be followed.

The following is deemed to be **improper usage** and **exempts the manufacturer from any liability**:

Any application deviating from the above points or usage extending beyond the given limits.

If the control is **not in a technically perfect condition** or is not operated under the necessary observance of safety and accident prevention regulations and according to instructions given in the relevant documentation.

If faults that might affect the safety of the equipment are not rectified **before** the control is started up.

Any **modification, bypassing or disabling** of items of equipment on the control that are required to ensure fault-free operation, unlimited use and active and passive safety.

Improper usage gives rise to **unforeseen danger** to:

- Life and limb of personnel,
- The control, machine or other assets of the owner and the user.



Introduction

1.1	The product SINUMERIK 840D/810D	1-22
1.2	Handling instructions	1-24
1.3	Switching the control ON and OFF	1-25

1.1 The product SINUMERIK 840D/810D

General information

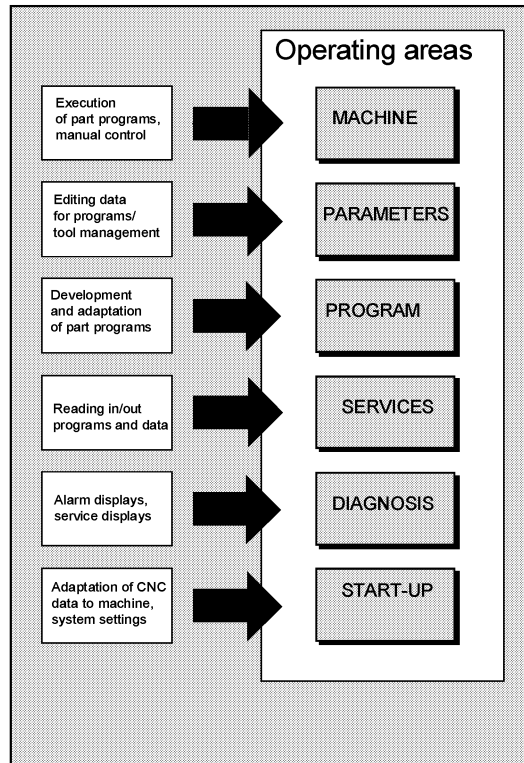
The SINUMERIK 840D, 810D is a CNC control (Computerized Numerical Control) for machine tools.

You can implement the following basic functions (for a machine tool) via the operator panel front of the CNC control:

- Development and adaptation of parts programs,
- Execution of parts programs,
- Manual control,
- Reading in and out parts programs and data,
- Editing of data for programs,
- Display alarms and eliminate specifically,
- Editing of machine data,
- Establishment of communication links between 1 or more control units (m) or 1 or more NCs (n)
(m:n, m control units and n NCK/PLC units).

Operating areas

Basic functions are grouped to form the following operating areas in the control (on gray background):



The user can call up all the functions via the user interface.

The user interface consists of:

- Display units, such as monitor, LEDs etc.
- Operator elements such as keys, switches, handwheels etc.

Read Chapter 2 "Operator Components" carefully before proceeding with further chapters.

All subsequent chapters are written on the assumption that you have done so!



1.2 Handling instructions**Caution**

The operator panel front/machine control panel may only be opened by trained personnel for servicing purposes.

**Danger**

Never open the operator panel front/machine control panel unless the power supply has been disconnected! Failure to comply could result in fatal injury!

**Warning**

Electronic components inside the operator panel/machine control panel might be destroyed by electrostatic discharge if they are handled incorrectly.



Before operating any of the control elements on this operator panel front:

Please first read the explanations supplied in this documentation!

1.3 Switching the control ON and OFF



Function

Switching the control ON

The control and the entire system can be switched on in different ways.



Machine manufacturer

Please follow the machine manufacturer's instructions!

After the control has been switched on, the "Reference point approach" display or another basic display programmed by the machine manufacturer will appear.

Machine		Jog	
Channel Reset			
Program aborted			
MCS	Position	Master spindle S1	
-X	0.000 mm	Act. +	0.000 rev./min
+Y	0.000 mm	Set	0.000 rev./min
+Z	0.000 mm	Pos.	0.000 deg.
+	0.000 mm	Power [%]	<input type="text"/>
		Feedrate mm/min	REF
		Act. 0.000	0.000 %
		Set 0.000	
		Tool	
		preselected tool :	
		G0	G91

Switching the control OFF

Please follow the instructions for switching off the control or the entire system!



Machine manufacturer

Please follow the machine manufacturer's instructions!



Sequence of operations

When you press the "Area switchover" key, operating areas are displayed on the horizontal softkey bar and operating modes are displayed on the vertical softkey bar. You can use this key to go to the area menu bar from any location in the menu hierarchy if you wish to select another operating mode or a different operating area.

Machine	Jog				
Channel Reset	Program aborted		Auto		
			MDI		
MCS	Position	Auxiliary Functions			
+ X	900.000 mm	MO			
- Y	-156.000 mm	MO			
+ Z	230.000 mm	MO			
		H0.000000			
		H0.000000			
		H0.000000			
		Feedrate mm/min			
		Act. 3000.000 0.0 %			
		Set. 3000.000			
		Tool			
		▶ T0 D0 ◀			
		▶ T0 D0 ◀			
		▶ T0 D0 ◀			
		G1			
Machine	Parameters	Program	Services	Diagnosis	Start-up



By pressing the "Area switchover" key twice, you can toggle between the operating areas last selected, e.g. between the "Parameters" and "Machine" areas.



Operator Components/Operating Sequences

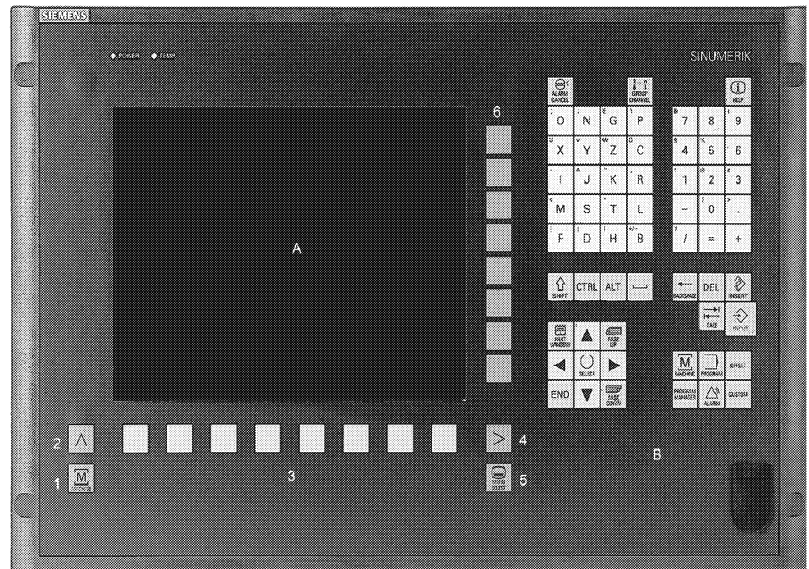
2.1	Operator panel fronts	2-29
2.1.1	Operator panel front OP 010	2-29
2.1.2	Operator panel front OP 010S	2-30
2.1.3	Operator panel front OP 010C	2-30
2.1.4	Operator panel front OP 012	2-31
2.1.5	Operator panel front OP 015	2-31
2.1.6	Full standard keyboard	2-32
2.2	Operator panel front keys	2-34
2.3	Machine control panels	2-39
2.4	Keys of the machine control panel	2-40
2.4.1	EMERGENCY STOP	2-40
2.4.2	Operating modes and machine functions	2-40
2.4.3	Feed control	2-42
2.4.4	Spindle control	2-44
2.4.5	Keyswitch	2-45
2.4.6	Program control	2-47
2.5	Screen layout	2-49
2.5.1	Overview	2-49
2.5.2	Global machine status display	2-50
2.5.3	Program control display	2-55
2.6	General operating sequences	2-57
2.6.1	Program overview and program selection	2-57
2.6.2	Changing the menu window	2-58
2.6.3	Selecting a directory/file	2-59
2.6.4	Editing entries/values	2-60
2.6.5	Confirming/canceling an input	2-62
2.6.6	Editing a parts program in the ASCII editor	2-62
2.6.7	Switch channel over	2-69
2.6.8	m:n communication links	2-70
2.6.9	Pocket calculator function	2-74
2.7	Help	2-75
2.7.1	Editor Help	2-77
2.7.2	Short help for program commands	2-78
2.7.3	Extended help for programming commands	2-82
2.8	Job list	2-83
2.8.1	Description of syntax for job lists	2-85
2.8.2	Example of a job list for two-channel 1:1 links	2-89
2.8.3	Example of a job list for multi-channel m:n links	2-90

2.8.4	Sequence of operations "Executing job list".....	2-91
2.8.5	Rename workpieces with job lists	2-93
2.8.6	Copy workpieces with job lists	2-94
2.8.7	Archiving workpieces with job lists with m:n	2-94

2.1 Operator panel fronts

2.1.1 Operator panel front OP 010

- A Display
 - B Alphanumeric block
Override/cursor keys
- 1 Machine area key
 - 2 Recall (Return)
 - 3 Softkey bar (horizontal)
 - 4 Etc. key
(menu expansion)
 - 5 Area switchover key
 - 6 Softkey bar (vertical)



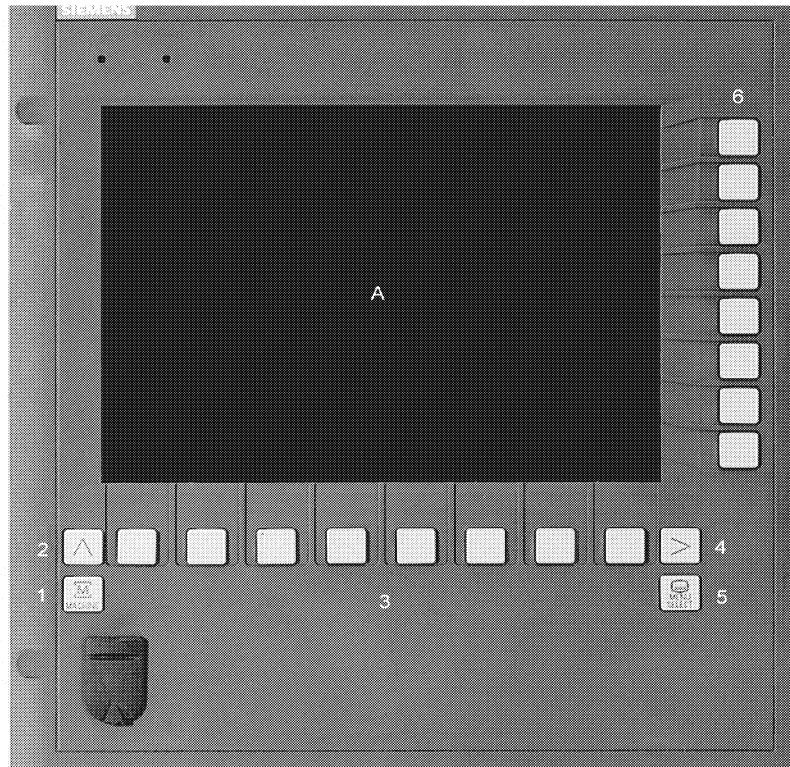
All keys are described in the following sections.

2.1 Operator panel fronts

2.1.2 Operator panel front OP 010S

A Display

- 1 Machine area key
- 2 Recall (Return)
- 3 Softkey bar (horizontal)
- 4 Etc. key
(menu expansion)
- 5 Area switchover key
- 6 Softkey bar (vertical)

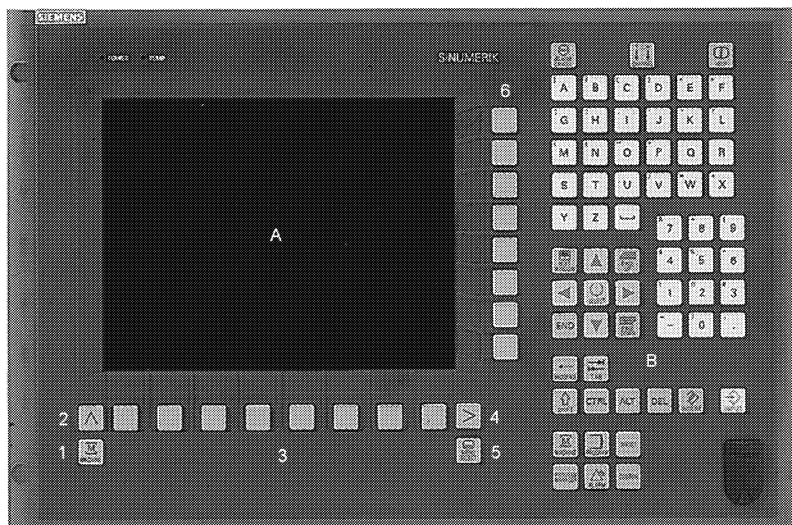


2.1.3 Operator panel front OP 010C

A Display

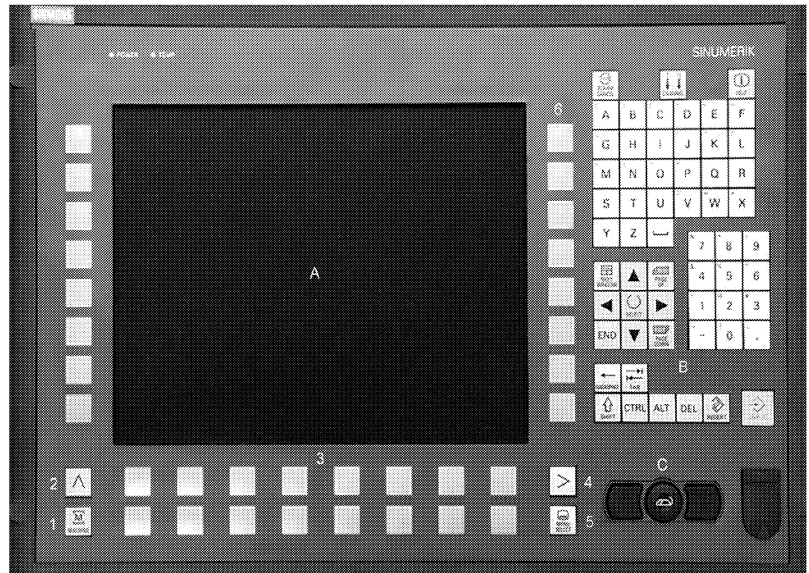
B Alphanumeric block
Override/cursor keys

- 1 Machine area key
- 2 Recall (Return)
- 3 Softkey bar (horizontal)
- 4 Etc. key
(menu expansion)
- 5 Area switchover key
- 6 Softkey bar (vertical)



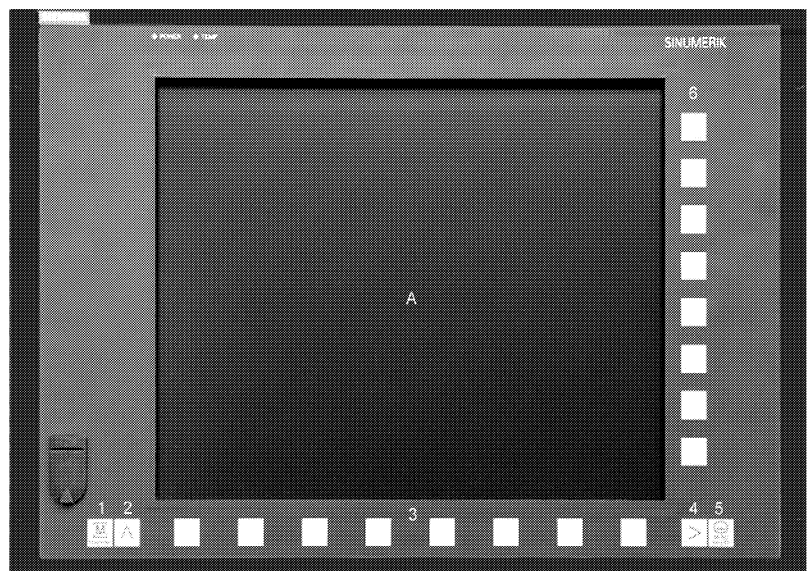
2.1.4 Operator panel front OP 012

- A** Display
B Alphanumeric block
 Override/cursor keys
C Mouse and mouse buttons
- 1 Machine area key
 2 Recall (Return)
 3 Softkey bar (horizontal)
 4 Etc. key
 (menu expansion)
 5 Area switchover key
 6 Softkey bar (vertical)



2.1.5 Operator panel front OP 015

- A** Display
- 1 Machine area key
 2 Recall (Return)
 3 Softkey bar (horizontal)
 4 Etc. key
 (menu expansion)
 5 Area switchover key
 6 Softkey bar (vertical)



For further information about operator components, please see:
 /BH/ Operator Components Manual.



2.1.6 Full standard keyboard

A full standard keyboard can be connected. However, a machine control panel is required additionally.

The special function keys of the operator keyboard can also be used with the full PC keyboard. The following table shows how the horizontal and vertical softkeys and the special keys for the operator panels are mapped onto the PC keyboard keys:

Softkey assignment

Full keyboard	F1	F2	F3	F4	F5	F6	F7	F8	F9	F10	F11	F12
with SHIFT	vertic soft. 1	vertic soft. 2	vertic soft. 3	vertic soft. 4	vertic soft. 5	vertic soft. 6	vertic soft. 7	vertic soft. 8	>	M MACHINE		
without SHIFT	horiz soft. 1	horiz soft. 2	horiz soft. 3	horiz soft. 4	horiz soft. 5	horiz soft. 6	horiz soft. 7	horiz soft. 8	△	MENU SELECT	CHANNEL	HELP
Full keyboard	5	Esc	Insert	Home	Page Up	Page Down	Enter	End				
without SHIFT	SELECT	ALARM CANCEL	INSERT	NEXT WINDOW	PAGE UP	PAGE DOWN	INPUT	END				

The following table shows how the hardkeys of the MF II keyboard on the PC are mapped onto the keys for the operator panels:

Hardkey assignment

Hardkey	Hard key 1	Hard key 2	Hard key 3	Hard key 4	Hard key 5	Hard key 6	Hard key 7	Hard key 8				
MFII with SHIFT	F11					F12	F10					
MFII without SHIFT								F10				
MFII-Num Block		END	Page Down	Home	Page Up							
operator keyboard	M-Position	PROGRAM	OFFSET	PROGRAM MANAGER	ALARM	CUSTOM	M MACHINE	MENU SELECT				

Caution

The full standard keyboard does not meet the requirements (EMC) of a SINUMERIK control. For this reason it can be used only for installation and service purposes.

For further information about the configuration of operating keys, please see: /IAM/, IM4 Installation and Start-Up, Chapter 5 Functions/Parameters.





Additional notes

Since the English version of Windows is used in the control, the keyboard language is English. A different keyboard language cannot be set.

The following keys, assigned in the hotkey pad on operator panels **OP 010** and **OP 010C**, are located on the MF II keyboard in the alpha/numeric keypad with NumLock off:

- END PROGRAM
- PAGE DOWN OFFSET
- HOME PROGRAM MANAGER
- PAGE UP ALARM

Hardkeys on the MF II keyboard

NumLock off
END



Jump directly to Program operating area.

NumLock off
PAGE DOWN



In the Parameter operating area, call up the last tool offsets directly.

NumLock off
HOME



Jump directly to the program overview of the last selected program manager.

NumLock off
PAGE UP



In the Diagnosis operating area, call up the last alarms, messages, service displays or PLC status.

SHIFT F12



Can be configured by the user.

SHIFT F10



Jump directly to the Machine operating area.

F10



Jump directly to the last selected operating area: Machine, Parameter, Program, Services, Diagnosis or Start-Up

2.2 Operator panel front keys

The elements of the operator panel keyboard and the symbols used to represent them in this manual are shown and explained below.

The keys marked with an * correspond to the key symbols in US layout.



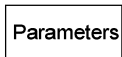
Softkeys

Keys to which functions are assigned by means of a menu bar displayed on the screen.

- It is possible to access further menu levels via the horizontal softkeys in any operating area. Each horizontal menu item has a vertical menu bar/softkey assignment.
- The vertical softkeys are assigned functions for the currently selected horizontal softkey.

A function is called up by pressing one of the vertical softkeys.

The assignments of the vertical softkey bar can change if further subsidiary functions are classified under a function.



Softkey (horizontal or vertical):

This key symbol indicates that you must have selected an operating area or a menu item or have already performed certain functions before you are able to execute the function described in the relevant section.



Machine area key

Direct branch to the "Machine" operating area.



Recall key

Return to the next higher menu. Recall closes a window.



Etc. key

Expansion of the softkey bar in the same menu.



Area switchover key

You can call the basic menu from any operating area by pressing this key. Pressing the key twice in succession changes from the current operating area to the previous one and back again.

The standard basic menu branches into the following operating areas:

1. Machine
2. Parameters
3. Program
4. Services
5. Diagnosis
6. Start-up



Shift key

Switches between functions on keys with double assignment.



Switch over channel

If several channels are in use, it is possible to switch over channels (from channel 1 to channel n).

When a "Channel menu" is configured, all existing communication links to other NCUs plus the associated channels are displayed on softkeys.

(See also Section "Switch over channel")



Alarm acknowledgment key

By pressing this key, you can acknowledge the alarm marked by this Cancel symbol.



Information key

By pressing this key, you can call up explanations and information on the current operating status of the machine (e.g. help for programming, diagnosis, PLC, alarms).

The letter "i" displayed in the dialog line indicates that information is available.



Window selection key

If several windows are displayed on the screen, it is possible to make the next window the active one using the window selection key (the active window has a thicker border). Keyboard input e.g. the page keys, is possible only in the active window.



Cursor up



Page up

You "page" up by one display.



In a parts program, you can "page" the display up (to end of program) or **down (see below)** (to start of program).

With the page keys you scroll the visible/displayed area of the window that is active. The scroll bar indicates which part of the program/document/... is selected.



Delete key (backspace)

Delete characters from right



Blank



Cursor LEFT



Selection key toggle key

- Selection key for values entered in input fields and selection lists that are marked by this key symbol.
- Activate or deactivate a field:

= active

= active

= not active

= not active

Multiple selection button

(you can select several options or none)

Single selection button/option

(only one option can be active at a time)



Cursor RIGHT



Edit key/Undo key

- Switchover to Edit mode in tables and input fields (in this case, Insert mode is set in the input field) or
- UNDO function for table elements and input fields (when you exit a field with the edit key, the value is not saved and the field is reset to the previous value = UNDO).



End of line key

- By pressing this key, the cursor in the editor is moved to the end of the line on the open page.
- Rapid positioning of the cursor on a group of related input fields.
- Has same effect as Tab key.



Cursor DOWN



Page up

You "page" up by one display. With the page keys you scroll the visible/displayed area of the window that is active. The scroll bar indicates which part of the program/document/... is selected.



Delete key

The setting in a parameterization field is deleted.
The field remains blank.



Input key

- Accepts an edited value
- Opens/closes a directory
- Opens file



Tab key



Ctrl key



Alt key



Program	See "PROGRAM" hardkey
Tool Offset	Jump directly to: Tool offset
Program manager	Program overview A program can be opened in the text editor.
Alarm	Jump directly to the alarm display
User key	Configured by user

Notes

The keys marked with an * also have a function in conjunction with ShopMill/ShopTurn.

SW 6.3 and higher



"PROGRAM" hardkey

Press this hardkey in any operating area to open and display the parts program or file last edited in the Program operating area:

- If you are already in the Program operating area and the editor is open, the program you last edited is displayed.
- If you are in any other operating area, and the editor is open, you jump back to the editor in the Program operating area and the most recent editor status is displayed.

If the editor is **not open**:

- If you are in another user application, you jump back to the Program operating area and the editor is opened with the program you last edited.

For this function to work, at least one recently edited program must be found with sufficient read rights. The program must not be already open in either a simulation or any other application. Actions such as load, copy, select, etc. must not be in progress and the parts program must not be running on the NC.

In the above cases, the operation is denied with alarms 1203xx.

2.3 Machine control panels

Standard turning machines/milling machines

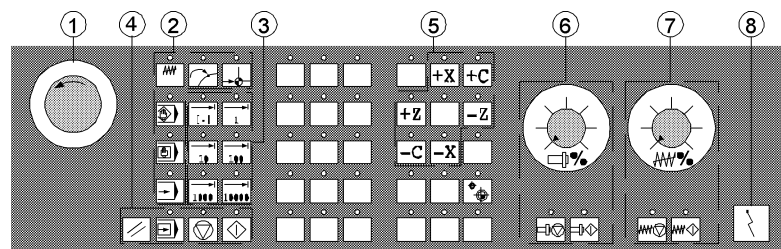
Actions on the machine tool, for example traversing the axes or program start, can only be initiated via a machine control panel.

The machine tool can either be equipped with a standard machine control panel from SIEMENS (ordering data option) or with a specific machine control panel from the machine-tool manufacturer.

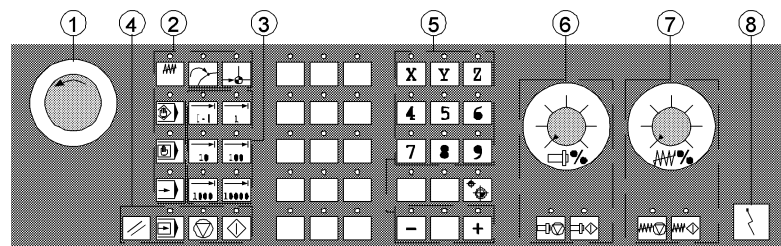
The following description applies to the 19" machine control panel supplied by SIEMENS. If you are using another machine control panel, please consult the operating instructions of the machine-tool manufacturer.

The standard machine control panel from SIEMENS is equipped with the following operator controls:

- 1 EMERGENCY STOP button
- 2 Operating modes (with machine functions)
- 3 JOG/Increment keys
- 4 Program control
- 5 Direction key with rapid traverse override
- 6 Spindle control
- 7 Feed control
- 8 Keyswitch



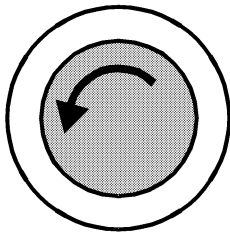
Machine control panel for turning machines



Machine control panel for milling machines

2.4 Keys of the machine control panel

2.4.1 EMERGENCY STOP



Emergency stop key

Press the red button in emergency situations, i.e.

1. if human life is in danger.
2. if there is a risk of damage to the machine or workpiece.

As a rule, emergency stop causes all drives to be stopped under control with the greatest possible braking torque.

For other responses when the emergency stop switch is operated: Please consult the instructions of the machine-tool manufacturer!



2.4.2 Operating modes and machine functions

The keys marked with an * correspond to the key symbols in US layout.

If you press a "Mode key", the corresponding mode is selected if permissible, and all other modes and functions are deselected.

The active mode is signaled and confirmed by the associated LED which lights up.



Jog

Jogging

Axis jog mode implemented by:

- continuous motion of the axes using the direction keys or
- incremental motion of the axes using the direction keys or
- the handwheel.



MDI

Manual Data Input

Control of machine through execution of a block or a sequence of blocks. The blocks are entered on the operator panel front.



Automatic

Control of machine through automatic execution of programs.

Inc keys



You can activate the Inc functions in conjunction with the following modes:

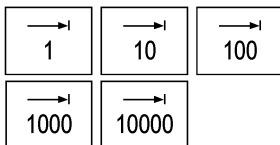
- "Jog" mode
- "MDI/Teach In" mode



Inc Var

Incremental feed variable

Incremental traverse with variable increment size (see "Parameters" operating area, setting data).



Inc

Incremental feed

Incremental traverse with preset increment size of 1, 10, 100, 1000, 10000 increments.



The way that the incremental value is evaluated depends on the setting in the machine data.

Machine functions



Teach In

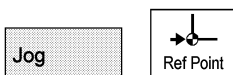
Creation of programs in interactive mode with the machine in "MDI" mode.



REPOS

Reposition

Reposition, re-approach contour in "Jog" mode.

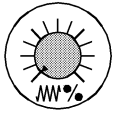


Ref point

Reference point approach

Approach the reference point (Ref) in "Jog" mode.

2.4.3 Feed control



Feedrate rapid traverse override (feedrate override switch)

Control range:

0% to 120% of programmed feedrate.

In rapid traverse, the 100% value is not exceeded.

Settings:

0%, 1%, 2%, 4%, 6%, 8%, 10%, 20%, 30%, 40%, 50%, 60%, 70%, 75%, 80%, 85%, 90%, 95%, 100%, 105%, 110%, 115%, 120%



Feed stop

If you press the "Feed stop" key:

- execution of the current program is stopped,
- the axis drives are stopped in a controlled manner,
- the associated LED lights up as soon as feed stop has been accepted by the control and
- The following appears in the header (program control display):
FST (= Feed Stop)

Example:

- An error is detected while a block is being processed in "MDI" mode.
- The tool must be changed.

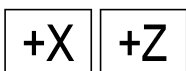


Feed start

If you press the "Feed start" key:

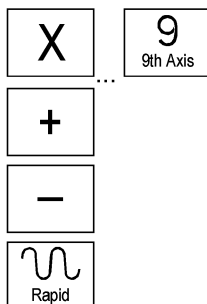
- the parts program is continued at the current block position,
- the feedrate is accelerated to the value defined in the program and
- the associated LED lights up as soon as feed start has been accepted by the control.

Axis keys (for turning machines):



Press these keys to traverse the selected axis (X...Z) in a positive direction.

Press these keys to traverse the selected axis (X...Z) in a negative direction.



Machine manufacturer

Axis keys (for milling machines):

You select the axis (X...9) to be traversed,

press the "+" key to traverse in the positive direction or

press the "-" key to traverse in the negative direction.

Rapid traverse override

If you press this key together with key "+" or "-", the axis moves in rapid traverse mode.

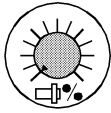
- The specified increments and control range apply to standard machines.
- Increments and control range can be modified by the machine tool manufacturer to suit specific applications.
- Feedrate/rapid traverse feedrate and the values for the feedrate override positions (if the feedrate override switch is also active for rapid traverse) are defined in the machine data (see the information supplied by the machine manufacturer).



MCS/WCS

You can switch between the machine and workpiece coordinate systems in the Machine operating area using softkeys MCS/WCS or the corresponding key on the machine control panel.

2.4.4 Spindle control



Spindle override (spindle speed override switch)

- The rotary switch with latch positions allows you to increase or decrease the programmed spindle speed "S" (equivalent to 100%).
- The set spindle speed value "S" is output as an absolute value and a percentage in the "Spindles" display (vertical softkey in basic display).

Control range:

50% to 120% of programmed spindle speed

Increment:

5% between latch positions



Spindle stop

When you press the "Spindle stop" key:

- the spindle is decelerated down to zero speed and
- the associated LED lights up as soon as "Spindle stop" is accepted by the control.

Example:

- to change a tool.
- to enter S, T, H, M functions during setup.



Spindle start

When you press the "Spindle start" key:

- the spindle speed is accelerated to the value defined in the program and
- the associated LED lights up as soon as "Spindle start" has been accepted by the control.



Machine manufacturer

- The specified increment and the control range apply to standard machine data (MD). These MD can be changed by the machine manufacturer to suit the application.
- The maximum spindle speed and the values for the spindle speed override position are defined in the machine data and setting data (see information supplied by the machine manufacturer).

2.4.5 Keyswitch

SIEMENS keyswitch

The keyswitch on the SINUMERIK 840D, 810D has 4 positions to which protection levels 4 to 7 are assigned.



Machine manufacturer

Functions can be assigned to keyswitch positions by the machine manufacturer. Using machine data it is also possible to set access to programs, data and functions to suit the user's requirements.

The keyswitch has three different colored keys which can be removed in the specified positions:

Switch positions



Position 0
No key
Protection level 7



Position 1
Key 1 **black**
Protection level 6



Position 2
Key 1 **green**
Protection level 5



Position 3
Key 1 **red**
Protection level 4

Lowest
access rights



Highest
access rights



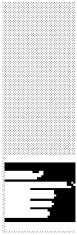
Changing access rights

The screen is not automatically updated after a change in access authorization (e.g. when the keyswitch position is changed), but only when the screen is next refreshed (e.g. on closing and opening a directory).

The currently valid access authorization is checked every time a function is executed.

If the PLC is in the stop state, the input image of the machine control panel is not scanned. For this reason the keyswitch positions are not evaluated during start-up.

2.4 Keys of the machine control panel



Passwords

As an additional option for setting access authorization, it is possible to enter three passwords in the "Start-Up" operating area.

If the password is set, the keyswitch positions are irrelevant.

/IAD/, Installation and Start-Up Guide 840D or

/IAC/, Installation and Start-Up Guide 810D

2.4.6 Program control



Cycle Start

NC Start

If you press the "NC Start" key, the selected parts program (parts program name is displayed in header) is started at the current block and the associated LED lights up.



Cycle Stop

NC Stop

If you press the "NC Stop" key, processing of the active parts program is halted and the associated LED lights up.

After this, you can continue processing with NC start.



Single Block

Single block

This function allows you to execute a parts program block by block. You can activate the "Single block" function in "Automatic" and "MDI" modes. If single block is activated, the associated LED on the machine control panel lights up.

If single-block processing is active

- Halt in Cycle is displayed on the screen (in the program control display),
- the text "Stop: Block ended in single block" is output in the channel operational message line (when program is interrupted),
- the current block of the parts program is not processed until you press the "NC Start" key,
- processing is stopped after execution of one block,
- the following block can be executed by pressing the "NC Start" key again.

You can deselect the function by pressing the "Single block" key again.

This function is dependent on the settings under "Program control" in the Machine operating area.





Reset

When you press the "Reset" key:

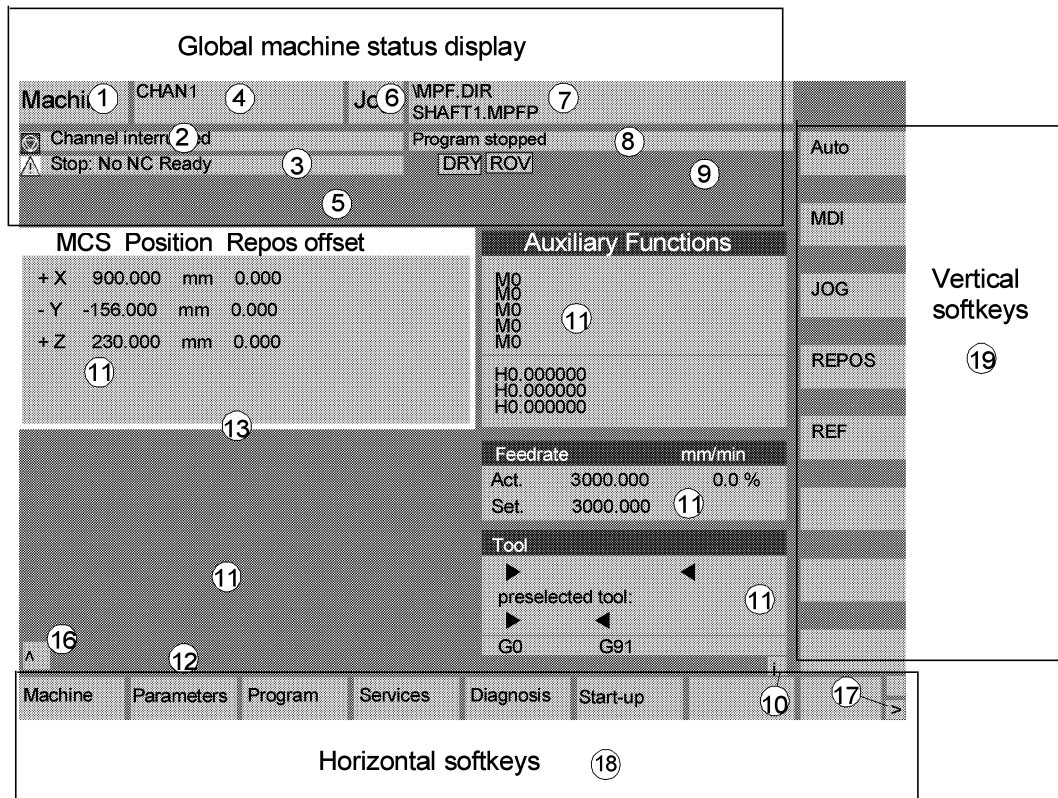
- processing of the current parts program is aborted,
- signals from the monitoring function are cleared (except for alarms signaling POWER ON, NC Start and Acknowledge alarm),
- the channel is switched to the "Reset" state, i.e.
 - the NC control remains synchronized with the machine,
 - the control is in the initial state and ready for another program run.

(See also /FB/, K1, Description of Functions Mode Group, Channel, Program Operation Mode)

2.5 Screen layout

In HMI Advanced SW 6.2 or later the states of the control are clearly indicated by additional indicators and icons.

2.5.1 Overview



- 1 Operating areas
- 2 Channel status
- 3 Channel operational messages
- 4 Channel name
- 5 Alarm and message line
- 6 Operating mode, submode (increment, if relevant)
- 7 Program name of selected program
- 8 Program status
- 9 Program control
- 10 Additional explanatory text (Help) can be called
 - i Information can be displayed by means of the i key
 - ^ Recall: Return to higher-level menu
 - > Etc.: Extension of horizontal softkey bar within the same menu

11 Working windows, NC displays

The working windows (program editor) and NC displays (feedrate, tool) available in the selected operating area are displayed here.

SW 6.2 or later

Positional data in working windows displays the diameter symbol \varnothing before the unit if the current axis is the transverse axis and the workpiece coordinate system is set. If the diameter programming is reset with DIAMOF, the symbol before the unit is removed.

12 Dialog line with operator prompts

Operator prompts (if available) relating to the selected function are displayed here.

13 Focus

The selected window is clearly identified by a special border. The window header display is inverted. Data entered on the operator panel front apply to this window.

16 Recall function, i.e. ^ key is active**17 ETC. function, i.e. > key is active****18 Horizontal softkeys****19 Vertical softkeys**

The softkey functions available in the selected operating area are displayed in the horizontal and vertical softkey menus (corresponding to F1 to F8 on the full keyboard).

Additional notes

The screen layout may differ slightly from the layout displayed above depending on the screen size or resolution used.

2.5.2 Global machine status display**1 Operating areas**

The currently selected operating area is displayed (Machine, Parameters, Program, Services, Diagnosis, Start-Up).

2 Channel status


The current channel status is displayed,



- Channel reset
- Channel interrupted
- Channel active

3 Channel operational messages


SW 6.2 or later: Displays the channel operational messages with symbols.

See Subsection 4.1.2

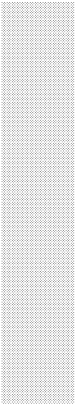
Action is required for states with a  icon.


-  1 Stop: No NC ready
- 2 Stop: Mode group ready
- 3 Stop: Emergency Stop active
- 4 Stop: Alarm with stop active
- 5 Stop: M0/M1 active
- 6 Stop: Block ended in SBL mode
-  7 Stop: Cycle Stop active
- 8 Wait: Read-in enable missing
- 9 Wait: Feedrate enable missing
- 12 Wait: Axis enable missing
- 17 Wait: Feedrate override > 0%
- 18 Stop: Error in NC block
- 19 Wait: for NC blocks from external
- 22 Wait: No spindle enable
- 23 Wait: Axis feedrate value is 0
- 31 Stop: No channel ready
- 45 Stop: SERUPRO found the search item and the NCK stopped. SERUPRO is the abbreviation for SEArchRUn by PROgram testing, which is a new type of search mode.

Action is not usually required for states with a  icon.

-  10 Wait: Dwell active
- 11 Wait: Aux. funct. ackn. missing
- 13 Wait: Exact stop not reached
- 14 Wait: for positioning axis

2.5 Screen layout

- 
- 15 Wait: for spindle
 - 16 Wait: for other channel
 - 20 Wait: due to SYNACT instruction
 - 21 Wait: Block search active
 - 24 Wait: for tool change acknowledgement
 - 25 Wait: for gear stage change
 - 26 Wait: for position control
 - 27 Wait: for thread cut

-  29 Wait: for punching
- 30 Wait: for safe operation
- 32 Stop: Reciprocation active
- 33 Stop: Axis replacement active (block change inhibited by initiation of an axis replacement)
- 34 Wait: for axis container rotation
- 35 Wait: AXCT axis active as following axis
- 36 Wait: AXCT axis active as leading axis
- 37 Wait: AXCT axis changes to correction
- 38 Wait: AXCT axis internal state change
- 39 Wait: AXCT axis drive disable
- 40 Wait: AXCT axis overlaid movement active
- 41 Wait: AXCT axis axis replacement active
- 42 Wait: AXCT axis interpolator active
- 43 Wait: WAIT_FOR_CC_ENABLE: Wait for compile cycle
- 44 Wait: on access to system variable
- 46 Stop: ESR triggered
- 47 Wait: Axis container rotation waiting for spindle stop
- 48 Wait: Axis container rotation waiting for MD match data (New Config)
- 49 Wait: for axis replacement: Axis currently linking
- 50 Wait: for axis replacement: Liftfast active
- 51 Wait: for axis replacement: New Config active
- 52 Wait: for axis replacement: Axis container rotation active
- 53 Wait: for axis replacement: Waitp active
- 54 Wait: for axis replacement: Axis is currently in another channel
- 55 Wait: for axis replacement: Axis is currently PLC axis
- 56 Wait: for axis replacement: Axis is currently reciprocating axis
- 57 Wait: for axis replacement: Axis is currently jog axis
- 58 Wait: for axis replacement: Axis is currently command axis
- 59 Wait: for axis replacement: Axis is currently OEM axis



60 Wait: for axis replacement: Axis is currently following axis coupled to master value

61 Wait: for axis replacement: Axis is currently coupled-motion following axis

62 Wait: for axis replacement: Axis is currently coupled slave axis

4 Channel name	Name of the channel in which the program is running.
5 Alarm and message line	<ul style="list-style-type: none"> - Alarms and messages or - information programmed with command MSG in the parts program (if no alarms are active)
6 Operating mode display	<p>The currently selected operating mode, i.e. Jog, MDI or AUTO (automatic) is displayed.</p> <p>SW 6.2 or later:</p> <p>The active submode is displayed next to the operating mode. An active increment is also displayed below. E.g.</p> <p>JOG REPOS 1000</p>
7 Program name	<p>Press NC Start to execute this program.</p> <p>SW 6.2 or later:</p> <p>The "Program name" output field can be configured for JOG and MDI. See machine manufacturer's specifications.</p>
8 Program status	<p>The current status of the part program being executed is output</p> <ul style="list-style-type: none"> - Program aborted - Program running - Program stopped <p>SW 6.2 or later:</p> <p>The "program status" output field can be configured, e.g. with the "Channel-wide status display with symbols" function (Subsection 4.1.3). See machine manufacturer's specifications.</p>
9 Program control display	Functions that have been activated are visible (can be set via "Program control"), see Subsection 4.6.12

2.5.3 Program control display



Function

Functions that have been activated are visible in the program control display (can be set via "Program control", see Subsection 4.6.12). The functions are displayed irrespective of the selected menu.

SKP

Skip block

Program blocks in which the block no. is preceded by a slash are skipped when you run the program (e.g. "/N100 ...").

Up to 8 program levels can be skipped (e.g. "/6N100 .."; the 6th program level is skipped).

References: /PG/, Programming Guide, Fundamentals, Chapter 2

SKPn

n = active skip levels

SBL1

Single block with stop after each machine function block

If this function is active, execution is interrupted after every block that triggers a function on the machine (calculation blocks do not cause an interruption).

SBL2

Single block with stop after each block

If this function is active, the part program blocks are executed singly as follows: Each block is decoded separately and execution is interrupted after every block.

SBL3

Stop in cycle

If this function is active, the part program blocks are executed individually in the cycle as follows:

Each block is decoded separately and execution is interrupted after every block. Parts program blocks are

- Traversing blocks
- Switching functions and auxiliary functions
- Blocks generated internally by the control (e.g.: blocks inserted by tool radius compensation)
- Thread blocks after retraction
- Thread blocks with dry run feedrate

Thread blocks without dry run feedrate are an exception to this. Here, execution is only interrupted at the end of the current thread block.

SBL2 can only be selected in the Reset state.

Either SBL1 or SBL2 can be selected!

This function can be activated only in the "Single block" state.

DRY Dry run feedrate	<p>Traverse movements are performed with the feedrate value set in the setting data "Dry run feedrate".</p> <p>This dry run feedrate function replaces the programmed motion commands.</p>
ROV Rapid traverse override	<p>The override switch for the feedrate also applies to the rapid traverse feedrate.</p>
M01 Programmed stop 1	<p>When this function is active, processing of the program is stopped at every block in which miscellaneous function M01 is programmed. The message "Stop: M00/M01 active" is then displayed on the screen. You restart the program run by pressing the NC Start key. If the function is not activated, then miscellaneous function M01 (from the part program) is ignored.</p>
Additional M fct. Programmed stop 2	<p>SW 6.3 and higher:</p> <p>When this function is active, program execution on the NC is interrupted on request by the PLC at blocks containing the additional function for a conditional stop. You can define a function associated with M01 in machine data MD 22256: AUXFO_ASSOC_M1_VALUE. The value of this miscellaneous M function number is then equivalent to a programmed stop 2.</p> <p>References: /FB1/ Description of Functions Basic Machine, Predefined Auxiliary Function "Associated auxiliary functions for M0, M1"</p>
DRF DRF selection	<p>When the "DRF" function is active, DRF offsets are applied.</p>
PRT Program test	<p>In program test mode, setpoint outputs to the axes and spindles are disabled. The setpoint display "simulates" the traverse movements.</p>
FST Feed stop	<p>The activated feed stop is displayed.</p> <p>SW 6.2 or later:</p> <p>"Feed stop" is displayed in the feed window with the "Feed not available" symbol and is not included in the program control display. This function is not activated/deactivated under Program Control, but via the Feed Start/Feed Stop keys on the machine control panel.</p>

2.6 General operating sequences



Keys

A range of keys and menus is provided in the various operating areas. The function of the keys and menus is the same in all operating areas.



This identical functionality applies only if the operating areas are left intact as supplied by Siemens AG and no modifications have been configured by the user.



Additional notes

In **SW 6.3** and higher, it is possible to customize the menu trees. Users can create and define softkeys to meet their individual needs. Due to this capability, the menus and softkeys on your control may be different to those described in this Operator's Guide.



Functions

/IAM/, IM4 Chapter 5 "Functions/Parameter settings" see User-specific menu trees"

The following section describes functions which you can select in several operating modes.

2.6.1 Program overview and program selection



Function

After selection of a workpiece or program overview, individual workpieces or programs can be enabled or disabled for execution.



Sequence of operations

"AUTO" is selected in the "Machine" operating area.
The appropriate channel is selected.
The channel is in reset state.
The workpiece/program to be selected is in the memory.

An overview of all workpiece directories/programs that exist is displayed.

Position the cursor on the desired workpiece/program.

Select the workpiece/program for execution:

Select
program

The name of the selected workpiece is displayed on the screen in field "Program name" at the top. The program is then loaded.

2.6.2 Changing the menu window

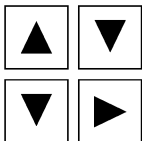


If a screen comprises several windows, you can use the "Window selection" key to switch between the individual windows. You only need to do this if you wish to enter data on the operator panel front. The focus changes to the selected menu window (the header and border of the active window appear in a different display format).



Scroll in menu window:

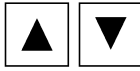
If the contents of a window cover several screen pages, you can use the "Page" keys to scroll up or down through the information. A scroll bar indicates that the contents of the window extend beyond the visible display.



Position cursor in menu window:

You can position the cursor at the desired point in the menu window with the "Direction keys".

2.6.3 Selecting a directory/file



The "Direction keys" can be used to position the cursor on the desired directory/file.

If you enter a character on the alphanumeric keyboard, the cursor moves to the first name that begins with the character.



Open/close directory:

Press the "Input" key to open or close a directory.



Open file:

You can open a file with the "Input" key if you wish to edit it in the ASCII editor. The editor is opened automatically.



Mark file

This key is used to select a file. It is possible to select multiple files.

The symbol appears next to the cursor bar when a file is selected.



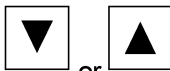
Select multiple files

In order to select a block of files, press the "Shift" and "Cursor down" keys simultaneously.



The first time you press the keys, the start of the block is selected.

The subsequent files are selected continuously



until you press "Cursor up" or "Cursor down" (without "Shift") key.



Deselects a selected file.



Cancels all selections.

2.6.4 Editing entries/values

If you wish to edit inputs/values, the corresponding key is always displayed automatically on the right of the input field. The following input fields are available:



1. Option buttons (single selection button/multiple selection button):

You can use the "Selection" key to activate or deactivate a selection field.

Multiple selection button (you can select several options or none)	Single selection button/option button (only one option can be active at a time)
--	---

= active

= active

= not active

= not active

2. Input fields:

Position the cursor on the input field and start to type. When you begin typing, you automatically switch to Insert mode.



Confirm your input with the "Input" key. The value is accepted.



To change an existing value, press the "Edit" key to switch to insert mode.

Enter the value or word (e.g. file name, etc.) on the alphanumeric keypad.



In some fields it is possible to choose between several preset values by means of the "Toggle" key.

3. Selection list

Selection lists show you a selected value from a list of possible values.



Press the "Edit" key to open a complete list of possible or existing values.



Position the cursor on the value of your choice using the "Direction" keys.

Always confirm your input with the "Input" key. The value is accepted.

With this key, you can switch to the next value in the selection list without displaying the entire list (e.g. to select from only a small number of values/settings).

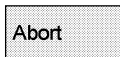
- The editor only displays those characters that can be entered on the operator panel front keyboard.
- A parts program opened by the editor cannot be simultaneously started in the NC (enable is canceled); an alarm (14014) is indicated in this case. If the control is switched off while the editor is open, the enable might have to be set manually.

2.6.5 Confirming/canceling an input



Confirm input:

Your inputs are accepted when you press softkey "OK". The selected function is executed. The window is closed and you return to the menu level from which the window was called.



Cancel input:

Your inputs are rejected if you select the softkey "Abort". The selected function is aborted. The window is closed and you return to the menu level from which the window was called.

The response is the same on return from a function (vertical softkey bar).



The "Edit" key can act as an "Undo" function if you abandon the input/modification you have just entered. The cursor remains positioned in the currently selected field.



Switch from the horizontal menu level back to the call menu level.

2.6.6 Editing a parts program in the ASCII editor



Function

The ASCII editor provides you with the following functions:

- Switch between insert and overwrite mode
- Mark, copy, delete block
- Paste block
- Position cursor/Find text/Replace
- Save file
- Generate contour (programming support)
- Configure cycle parameters (drilling, milling, turning)
- Start simulation
- Recompile (cycles, free contour programming)
- Renumber blocks
- Change settings
- Open 2nd file



Additional notes

As a rule, a parts program selected in the NC can only be edited in the reset state of the channel.

When a parts program is selected and the relevant channel is in the "Channel Reset" state, the program can be fully edited.

Please note the following differences in the ASCII editor:

- You can edit a program that has been loaded to the NC or a program stored on the hard disk. How programs are saved on the hard disk is determined by settings.
- The end of block character is displayed not as " L_F " but as "¶".

Sequence of operations

The following functions are fully enabled in the Program and Services operating areas but only partially in the Machine operating area.

In the Machine operating area, the ASCII editor is called via the program editor, in Services, by selecting a file in the file manager.

You have selected the file you wish to edit in the directory and pressed the Enter key, the vertical softkey bar changes. Your selected file is opened in the text editor.

Cursor block:

Use the "Direction" keys to position the cursor in the text.

You can page up and down using the "Page" keys.

The character on which the cursor is positioned is deleted.

Press the "Backspace" key to delete the character to the left of the cursor.



Press the "Input" key to end a block. The characters "LF" ("Line Feed") are automatically generated.

Vertical softkeys

Overwrite

Overwrite

The softkey toggles between insert and overwrite modes.

Mark
block

Mark (select) block

When you press this softkey, the vertical softkey bar changes. The softkey marks the beginning of a block.

Now position the cursor at the end of the block.

The block is selected automatically.

Copy
block

The softkey copies the selected block into the buffer.

If a line which was generated by a support function is selected, the complete support block is copied.

The block remains stored in the buffer even after the replacement of a parts program.

Delete
block

The selected block is deleted.

If a line which was generated by a support function is selected, the complete support block is deleted.

Mark
block

Selection mode is canceled when you select "Mark block".

Paste
block

Paste block

The softkey pastes the cut or copied block from the buffer into the text in front of the cursor position.

Find/
Go to ..

Find/Go to...

The window "Find/Go to..." is opened.

You can select functions for positioning and searching via the vertical softkeys:

The following search methods are available:

- to the beginning of the parts program (cursor on the first character in the program),
- the end of the parts program (cursor on the last character in the program) and
- position on a particular NC block with "Go to .."

Start of
Program

End of
Program

Go to ..



- or search for a particular character string with "Find".

"Go to.....": Enter the block number you are looking for.

- If the line being searched for contains an "N" or ":", you are taken to that line,
- If there is no block number, the cursor is positioned on the block with the specified number.

Press softkey "OK" or the "Input" key to position the cursor on the block number/line number of your choice.

The "Go to..." window is closed.

If you abort positioning, the process is stopped and the window closed.




"Find":

Enter the character string you wish to find.

The string you enter is sought downwards from the current cursor position, the find result appears as highlighted text.

You can start a new search by pressing softkey "Find next" or the "Input" key.

Enter the new text with the "Replace" softkey.

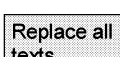
The text found is replaced by the "replacement text". The new text is replaced when you press "Input". Every time you press "Input", a new find and replace process is started.



or




or

Enter the new text with the softkey "Replace all texts". The query asks you "Do you really want to replace all non write protected strings: ... globally with ... ?".

Note: This function can be interlocked with a password, see /IAM/ IM4 Installation and Start-up of HMI Advanced (PCU 50).

The function "Replace all texts" is only enabled for files located on the hard drive (i.e. not in the NC memory).



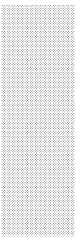
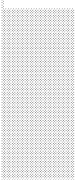
If you abort Find + Replace, the process is stopped and the window closed. You are in "Edit mode" again.



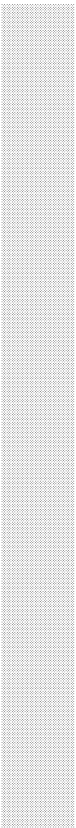
Save file

Changes are saved in the file loaded in the editor.





Close editor



Contour

Generate contour

Accept contour

Drilling

Milling

Turning

OK



Additional notes

Please note that the changes to programs stored in the NC memory take immediate effect.

The save options for the control system can be altered in the "Settings" menu (e.g. save automatically, etc.).

(See "Start-up" section)

Close editor

When you select softkey "Close editor", a dialog box may appear in which you must confirm whether or not to save the changes. The text editor is then closed and the current program overview is displayed again.

Horizontal softkeys

Free contour programming

Use the "Contour" and "Generate contour" softkeys to call up the free contour programming function.

The parts program block is inserted in the parts program with the appropriate parameters.

Cycle parameterization

The following functions are provided as programming support:

- Drilling, milling, turning (cycles)
- Contour (free contour programming)

You can call up the relevant cycle compilation via the vertical softkeys Drilling, Milling, Turning.

Enter the new values for the cycle parameters.

The parts program block is inserted in the parts program with the appropriate parameters.

Example:

```
CYCLE81 (110, 100, 2, 35)
```

/PGZ/, Programming Guide, Cycles



Simulation

The simulation is called (see /BA/, Operator's Guide, Interactive Programming or Chapter 6 Program Operating Area).

Softkey "Simulation" is displayed only if the simulation function can be called in the current operating situation.




Cycle recompilation

If program steps (cycle/contour) have already been parameterized but must be changed, you can display and edit the parameter values with their meanings.

Position the cursor in the text editor on the line with the program step (contour/cycle) whose parameters you wish to change.

A screen form containing the parameter settings for the selected cycle/contour appears on the screen.

Change the parameters.



The parts program block is automatically inserted in the parts program with the new parameters.



The block numbering in the program shown in the editor is executed again according to the values defined under the "Settings" softkey.



Settings

Define the following values in the window "Editor Settings":

- Horizontal scrolling ON/OFF
- Display hidden lines ON/OFF
- Deactivate LF in program

If this setting is selected, a space is displayed in the editor window on the monitor in place of the linefeed character. In the file to be edited, the linefeed character is retained.

- Time interval for Automatic Save

When you set Automatic Save, you can also set the time intervals at which text must be saved automatically (applicable only to files on hard disk). If a value $\neq 0$ is entered, the softkey "Save file" is not displayed. If the value 0 is entered, automatic save is not performed.

- Automatic numbering ON/OFF
A new block number is automatically inserted on every new line. If you wish to assign new block numbers to an existing program, use the "Renumber" function.
- Number of first block
- Incrementation of block numbers (e.g. 1, 2, 10)

Setting
contour prg.

The following settings are possible for contour programming:

- Last line
Each time you complete a program step in the contour programming you can insert a text in the last line (e.g. "End of contour").

Additional notes

- The coordinate system and the technology to be used are set via the machine data; see /IAM/ Installation and Start-Up Guide IM4.
- Edited programs are automatically enabled after saving.

2.6.7 Switch channel over



It is possible to switch between channels when several are in use. Since individual channels may be assigned to different mode groups, a channel switchover command is also an implicit mode switchover command.

If the selected channel is linked to another NCU (m:n link), the HMI Advanced is also switched over implicitly to the relevant NCU.

When a "Channel menu" is configured, all existing communication links to other NCUs plus the associated channels are displayed on softkeys.

Channel states

The following three channel states can occur in each of the above modes:

1. Channel reset

The machine is in the initial state, e.g. after power-on or after end of program. The initial state is defined by the machine manufacturer in the PLC program.

2. Channel active

A program has been started, the program is being executed or a reference point approach is in progress.

3. Channel interrupted

The current program or reference point approach has been interrupted.

In this context a program can be a main program, subprogram, cycle or a series of NC blocks.



There are 3 different switchover levels:

1. Switch to next channel.
2. Switch over configured channel group/channels (1 NC).
3. Switch to another NC (with m:n link involving several NCs).

2.6.8 m:n communication links

General information

The term m:n link is used to indicate a configuration in which **m** HMI Advanced units and **n** NCU/PLC units are interconnected. This does not, however, mean that all possible links are actually active.

In this case, an HMI Advanced is linked to only one NC at any given point in time (cf. 1:1 link) and communicates only with this unit. With an m:n link, the link can also be switched over to another NC. Using the channel switchover key and channel menu, you can switch the MMC over to another link.

The Channel Menu function is an option and must be configured in the "NETNAMES.INI" file.

You can go to the channel menu by pressing the channel switchover key in any of the operating areas. When you do so, only the horizontal and vertical softkeys change.

Use the horizontal softkeys to select a channel group (max. 24) 8 links to channels on different NCUs can be set up in each channel group.

All current communication links and associated symbol names are listed in display "Channel menu".



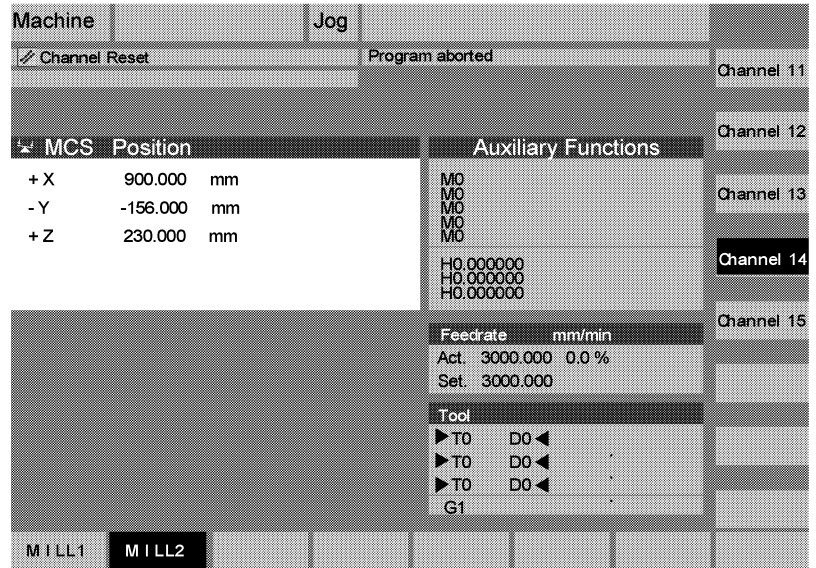
Important

Only two links may be active simultaneously on one NCU.



Function

You can establish a link via the operator interface in any operating area between the HMI unit and other connected NCU/PLC units.



Press the channel switchover key. The current link can be identified by highlighted horizontal and vertical softkeys when the channel menu is active.

Switch channel over

It is possible to switch over to other channels by means of the softkeys in the vertical menu.

Switch group over

You can switch over to another group by selecting a softkey in the horizontal menu; the vertical softkeys now display the channels of the currently selected group. You can only switch over to another channel (and thus possibly to another NC) by selecting one of the vertical softkeys.

Switch NC over

You can switch over to another NC by selecting a channel which is not linked to the current NC on one of the vertical softkeys.

Additional notes

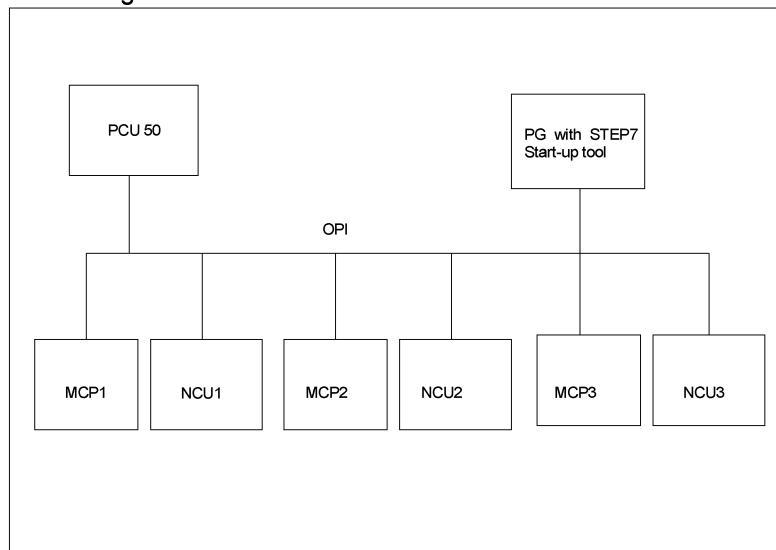
- Horizontal softkeys are assigned to vertical softkeys in the NETNAMES.INI file. The assignment merely represents an HMI-specific grouping characteristic.
- When you select a vertical softkey, you are selecting a channel and potentially an NC as well.
- Channels that are configured in the channel menu, but defined as a channel gap in the corresponding NC, will not be displayed.
- If an application (e.g. processing from external source) disables switchover to the selected NC, then only the channels of the current NC are displayed in the channel menu.

Example: 1 HMI Advanced and 3 NCUs

An HMI unit can be linked to several NCU/PLC units. The machine control panel MCP is assigned permanently to the NCU. You can connect an additional programming device (PG) with start-up tool.

The configuration illustrated below allows several NCUs to be operated from one HMI, i.e.

- several autonomous machines with several NCUs or
- one large machine with several NCUs.



The following rules apply to the operation of several NCUs from one HMI:

- The NCU to be operated is selected via the channel switchover key and the channel menu.
- The softkeys highlighted in the channel menu indicate the mode group/NC/channel to which the HMI is currently linked.

After setting up a link to another NCU, the operating area selected last is always available for this (as for the NCU whose link was separated).

2.6.9 Pocket calculator function



Precondition:

The cursor is positioned on an input field or input/output field.

Equals key

Press this key to switch to **pocket calculator mode**.

If a fundamental operation of arithmetic sign (+, -, /, *), followed by a value (e.g. 13.5) is entered in this mode



and then the input key is pressed, the next value to be entered is calculated with the previous value.

If the input/output field is opened with the input or equals key, the editor is in insert mode; if the field is opened directly with a character, the editor is in overwrite mode.

Inch/metric conversion

In pocket calculator mode, you can convert numeric values from the metric to inch system by entering "I" and from the inch to metric system by entering "M".

Proceed as follows to convert values:

- Position the cursor on an input field which contains a numeric value or enter a value
- Press the equals key
- Enter the letter "I" (convert to inches) or "M" (convert to metric)
- Press the "input" key, the value is converted.




2.7 Help



Function

Whenever the symbol "i" appears in the dialog line, this means that

additional information can be called via the information key .

A comment appears in the dialog line.

Analogous to the Windows Help, an HMI Help function is provided on the HMIs. (The term MMC Help still appears in the SW from time to time.) If, for example, you have made an input error, you can select the HMI Help to access detailed information about the error that has just occurred, e.g. the Diagnostic Guide is displayed.

Available help functions include:

- Alarm Help
Detailed information on displayed alarm/message
- MD Help
Detailed information on selected MD/SD
- Editor Help
Press once for information in brief and press twice for detailed information on the command/function on which the cursor is positioned.



Page up

or

Page down

Next entry

Follow cross ref.

Go to ...

Zoom +

or

Zoom -

Exit help

By pressing the "Information" key, e.g. in the "Diagnosis" operating area, the HMI Help to access information about the alarm that is currently being indicated is automatically called up and displayed.

You can page through the document with the softkeys "Page up" and "Page down",

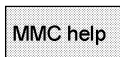
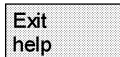
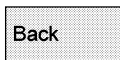
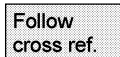
you jump to the next hit in the document with "Next entry".

Where there are cross references to other documents you can jump to a particular point in that document with this softkey.

You can search for any words in the document with the search function "Go to ...".

With the softkeys "Zoom +" and "Zoom -" you can increase or decrease the zoom factor in the document view,

with "Exit help" you return to the editor.



Analogous to the Windows Help, help is provided in some cases by the following softkeys:

Press the "Page down" or "Page up" softkey.
The screen contents are scrolled up or down.

Use the "Highlight fwd" or "Highlight back" softkeys to go to the entry of your choice.

Press the "Follow cross ref." softkey.

The entry you chose appears on the screen.

Press the "Back" softkey to return to the entry displayed beforehand.

Select and display an entry in HMI Help:

Press "Contents" softkey.

The current contents of HMI Help are displayed.

You exit the HMI Help and return to the previous menu.

Context-independent Help call:

When you press the "Area switchover" key you are taken back to the basic menu and
when you then press the "Etc." key

the function "MMC help" is directly invoked (corresponds to HMI Help).

2.7.1 Editor Help



Function

The following help functions are provided as programming support with the editing of parts programs in the editor via the "information" key:

- **Short help for programming commands**
configuring see /IAM/ HE1, Help in Editor.
 - Help in parts program for instructions: Display descriptive text (e.g. G9 "Exact stop – velocity reduction")
 - Display an overview of topics (e.g. "Preparatory functions", "Path commands", "Path travel behavior" etc.), to which instructions are assigned.
 - Display an overview of instructions with descriptive text
 - Search selectively in special screen forms on the basis of topic assignment or by entering a character string
 - Transfer the selected instruction to the Editor
- **Short help "Parameterization form" + extended help "pdf"**
Configurable parameterization forms from which you can jump to a particular page in the documentation (pdf file), parameterization forms for cycles, you jump to the Configuring Guide Cycles; for configuration see: /IAM/, BE1 Expanding the Operator Interface.
- **Short help for program commands + extended help "pdf"**.
From the context-sensitive help you can jump to the corresponding page in the documentation with the "information" key, e.g. jump to the Programming Guide, Fundamentals.

2.7.2 Short help for program commands



Function

You can call up a help function to provide programming support with the editing of parts programs via the "information" key. This help function can assist in the following ways:

- Display instructions with descriptive text context-sensitively with reference to the cursor position (e.g. G9 "Exact stop – velocity reduction")
- Display an overview of topics (e.g. "Preparatory functions", "Path commands", "Path travel behavior" etc.), to which instructions are assigned.
- Display an overview of instructions with descriptive text
- Search selectively in special screen forms on the basis of topic assignment or by entering a character string
- Jump from the context-sensitive help via the "information" key onto the corresponding page of the documentation, e.g. jump to Programming Guide, Fundamentals
- Jump into an input screen form with softkey "Input screen form" in which, for example, a cycle is assigned new parameters.
- Transfer the selected instruction to the Editor

Notes

If the help function is being used by one editor, it is not available for use by other editors.

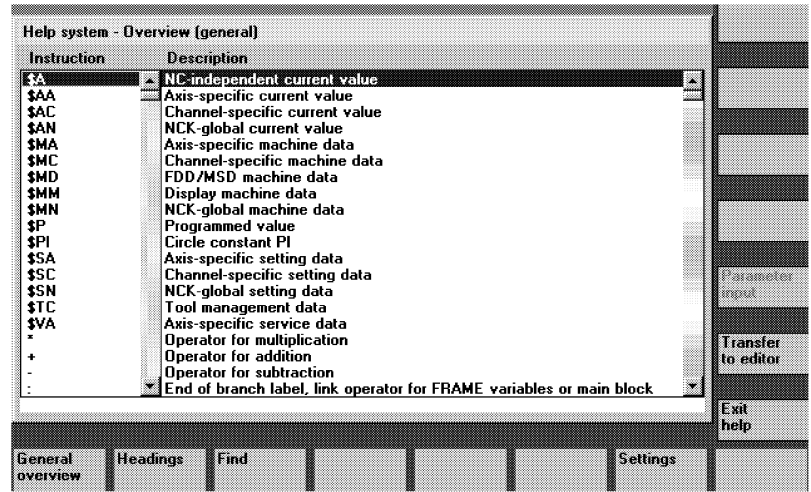
Sequence of operations

You call the help function in the editor with the "information" key.

Depending on the current context of the cursor:

- With standard context sensitivity ("displayed if wording is identical"), only the programmed instruction with descriptive text or
- with extended context sensitivity ("displayed if initial wording is the same"), additionally all instructions with the same initial wording or
- if there is no match, a full overview (see below) is displayed.





Transfer to editor

If transfer is possible, the instruction selected in the overview is inserted directly in the parts program with "Transfer to editor".

If an instruction selected with context sensitivity active is different to the programmed instruction, the programmed instruction is overwritten.

If extended context sensitivity is not active or if a different selection is displayed with "Find", "Headings" or "General overview", the selected instruction text is inserted in the parts program behind the instruction marked by the cursor.

Exit help

As an alternative to returning to the Editor by transferring an entry, this softkey can be selected to close the Help window and return to the parts program writing screen.

Apart from context-sensitive help, it is also possible to search independently of context for instructions, descriptive texts or topics.

General overview

With "General overview" a full overview of the instructions and relevant descriptive text stored in the help system is displayed.

Headings

With "Headings" the stored headings are listed for which a functional grouping of the instructions can be displayed.



Display heading

Find



Start search

Settings

To select a heading, you can either use the cursor keys or enter the heading number in an input box.

If a heading is selected and you press "Input" or the softkey "Display heading", the instructions belonging to the selected heading are displayed.

With the softkey "Find" you can define a search text in an input window, which you can search under

- "Instruction text only"
- "Descriptive text only"
- "Instruction and descriptive texts".

No distinction is made between upper and lower case.

With "Input" or "Start search" a search is made using the defined search text in accordance with the instruction or descriptive part. Any matching instructions or descriptions found during the search are displayed.

Notes for setting the help system in the editor

The editor help uses a standard text file (see /IAM/, "Installation and Start-up Guide IM4", Chapter "Help in editor"), which contains topics and instructions with descriptive text.

If you want to create an end user text file for the help in order to record your own instructions/topics, you can enter the path/name of the text file under softkey "Settings" in an input window as "end user text file". You can also change the context sensitivity under "Settings".

You can choose between:

- "Display with same initial wording" (extended context sensitivity) and
- "Display with identical wording"

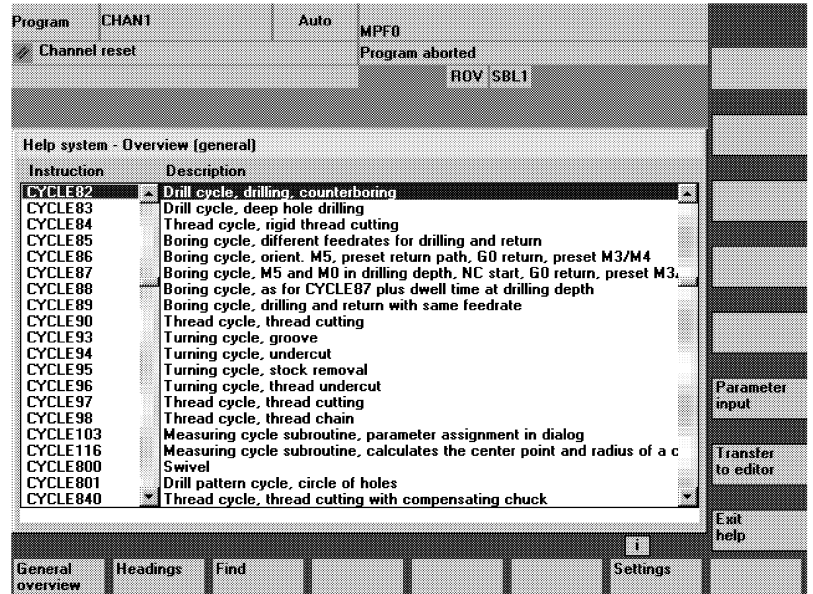
If, for example, in the option "Display with same initial wording" the cursor is located to the right of the instruction "G4", all instructions with the same initial wording, e.g. "G40, G41, G42" are also displayed.

With "Display with identical wording" the current instruction, e.g. "G4" is displayed.

The settings are activated by restarting the help system.



If you select the individual descriptions in the full overview with the cursor key, you can jump directly to the Programming Guide, for example, with the "info" key, if the symbol for the info key is displayed on the bottom right part of the screen (see Fig. below).



Parameter input

If "Parameter input" appears in the vertical softkey menu, you can assign parameters in an input screen for a particular instruction (e.g. a cycle).

2.7.3 Extended help for programming commands



Function

You can call up a help function (short help) to provide programming support for editing parts programs via the "information" key in the program command editor.

If the short help is not sufficient, you can also open the Programming Guide (pdf file) by pressing the "information" key again. The command you are looking for is highlighted in the document.



Sequence of operations

Precondition:

The cursor is positioned on a programming command (e.g. G01).

You can call up the help function (short help) in the editor with the "information" key.

Press the "information" key once more to open the Programming Guide (pdf file) with the Adobe Acrobat Reader.



2.8 Job list



Function

With SW 5 and higher, it is possible to create a job list (loading list) for each workpiece.

This list contains instructions which prepare the following for the execution of parts programs (in several channels if necessary):

- Parallel setup (LOAD/COPY) i.e.:
Load or copy main programs and subprograms and associated data such as
 - initialization programs (INI)
 - R parameters (RPA)
 - user data (GUD)
 - zero offsets (UFR)
 - tool/magazine data (TOA/TMA)
 - setting data (SEA)
 - protection zones (PRO) and
 - sag/angularity (CEC)from the hard disk of the HMI to the main memory of the NC.
- Preparations for NC Start (SELECT), i.e.:
Selection of programs in various channels and start preparations for execution.
- Parallel cleanup (reversed LOAD/COPY) i.e.:
Unload main programs and subprograms and the associated data from the main memory of the NC onto the hard disk of the HMI.
- Backup (in preparation for the next software version)

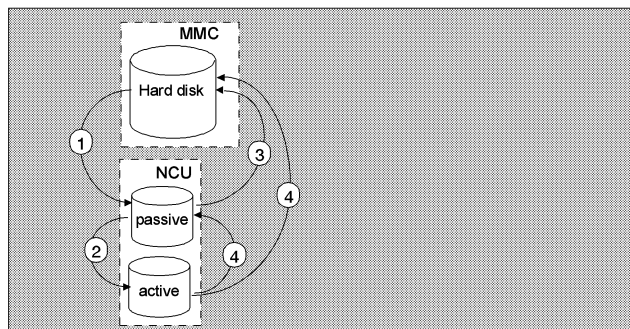
Notes

Parallel setup, preparations for NC Start, parallel cleanup and backup can also be executed from the PLC.

The job list is executed if the workpiece contains a job list of the same name.

The instructions of the job list are activated (see diagram) on

- ① Parallel setup with "Load" (LOAD/COPY)
- ① "Select" (LOAD/COPY/SELECT)
- ② "NC Start" (program is executed and SELECT is activated)
- ③ Parallel clearing with "Unload" (reversed LOAD/COPY)
- ④ "Back up" (being prepared for the next software version)



Creating file "Workpiece.JOB" (e.g. SHAFT.JOB)

There are various methods by which job lists can be created:

- While creating a workpiece directory with the function "New", generate a standard job list as a file in this workpiece. The job list syntax is included as a comment in this standard job list.

To do that, check the field "Create templates for job list" under "Start-up/MMC/System settings/Templates.

With this method, the file is automatically assigned the name of the relevant workpiece directory `workpiece.JOB` (e.g. `SHAFT.JOB`).

- Creating job lists with different names in an existing workpiece directory with the function "New".
- Job lists can be inserted in an existing workpiece directory.

The job list can be modified with the Editor.

Notes

You can create your own templates for job lists or standard parts programs/subprograms in the directory `\Templates\Manufacturer` or `\Templates\User`. The data manager always searches the User directory first, then the Manufacturer directory, and finally the Siemens directory.

It is recommended that you name the template `_templ_`. `_templ_` is replaced by the workpiece name when you create a new workpiece. For example, if `_templ_.job` or `_templ_.mpf` is found, it is renamed `SHAFT.JOB` or `SHAFT.MPF` in the workpiece `SHAFT.WPD`. If the templates are to be language-dependent, the language abbreviation must be appended with an underscore prefix to the end of the template name. When you create a new workpiece, the template with the abbreviation of the current language is used. The abbreviation is removed when the file is copied (e.g. `_templ_.gr.job` becomes `SHAFT.JOB`). Templates without a language abbreviation are always copied. The template is always selected when the workpiece is created, i.e. the language cannot be changed subsequently.

2.8.1 Description of syntax for job lists



Explanation

The job list syntax consists of 3 instructions

- Load instruction `LOAD`
- Select instruction `SELECT`
- Copy instruction `COPY` (for m:n link only)

Notes

As regards the job list commands, a distinction must be made between an m:n network and a 1:1 link between the HMI and NCs. It is advisable to use the instruction `LOAD` for a 1:1 link and `COPY` for an m:n link, at least for global programs and, in particular, cycles used in several NCUs.

Comment

All terms placed inside "brackets" or ";" are comments and are ignored when the job list is processed.



Description of syntax

LOAD [source]

The LOAD instruction loads one or several files from the HMI to the NC main memory, deleting the source file on the HMI. In other words, the files are stored only once.

This instruction is recommended for a 1:1 link.

[source] is [path]/[name]

The path/name defines the relevant path within the file tree of the data manager.

Wildcards (e.g. *) may also be used in the name.

Examples:

LOAD *

(load all files from workpiece directory of job list)

LOAD /MPF.DIR/*

(load all files from one directory, e.g. in this example, all files from parts programs (MPF.DIR))

LOAD PART1.MPF

(load one file, e.g. PART1.MPF, from the workpiece directory currently selected in the job list)

LOAD /SPF.DIR/PART1.SPF

(load one file from a directory, in this case from subprogram directory SPF.DIR)

SELECT [source] [destination] [DISK]

The SELECT instruction selects a program for execution. The selected program must be loaded to the main memory of the NC. It can then be started with NC-START.

If programs on the hard disk must be executed, then the command must include vocabulary word DISK.

[source]

is the name of the main program which is selected for execution in a specific channel in the NCK.

[destination]

A channel must be specified as the destination.

CH=

Channel number (for a 1:1 link only);

or

with NETNAMES.INI: Channel name (channels are unambiguously distributed across all NCs);

or

NC name, channel number

Example:

CH=2

(2 is the channel number)

CH=Station5

(Station5, corresponds to channel name from NETNAMES.INI)

CH=ncu_b,1

(ncu_b corresponds to NCU name from NETNAMES.INI
1 is the local channel number of this NCU)

[DISK]

can be optionally specified for program execution from hard disk.

Examples:

```
SELECT PART12 CH=CHANNEL22
```

```
SELECT PART12 CH=NCU_2,2
```

(PART2 is selected in the 2nd channel of NCU_2)

```
SELECT /shaft1.wpd/side1.mpf CH=2 DISK
```

(Parts program SIDE1.MPF of workpiece SHAFT1.WPD
is executed from the hard disk in the 2nd channel)

COPY [source] [destination]

The COPY instruction copies one or several files from the HMI to an NC main memory. The original files remain stored on the MMC.

The COPY instruction is executed only if the relevant file does not yet exist at the destination or if it has a different time stamp.

This instruction is recommended for an m:n link.

If this type of file needs to be modified, it must always be edited on the NC. If the file has been distributed more than once via the job list, and you want the changes to be active in all NCs, you must first unload that file, then edit it, and finally redistribute it via the job list.

[source] is [path]/[name]

[destination] is the NCU/channel address:

The destination can be specified using one of the three address categories. It must be noted that only the logical names from NETNAMES.INI are taken into account.

NC= Name of NCU

Without NETNAMES.INI, the only option in this case is to specify the NC name with NC=.

CG= Name of the channel group, i.e. copy in every channel of this group (therefore in all NCs to which the channels are assigned).

Parameter CG can be specified only if a channel menu is configured.

CH= Name of channel

Channel names are only assigned uniquely across NCs when a channel menu is configured.

If no destination is specified, the source is copied to the NC currently linked to the MMC. If * is entered as the target, the source is distributed to all the NCs configured in NETNAMES.INI.

Examples:

COPY * *

(copies all files from the workpiece of the job list to all NCs that are configured in NETNAMES.INI)

COPY PART12.MPF NC=NCU_2

(copies a file from the workpiece of the job list into the NC "NCU_2")

COPY /SPF.DIR/PART1.* CG=MILL2

(copies all files with a name from a directory e.g. PART1.* from subroutines (SPF.DIR) into a channel group, i.e. to all NCs to which channels of this group are assigned)

COPY /MPF.DIR/* CH=CHANNEL22

(copies all files from a directory, e.g. all from parts programs (MPF.DIR) to the NC which is assigned to this channel)

2.8.2 Example of a job list for two-channel 1:1 links



Example

If only channel1 and channel2 on NCU1 (1:1 link) were involved in machining of the workpiece, then the most suitable job list structure would be as follows:

```
LOAD /MPF.DIR/Gen.MPF
LOAD /WCS.DIR/Part1.WPD/WpdGen.MPF
```

```
LOAD /WCS.DIR/Part1.WPD/ Channel1.MPF
LOAD /WCS.DIR/Part1.WPD/ Channel1.INI
LOAD /WCS.DIR/Part1.WPD/ K12.MPF
```

```
LOAD /WCS.DIR/Part1.WPD/ Channel2.MPF
LOAD /WCS.DIR/Part1.WPD/ Channel2.INI
LOAD /WCS.DIR/Part1.WPD/ K22.MPF
```

```
SELECT /WCS.DIR/Part1.WPD/Channel1.MPF CH=1
SELECT /WCS.DIR/Part1.WPD/Channel2.MPF CH=2
```

The destination is not specified for loading in a 1:1 link, the current NC is the default.

2.8.3 Example of a job list for multi-channel m:n links



Example

HMI1 to two NCs
 NCU1 with channel1 and channel2
 NCU2 with channel3

Part1.JOB:

```

COPY /MPF.DIR/Gen.MPF NC=NCU1 (or CH=CHANNEL1)
COPY /WCS.DIR/Part1.WPD/WpdGen.MPF NC=NCU1
    (or CH=CHANNEL1)
COPY /WCS.DIR/Part1.WPD/Channel1.MPF NC=NCU1
    (or CH=CHANNEL1)
COPY /WCS.DIR/Part1.WPD/Channel1.INI NC=NCU1
    (or CH=CHANNEL1)
COPY /WCS.DIR/Part1.WPD/K12.MPF NC=NCU1
    (or CH=CHANNEL1)

COPY /WCS.DIR/Part1.WPD/Channel2.MPF NC=NCU1
    (or CH=CHANNEL2)
COPY /WCS.DIR/Part1.WPD/Channel2.INI NC=NCU1
    (or CH=CHANNEL2)
COPY /WCS.DIR/Part1.WPD/K22.MPF NC=NCU1
    (or CH=CHANNEL2)

COPY /MPF.DIR/Gen.MPF NC=NCU2 (or CH=CHANNEL3)
COPY /WCS.DIR/Part1.WPD/WpdGen.MPF NC=NCU2
    (or CH=CHANNEL3)

COPY /WCS.DIR/Part1.WPD/Channel3.MPF NC=NCU2
    (or CH=CHANNEL3)
COPY /WCS.DIR/Part1.WPD/Channel3.INI NC=NCU2
    (or CH=CHANNEL3)
COPY /WCS.DIR/Part1.WPD/K32.MPF NC=NCU2
    (or CH=CHANNEL3)

SELECT /WCS.DIR/Part1.WPD/Channel1.MPF CH=CHANNEL1
SELECT /WCS.DIR/Part1.WPD/Channel2.MPF CH=CHANNEL2
SELECT /WCS.DIR/Part1.WPD/Channel3.MPF CH=CHANNEL3
  
```


2.8.4 Sequence of operations "Executing job list"



Manage data

E.g. press the softkey "Manage data" under "Services".
The horizontal and vertical softkey bars change.

Load

Position the cursor on the correct workpiece directory.
Then execute functions

- "Load"
- "Select"
- "Unload".

Select

Unload

If a job list named `workpiece.JOB` is stored in a workpiece directory, then the functions are executed for that job list.

However, the functions can be executed directly on the job list, e.g. if the directory contains job lists that are named differently from the workpiece.



Load

Additional notes

"Load" job list

"Load" means that all the prepared job list instructions are executed. The data are distributed from [Source] to [Target], to the target NCs with the instruction LOAD or COPY. The workpiece is then marked as "loaded".

SELECT instructions are ignored.

If the workpiece/job list is loaded, then the log window displays a list of the files to be distributed.

The log window can be displayed in the event of an error. Loading of the job list can then be aborted if necessary.

Marking of files on user interface

If the file is stored only on the hard disk of the HMI, then it is marked as "not loaded".

If the file is only located in the RAM of the NC, it is marked as loaded with an "X".

If the file is located both in the HMI and in the NC, the identifier "X" is only removed when the files are no longer identical.

If the files have different time stamps or are of different lengths, the identifier is "!X!".



Select

"Selection" job list

If you select a job list or workpiece with job list, all instructions of the job list are executed.

LOAD instructions are executed if the files are still stored on the HMI. COPY instructions are executed only if the files are not stored on the NC or if the file copy on the NC has a different time stamp to the HMI copy. If the time stamps are different, the system inquires whether the file must be overwritten.

SELECT instructions are executed.

Unload

"Unload" job list

"Unload" means that the instructions of the job list are "undone", the instructions in the job list are executed in reverse order, e.g.:

Data which have been loaded to a destination NC with a LOAD instruction are unloaded from [destination] to [source] into the source directory on the HMI.

Data that have been copied to a destination NC with COPY are erased at the [destination], provided they have the same time stamp as the original. If the NC file copy has been modified, a query box appears, asking whether the NC version must be transferred to the HMI.

With "Unload" only files are transferred from the passive file system of the NC. If, for example, modifications have been made to the active data in parameters, these must be saved beforehand.

2.8.5 Rename workpieces with job lists



Function

When you rename a workpiece directory, all the workpiece files under that directory that have the same name as the directory are renamed. If a job list with the name of the directory exists, the instructions in that job list are also renamed.

Comment lines remain unchanged.

Example:

Workpiece directory `A.WPD` renamed to `B.WPD`:

All files with the name `A.XXX` are renamed to `B.XXX`, i.e. the extension is not altered.

If a job list called `A.JOB` exists it is renamed to `B.JOB`.

If this job list contains instructions of file `A.XXX` located in this workpiece directory, then that file is also renamed to `B.XXX`.

Example:

If job list `A.JOB` contains an instruction

```
LOAD/WKS.DIR/A.WPD/A.MPF
```

it is renamed to

```
LOAD/WKS.DIR/B.WPD/B.MPF
```

However, if a job list contains the instruction

```
LOAD/MPF.DIR/A.MPF
```

 or

```
LOAD/WKS.DIR/X.WPD/A.MPF
```

the files are not renamed.

Sequence of operations

The softkey "Manage programs" in the "Program" operating area must be pressed.

Place the cursor on the workpiece directory that you want to rename.

The "Rename" dialog window opens.

Enter the new name.



Manage
programs



Rename

2.8.6 Copy workpieces with job lists



This function applies to the "Program" operating area only.
When you copy under "Services", the names remain unchanged.



Sequence of operations

Manage
programs

The softkey "Manage programs" in the "Program" operating area must be pressed.



Copy

Position the cursor on the file that you want to copy and press the softkey "Copy".

The file is marked as the source for copying.

Insert

Press the softkey "Insert", enter a new name, if necessary, and confirm with "OK".

OK

2.8.7 Archiving workpieces with job lists with m:n



Function

When archiving workpieces that contain job lists of the same name, in the case of m:n you are asked whether the job lists to be unloaded are to be executed. You can terminate the action with "Cancel", otherwise all job lists are executed and archiving is then started.



Sequence of operations

Data
from

In the "Services" operating area press the "Data from" softkey. The "Programs/Data" file tree is displayed.

The vertical softkey bar changes.

Please refer to Chapter "Reading out data" in "Services" operating area for additional sequences of operation.



Operating Example

Typical operating sequence

To provide support for entry-level users or an orientation guide for others, this section uses a typical operating sequence (from control system power-up to back-up of a user-generated parts program) to explain how the functions described can be located.

	Step	Described in Section
Setup	• Switch on machine	1.3
	• Reference point approach	4.3
	• Clamp workpiece/blank	
	• Select tools	
	• Define workpiece zero for coordinate inputs	
	• Enter tool offsets	5.2
	• Calculate speeds and feedrates	4.2.4
	• Define a reference point (scratching)	4.4.6
Enter/test a program	• Create a parts program or read one in via an external data interface	2.6.6 7.5.3/7.5.6
	• Select a parts program	6.8.5
	• Test a program (without a tool) <ul style="list-style-type: none"> • Start a parts program (e.g. in single block) • Edit parts program using program editing function or diagnostics guide/help 	4.2.1 2.4.6 4.6.7/2.6.6 8, 2.7
	• Optimize a parts program	6.6.5
Machine a workpiece	• Insert tool and run machining program	4.2.1
Save a program	• Save a parts program <ul style="list-style-type: none"> • to hard disk • Read in/out via RS-232 interface 	6.8.7 6.2 7.5.3/7.5.4

"Machine" Operating Area

4.1	Data structure of the numerical control	4-99
4.1.1	Operating modes and machine functions	4-100
4.1.2	Operating mode group and channels	4-102
4.1.3	Status display with symbols for all channels (SW 6.2 or later)	4-103
4.1.4	Select operating mode, change operating mode	4-104
4.2	General functions and displays	4-107
4.2.1	Start/stop/cancel/continue parts programs	4-107
4.2.2	Display program level	4-108
4.2.3	Switching between the machine/workpiece coordinate system (Machine-MCS/Work- WCS)	4-109
4.2.4	Displaying axis feedrates	4-111
4.2.5	Displaying G functions and transformations	4-111
4.2.6	Displaying auxiliary functions	4-112
4.2.7	Displaying spindles	4-113
4.2.8	Handwheel	4-114
4.2.9	Status of synchronized actions	4-115
4.2.10	Preset	4-117
4.2.11	Set actual value	4-119
4.2.12	Inch/metric switchover	4-121
4.3	Reference point approach	4-123
4.4	Jog mode	4-126
4.4.1	Function and basic display	4-126
4.4.2	Traversing axes	4-129
4.4.3	Inc: Incremental dimension	4-130
4.4.4	Repos (repositioning)	4-131
4.4.5	SI (Safety Integrated): User confirmation	4-132
4.4.6	Scratching (SW 6 or later)	4-134
4.4.7	Scratching (SW 5 or earlier)	4-137
4.4.8	Display of system frames	4-140
4.5	MDI mode	4-142
4.5.1	Function and basic display	4-142
4.5.2	Save program, file function	4-145
4.5.3	Teach In	4-146
4.6	Automatic mode	4-149
4.6.1	Function and basic display	4-149
4.6.2	Program overview	4-151
4.6.3	Loading and unloading the workpiece/parts program	4-152

4.6.4	Log: Loading list of the programs	4-153
4.6.5	Executing programs from the hard disk.....	4-154
4.6.6	Access to external network drive	4-155
4.6.7	Program editing.....	4-157
4.6.8	Block search/setting the search destination	4-159
4.6.9	Fast external block search.....	4-162
4.6.10	Start of search in Program test mode, multi-channel	4-166
4.6.11	Overstore	4-169
4.6.12	Program control	4-171
4.6.13	DRF offset.....	4-175

4.1 Data structure of the numerical control



Environment

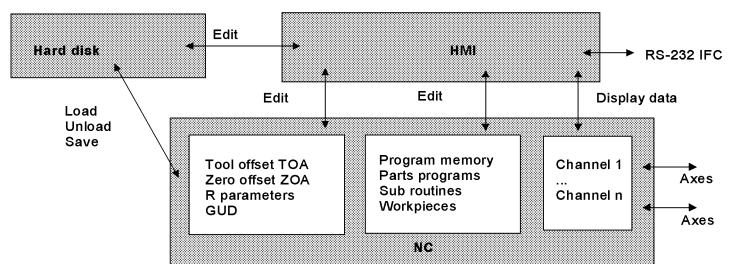
HMI

Function

The following components exist:

- NC with parts program memory
- HMI with hard disk

Use the softkeys "Load" - "Unload" to transfer the data to the NC or from the NC to the hard disk.



Data in the NC memory are retained when the control power supply is switched off. Only one copy exists of programs that are loaded from the hard disk to the NC memory. The program memory in the NC is limited (see memory display).

4.1.1 Operating modes and machine functions

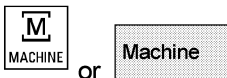


Function

The Machine operating area contains all the functions and control variables that trigger actions on the machine tool or detect its status. There are three different modes in this area:

- **Jog:** Jog is the mode required for manual operation and setting up of the machine. The setup functions provided are "Reference point approach", "Repositioning", "Handwheel" or "Traverse in preset increments" and "Redefine control zero" (preset).
- **MDI:** Semi-automatic operation
Parts programs can be set up and processed block by block in this mode. The tested blocks can then be stored in the parts program. In Teach In mode, positions can be traversed and stored to generate motion sequences which are then stored in the MDI program.
- **Automatic:** Fully automatic operation
Parts programs are executed fully automatically in Automatic mode, i.e. they are selected, started, corrected, selectively manipulated (e.g. single block) and processed.

Select machine



You can switch to the "Machine" area at any time from any of the other operating areas simply by pressing the "Machine area" key.



When you switch on the control, it is usually in the operating area "Machine" in operating mode "Jog". (Please consult the machine manufacturer's documentation!)



Machine manufacturer

The state after power-up can be configured and might therefore deviate from the default.

Machine functions



In operating mode "Jog" you can select the following machine functions via the machine control panel or softkeys in the basic menu:

Inc (traverse in preset increments)

Repos (reposition at a defined position)

Ref (reference point approach to coordinate machine and control zero points)

In "MDI" mode, it is possible to select "Teach In" (storage of motion sequences in a parts program through position approach) by pressing the MCP key.

Preparation for production

To start actual production some preparatory measures must be taken:

1. Load the tools and workpiece,
2. traverse the tools and the workpiece to the starting position defined in the setup plan,
3. load the parts program into the memory of the control,
4. check and enter the zero offsets,
5. check and enter the tool offsets.

4.1.2 Operating mode group and channels



Function

Every channel behaves like an independent NC. A maximum of one parts program can be processed per channel.

- Control with one channel:
One mode group exists.
- Control with several channels:
Channels can be joined to form several mode groups.

Example:

Control with 4 channels, where machining is carried out in 2 channels and 2 other channels are used to control the transport of the new workpieces.

Mode Group1	Channel 1 (machining)
	Channel 2 (transport)
Mode Group2	Channel 3 (machining)
	Channel 4 (transport)

Technologically related channels can be combined to form a mode group.

Axes and spindles of the same mode group can be controlled by one or more channels.

An operating mode group is either in operating mode "Automatic", "Jog" or "MDI", i.e. several channels of an operating mode group can never assume different operating modes.

Channel status with symbol (SW 6.2 or later)

The channel status and the channel operational messages are output with symbols in the channel status line:

Channel status

Channel interrupted

Channel active

Channel reset

Channel operational messages

Stop: Action is required

(e.g. cancel feedrate disable).

Wait: No action required, the NC is waiting

(e.g. for axis enable).



4.1.3 Status display with symbols for all channels (SW 6.2 or later)



Function

The channel, feedrate, spindle status and, if necessary, the machine or only the machine is output in the program status line with symbols via several channels.

The display can be configured by the machine manufacturer.

Manufacturer-defined symbols can also be used in addition to standard symbols.



Machine manufacturer

Please follow the machine manufacturer's instructions!

The following SINUMERIK-specific status displays are available:

- Channel status with overlaid spindle and feedrate stop
- Channel status with overlaid feedrate stop
- Channel status
- Spindle status

Meaning of colors:

Red Machine/PLC stopped

Yellow Waiting for an operation

Green Machine/PLC running

Gray Other

Channel status

Channel interrupted

Channel active

Channel reset

Feedrate status

Feedrate not enabled

Spindle status

Spindle turning left or right

Spindle not enabled

Spindle stop



or



Application example with 4 channels and 2 spindles:

Maschine	☐ CHAN1 ◊ CHAN2	AUTO	\WKS.DIR\DAUERTE_CYC950.WPD DAUERTE_NUMBERFEHL.MPF							
☐ Kanal unterbrochen	1	☐ 2	☐ 3	☐ 4	☐ S1	☐ S2				
⚠ Halt: Satz in Einzelsatz beendet	SKP1	DRY	ROV	SBL1	M01	M17	DRF	PRT		

4.1.4 Select operating mode, change operating mode



Function

The defined modes for operating a SINUMERIK control system are Jog, MDI and Automatic. They are selected via the MCP or by means of softkeys.



Machine manufacturer

Whether the requested mode can be accessed and the manner in which it is accessed can be configured machine-specifically in the PLC program.

Mode change

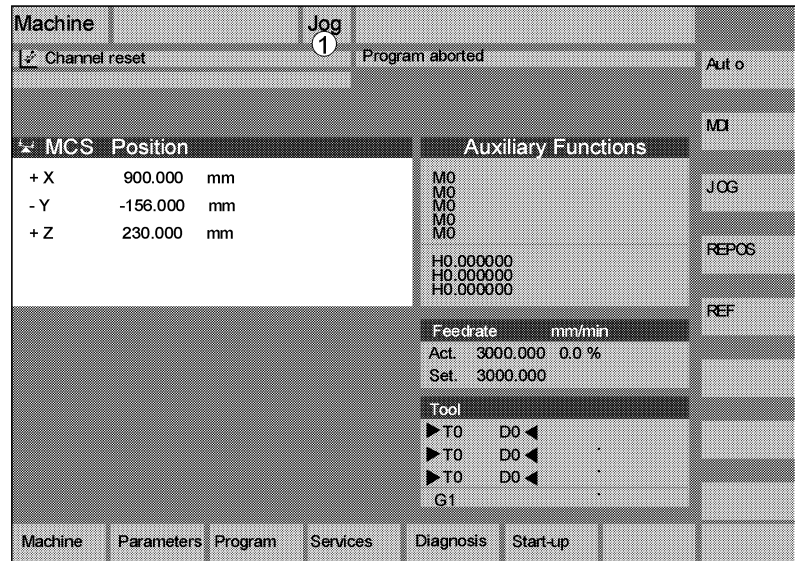
Not all mode changes are permitted.
An error message is output if a mode change request is rejected by the system. The error message will indicate the error cause and possibly the remedy.



Sequence of operations

Selection of mode

The selected mode is displayed in the mode field on the screen.



1 = operating mode

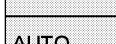
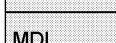
To select the modes:

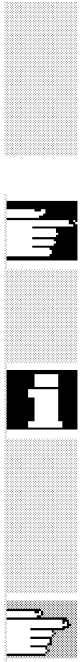
- Jog
- MDI
- Automatic

press one of the keys shown on the left on the machine control panel or

the corresponding vertical softkey that you can access via the "Area switchover key":

- JOG
- MDI
- Automatic





When a mode is selected, the LED next to the selection key on the MCP lights up. The same status is signaled in the mode field on the screen.

Additional notes

The basic display of the selected mode appears on the screen.

If a mode change is not possible, please contact your installation engineer, the machine manufacturer or our service personnel. In many cases, a mode change is enabled only for trained personnel for the purpose of safety. To provide this type of protection, the control system offers a facility for disabling or enabling mode changes.

/FB/, K1, Description of Functions

4.2 General functions and displays

4.2.1 Start/stop/cancel/continue parts programs



Function

The following section describes how to start and stop parts programs and to resume them after they have been aborted.

Sequence of operations

"AUTO" mode is selected in the "Machine" operating area.

Precondition:

No alarms are pending.
The program is selected.
Feedrate enable is active.
Spindle enable is set.



Start parts program:

The parts program is started and executed.



Stop/abort parts program:

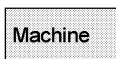
The program run is interrupted, but can be restarted with "NC Start".



The current program is aborted.

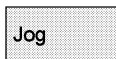
Resume parts program:

After interrupting the program with "NC Stop" you can retract the tool from the contour in "Jog" mode. The control saves the coordinates of the point of interruption. The distances traversed are displayed.



Repositioning:

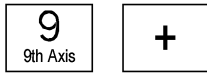
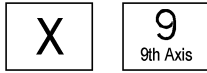
Select the "Machine" operating area.



Select "Jog" mode.



Press the "Repos" key to reposition the tool on the contour.

**Turning machine:**

Press the "+" or "-" key.

Milling machine:

Select the axis to be traversed and

then press the "-" or "+" key.

Traverse the axes to the point of interruption.

4.2.2 Display program level

**Function**

If subprograms are called while a parts program is running, the block numbers for the main program and subprograms together with their pass number (P) can be displayed.

**Sequence of operations**

"AUTO" mode is selected in the "Machine" operating area.

When you press softkey "Program level", the window headed "Program level" appears instead of the "Current block" window. The softkey labeling changes to "Current block".



Current
block

During execution of a parts program, the block numbers for the main program and subprograms, together with their pass number (P), are displayed in the "Program level" window. The main level is always visible, a nesting depth of up to 12 can be displayed.

When you press softkey "Current block", the "Current block" window containing the program blocks of the current parts program is displayed again.

4.2.3 Switching between the machine/workpiece coordinate system (Machine-MCS/Work-WCS)



Function

The display can be toggled between the machine and workpiece coordinate systems by means of special key "MCS/WCS" on the MCP or via softkeys (depending on MCP model and user program). The actual position display for the distance-to-go and the corresponding axes change.

Machine axes

Machine axes are axes that actually exist on the machine and have been parameterized during installation.

Geometry axes and special axes

These are the axes programmed in the parts program. Geometry axes and special axes are offset by the selected zero offset relative to the machine axes.

Three is the maximum number of Cartesian geometry axes.

Machine (MCS)

The machine coordinate system (MCS) refers to the coordinates of the machine axes, i.e. all machine axes are displayed in the machine coordinate system.

MCS position	Repos offset
X	
Y	
Z	

Work (WCS)

An offset (e.g. zero offset, rotation) can be used to set up a relationship, e.g. with the workpiece clamp. This relationship defines the position of the workpiece coordinate system (WCS) in relation to the machine coordinate system. The workpiece is always represented in a Cartesian coordinate system.

All geometry axes and special axes are displayed in the workpiece coordinate system.

WCS position	Repos offset
X1	
Y1	
Z1	



Machine manufacturer

Machine data are used to define whether the programmed frames are to be calculated when displaying the WCS (settable zero system display). Please consult the documentation provided by the machine manufacturer!



Sequence of operations

"Jog" mode is selected in the "Machine" operating area.

Act. val.
MCS

The actual values of the machine axes and their positions are displayed.

The softkey label changes to "Act. val. WCS".

The machine coordinate system is set up using all physically existent machine axes. Reference points, tool and pallet change points are defined in the Machine Coordinate System.

Act. val.
WCS

When you select softkey "Act. val. WCS", the geometry and auxiliary axes plus their positions appear in the "Position" window.

The softkey label changes to "Act. val. MCS".

The workpiece coordinate system is assigned to a specific workpiece. Settings in the NC program refer to the WCS.



WCS MCS

- You can also toggle between the workpiece and machine coordinate systems by pressing the "MCS/WCS" key on the MCP.
- The number of places displayed after the decimal point and units of measure can be set in machine data.

/PG/, Programming Guide, Fundamentals

4.2.4 Displaying axis feedrates



Axis
feedrate



or



Function

In operating mode "Jog" or "MDI" or "AUTO", you can display the current feedrate, residual path information and the associated override data.

Sequence of operations

"Jog" mode is selected in the "Machine" operating area.

Press the "Axis feedrate" softkey:

- When "MCS" is selected, the feedrate window containing the current feedrates, distance-to-go information and associated override is displayed.
- When "WCS" is selected, the feedrate window for the axes involved in interpolation, the current feedrate and the distance-to-go information with path override as well as the current feedrate and distance-to-go information with single-axis override for axes not involved in interpolation is displayed.

You can use the "Page" keys to display other axes if required.

4.2.5 Displaying G functions and transformations



G funct. +
transform.

Function

Active G functions and transformations in the currently active channel can be displayed.

Sequence of operations

"AUTO"/"MDI"/"Jog" mode is selected in the "Machine" operating area.

The "G functions + transformations" window containing active G functions and transformations is output on the screen.



or



Using the "Page" keys you can scroll up and down to display other G functions.



Additional notes

Every G group has a fixed location.

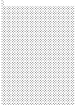
The group number (no.) and the current G function of the G groups are displayed only if a G function is active.

4.2.6 Displaying auxiliary functions



Function

Auxiliary functions active in the selected channel can be displayed.



Sequence of operations

"AUTO"/"MDI"/"Jog" mode is selected in the "Machine" operating area.

Auxiliary
functions

The "Auxiliary functions" window appears on the screen.

Up to 5 M functions and 3 H functions can be displayed.

4.2.7 Displaying spindles



or



Function

The current spindle values (actual speed, setpoint speed, position on oriented spindle stop and spindle override) can be displayed.

Sequence of operations

"AUTO"/"MDI"/"Jog" mode is selected in the "Machine" operating area.

The "Spindle" window appears on the screen.

The window contains the spindle setpoint and actual value, position, setting of spindle override switch and spindle output.




Using the "Page" keys you can scroll up and down to display other spindles if any are configured.

Additional notes

- The "Spindle" window is displayed only if at least one spindle is configured.
- If a master spindle is available it is displayed automatically in the spindle window, even if it is not meant to be the first spindle.

SW 6.2 or later:

The following spindle states are output as symbols for the spindles displayed in the spindle window:

- "Spindle stop" 
- "Spindle not enabled" 
- "Direction of spindle rotation left" or "... right" 
(= spindle running)

4.2.8 Handwheel



Function

Using the "Handwheel" function, you can assign an axis to the handwheels and activate them.

Sequence of operations

"Jog" mode is selected in the "Machine" operating area.

The "Handwheel" window appears on the screen.

Position the cursor on the handwheel of your choice (1–3).

An axis identifier is suggested in the "Axis" field. You can select all the other axes by means of the "Toggle key". The settings are accepted immediately and an axis is assigned to each handwheel (1–3).

Each time you press the "Toggle button" on the "Active" field you activate or deactivate enabling of the handwheel in question. The settings become active immediately.

When you turn the handwheel, the assigned axis traverses by the number of increments that have been set for it (Inc keys).

Machine manufacturer

The machine manufacturer is responsible for the design of handwheels. Operation may therefore differ from the explanation above.

Please consult the documentation provided by the machine manufacturer!



4.2.9 Status of synchronized actions



Function

Status information (such as "enabled", "disabled", etc.) can be displayed here to support start-up of synchronized actions.

References: /PGA/, Programming Guide Advanced, Chapter "Synchronized Actions"



Sequence of operations

"AUTO"/"MDI"/"Jog" mode is selected in the "Machine" operating area.

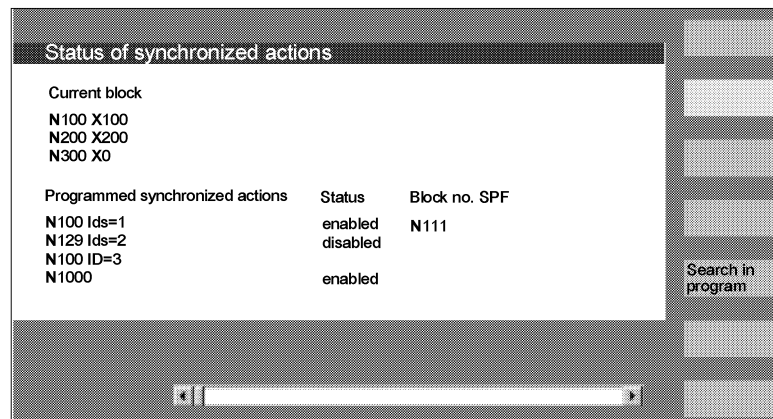
Press the ETC key followed by

the "Synchroniz. actions" softkey.

The display "Status of synchronized actions" appears.



Synchroniz.
actions



The display shows:

- Column "**Current block**":
Current section of selected parts program:
Preceding, current and following block

- Column "**Programmed synchronized actions**":
The programmed synchronized actions are listed block by block with block number
(static/modal actions are listed together with the number of the synchronized action)
- Column "**Status**"
 - " " (no information given)
the condition is checked in interpolation cycles.
 - "disabled"
LOCK was set from the PLC for the synchronized action
 - "active"
The action is currently running. If the instruction part of a programmed synchronized action starts a subroutine/cycle, the current block number of the cycle is additionally displayed in column "Block number SPF".



Search in
program



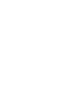
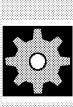
A window opens in which you must enter the system variable/
synchronized action of your choice.

The control system searches the current or an additionally specified
program for synchronized actions which match the current block and
synchronized action numbers.

If the control finds matching program blocks, the associated condition
and instruction parts are output with the other basic display data.

Press RECALL to return to the "Automatic" basic display.

4.2.10 Preset



Preset

Function

Resetting the actual value

A new position value is entered for the current axis positions.

Danger

After the actual value has been reset, none of the protection zones or software limit switches are operative! Only after another reference point approach are the protection zones and software limit switches active again.

The "Preset" function can be used to redefine the control zero in the machine coordinate system.

Preset values are applied to machine axes.

Axes do not move when "Preset" is active.

Machine manufacturer

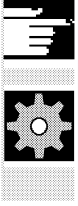
Please follow the machine manufacturer's instructions.

Sequence of operations

"Jog" mode is selected in the "Machine" operating area.

The "Preset" window appears on the screen.

Enter the new actual value, which must in future correspond to the current axis position, for each individual axis. By doing so, you are redefining the control zero in the MCS. When the control zero is redefined, the tool change point, for example, also changes.



Additional notes

Machine manufacturer

The "Preset" function can be disabled by means of protection levels (keyswitch position).

4.2.11 Set actual value



Set actual
value

1

9



Function

The "Set actual value" function is now available via display machine data 9422 as a functional alternative to the existing "Preset" function. The "Set actual value" function can be used only if the control system is operating in the workpiece coordinate system.

The functions are stored under the same softkey.

This function sets the workpiece coordinate system to a defined actual coordinate. The resultant offset between the old actual value and the newly entered WCS actual value is computed in the system frame or basic frame defined by the machine manufacturer.

Machine manufacturer

Please follow the machine manufacturer's instructions.

For configuring references, please see

/IAM/ Installation and Start-Up Guide, Section HMI/MMC, IM4

Sequence of operations

Precondition:

The function can only be invoked when G500 and Work (WCS) are active.

The new setpoint position of the axes in the workpiece coordinate system can be entered using "Set actual value" in the actual value window. When you transfer a value to the system by pressing "Input", the deviation from the current actual value is entered in the basic frame or system frame. The new actual value is displayed in the "Position" column.

4.2 General functions and displays

WCS	Position	
X	20.000	mm
Y	10.000	mm
Z	120.0	mm
U	0.000	grd
V	0.000	mm

Cancel

Using "Cancel", you can undo the entire offset entered up to now and exit the input screen.

4.2.12 Inch/metric switchover



Switch to
metric

Function

The control system can operate with the inch or the metric system of measurement. You can switch between the inch and metric systems of measurement in the "Machine" operating area. The control converts the values accordingly.

Machine manufacturer

The switchover is only possible if:

- The corresponding machine data have been set.
- All channels are in the Reset state.
- Axes are not traversing with JOG, DRF or PLC control.
- Constant grinding wheel peripheral speed (GWPS) is not active.

The display resolution for the inch system of measurement is defined by machine data.

Actions such as parts program start or mode change are disabled for the duration of the switchover.

Sequence of operations

"AUTO"/"MDI"/"Jog" mode is selected in the "Machine" operating area. Press the ETC key followed by

the softkey "Switch to metric".

When the system of measurement is changed, all length-related parameters are automatically converted to the new system of measurement from the perspective of the operator.

If the switchover cannot be performed, this is indicated by a message in the user interface.

References: /FB/, G2 Section "Metric/inch system of units"

**Warning**

If several NCUs are linked by an NCU link, then the switchover affects all linked NCUs. If the requirements for switchover to one of the linked NCUs are not met, the switchover is performed on none of the NCUs. Where an NCU link exists, it is assumed that cross-NCU interpolation is intended. This can only produce correct results when a harmonized system of units is in use.

References: /FB/, B3, Distributed Systems

4.3 Reference point approach



Function

The "Ref" function ensures that the control and machine are synchronized after power on.

Before a reference point approach can be carried out, the axes must be located at positions (if necessary, traversed to these positions using the axis keys/handwheels) from where the machine reference point can be approached without collision.

If reference point approach is called from a parts program all axes can be traversed simultaneously.

Reference point approach can only be performed by machine axes. The actual value display does not match the real position of the axes when the control is switched on.

Caution

- If the axes are not safely positioned, you must traverse them to safe positions in "Jog" or "MDI" mode.
- You must follow the axis motions directly on the machine!
- Ignore the actual value display until the axes have been referenced!
- The software limit switches are not active!

Sequence of operations

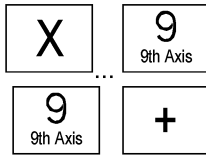
"Jog" or "MDI" is selected in the "Machine" operating area. The channel for reference point approach is selected.

The "Ref" machine function is selected.

Turning machine:

Press the "Axis keys".

4.3 Reference point approach



Milling machine:

Select the axis to be traversed and then press the "+" or "-" key.

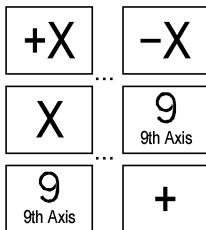
The selected axis moves to the reference point. The direction and sequence is defined by the machine manufacturer in the PLC program.

If you have pressed the wrong direction key, the action is not accepted and the axes do not move.

The display shows the reference point value.

No symbol is displayed for axes that are not referred to a reference point.

- This symbol appears for axes which must be referenced.
- This symbol is displayed next to the axis as soon as it has reached the reference point.



The axis, once started, can be stopped before it reaches the reference point.

Turning machine:

Press the "Axis keys".

Milling machine:

Select the axis to be traversed and then press the "+" or "-" key.

The selected axis moves to the reference point.



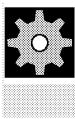
Caution

The machine is synchronized as soon as the reference point is reached. The actual value display is set to the reference point value. The display is the difference between the machine zero and the slide reference point. From now on path limits, such as software limit switches, are active.



You can end the function by selecting another operating mode ("Jog", "MDI" or "Automatic").

- All axes of a mode group can approach the reference point simultaneously (depending on the PLC program of the machine manufacturer).
- The feedrate override is active.



Machine manufacturer

Your machine manufacturer will instruct you how to select axes if you intend to enter more than nine.



Additional notes

The sequence in which axes must be referenced can be defined by the machine manufacturer.

The NC cannot be started in automatic mode until all axes with a defined reference point (see machine data MD) have actually reached this point.

4.4 Jog mode

4.4.1 Function and basic display



Function

You can perform the following tasks in Manual mode:

1. Synchronize the measuring system of the control with the machine (reference point approach).
2. Set up the machine, i.e. you can trigger manually-controlled motions on the machine using the appropriate keys and handwheels on the machine control panel.
3. Trigger manually-controlled motions on the machine using the appropriate keys and handwheels on the machine control panel while the parts program is interrupted.


The following basic display "Jog" is displayed when you press the "Area switchover key" and the "JOG" key.



The "Jog" basic display contains values relating to position, feedrate, spindle and tool.

Machine		Jog	
<input checked="" type="checkbox"/> Channel reset	Program aborted		G. fct. + transf.
			Display aux. funct.
MCS	Position	Repos offset	Master spindle S1
-X	90.646 mm	0.000	Act. + 0.000 rpm
+Y	113.385 mm	0.000	Set 0.000 rpm
+Z	109.131 mm	0.000	Pos 0.000 deg.
+V	0.000 mm	0.000	0.000 %
			Power [%]
			Feedrate mm/min
			Act. 7000.000 0.000%
			Set 7000.000
			Tool
			Preselected tool: ◀
			▶ G0 G91
			Act. val. WCS
Preset		Scratch	Handwheel INC

Explanation of the basic display Jog

 WCS
 + X2*
 Y2*
 Z2*

Displays the addresses of the existing axes with the machine axis identifier (MCS) or with the geometric axis identifier (WCS). (see also Section "Toggling between Machine/Workpiece Coordinate Systems (MCS/WCS)")



Machine manufacturer

Machine data are used to define whether the programmed frames are to be calculated when displaying the WCS (settable zero system display). Please consult the documentation provided by the machine manufacturer!

- If the axis identifier can only be displayed in an abbreviated form it is replaced by the character *.
- If you traverse an axis in the positive (+) or negative (–) direction, a plus or minus sign is shown in the relevant field.
The axis is in position if neither + nor – is displayed in the position display.



Position

0.0
0.1
–0.1
1.1
0.0

The actual position of each configured axis in the MCS or WCS is displayed in these fields.

The sign is only displayed for negative values.


Repos. offset

0.0
0.1
–0.1
1.1
0.0

If the axes are traversed in the "Program interrupted" status in "Jog" mode, the path traversed by every axis with respect to the point of interruption is displayed in the Repos offset.

Spindle

Spindle window (if spindle is available)

Displays the set and actual value of the spindle speed, the position of the spindle, the position of the spindle override switch,  and the spindle power.

Nibbling


The spindle window is replaced by a nibble window if the "Nibbling" technology option is set.

The active function, if any, appears at the top left in the window header:


- PON Punching ON
- SON Nibbling ON
- SPOF Punching/Nibbling OFF

Feedrate

Feedrate window

Display of the setpoint and actual value of the feedrate as well as the position of the feedrate override switch  (in %). The actual setpoint to be traversed is dependent on the override switch. The value of the rapid traverse override is displayed when G00 (rapid traverse movement) is programmed.

SW 6.2 or later:

If the feedrate is not enabled, the symbol "Feedrate not enabled"  is output in the feedrate window. (FST is not indicated in the "program control display" field).

Tool

Tool window

Display of active tool offset (e.g. D1), tool currently in use (T no.), preselected tool (on milling machines) plus currently active motion commands (e.g. G01, SPLINE, ...) or tool radius compensation not active (e.g. G40).

The following values are displayed:

- "Path segment" and "Number of strokes" are displayed in reverse video if you have used "Length of path segment" or "Number of path segments" when programming automatic block segmentation.
- The "Delay time" is not displayed unless you have programmed "Punching with delay time".



Zoom
act. val.

Increases the size of the actual-value display.

4.4.2 Traversing axes

Traversing speed

The initial settings for traversing velocity and feed mode are stored in setting data for JOG mode.

Traversing velocities are defined by the machine manufacturer. The default setting for the feedrate is mm/min.

See operating area "Parameters/Setting data/Jog data".



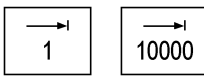
Sequence of operations

"Jog" mode is selected in the "Machine" operating area.

Traverse axes

Using the "Inc" (increment) function, manually traverse the selected axis in preset increments in the appropriate direction by pressing an "Axis key" repeatedly:

- [.] Variable increment can be set via softkey "Inc" (see Subsection 3.2.5).
- [1], [10], ..., [10000] fixed increment



Turning machine:

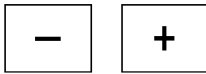
Press the "Axis keys".

If necessary, set the speed with the override selector.

If you press "Rapid traverse override" at the same time, you can traverse the axis in rapid traverse mode.

Milling machine:

Select the axis to be traversed and



then press the "-" or "+" key.

Feedrate and rapid traverse override switches can be operative.

One or several axes can be selected at the same time (depending on PLC program).

Additional notes

- After you have switched on the control, you can move axes into the limit range of the machine since their reference points have not yet been approached. In doing so, you may activate emergency limit switches.
- The software limit switches and the working area limitation are not yet operative!
- The feedrate enable must be set (FST display must not light up in the program control display).

Machine manufacturer

If it is not meaningful to traverse several axes simultaneously, the machine manufacturer must provide an interlock in the PLC program.

4.4.3 Inc: Incremental dimension



Function

The "Inc" (Increment) function can be used to enter a settable increment value for Inc variable traversing.

Sequence of operations

"Jog" mode is selected in the "Machine" operating area.

The "Increment" window appears on the screen.

Enter the desired increment size.





Press this key in manual mode together with the "Axis key" to traverse the axis in increments of the size set above in the appropriate direction (see also Section "Traversing axes").

Increment keys with preset increment sizes are temporarily inoperative.

4.4.4 Repos (repositioning)



Function

After a program interruption in automatic mode (e.g. to take a measurement on the workpiece or to compensate tool wear values or after tool breakage), you can retract the tool manually from the contour after selecting "Jog" mode. In such cases, the control stores the coordinates of the point of interruption and displays the path distances traversed by the axes in "Jog" mode as a "Repos" offset (Repos = repositioning) in the actual value window.

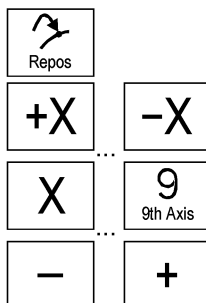
"Repos" offsets can be displayed in the machine coordinate system (MCS) or workpiece coordinate system (WCS).



Sequence of operations

"Jog" mode is selected in the "Machine" operating area.
The axes have been moved away from the point of interruption.

Select machine function "Repos".



Turning machine:

Press the "Axis keys".

Milling machine:

Select the axis to be traversed and then press the "-" or "+" key.

It is not possible to overtravel the point of interruption.
The feedrate override switch is active.



**Warning**

The rapid traverse override switch is active.
Non-compensated Repos offsets are compensated with programmed feedrate and linear interpolation on switchover to Automatic mode and selection of NC start.

4.4.5 SI (Safety Integrated): User confirmation**Function**

If the option "user enabling" is installed on the NC, you must enable or disable the function depending on the keyswitch position in operating mode "Approach reference point".



User enabling can only be granted if keyswitch position 3 or higher is selected for the keyswitch position.

The displayed values are always referred to the machine coordinate system (MCS).

**Sequence of operations**

"Jog Ref" mode is selected in the "Machine" operating area.
The channel to be referenced is already selected.

User con-
firmation

Press softkey "User confirmation".
The "Confirm machine positions" window is opened.
The machine axes in the MCS are displayed, together with the current position and a checkbox for activating/deactivating the user confirmation.



Machine axes that do not need to be referenced are displayed in reverse video with their current position, but without a checkbox.

Position the cursor on the desired machine axis.



Activate or deactivate user agreement for the selected machine axis by means of the "Toggle key".



Check whether the axis is referenced. If not, error message "Please reference axis first" is output. User agreement cannot be activated for the axis until it has been referenced.

Additional notes

The function user enabling is only provided if user enabling is required for at least one axis of the channel.

For more detailed information please consult the /FBSI/, SINUMERIK Safety Integrated, Description of Functions.

4.4.6 Scratching (SW 6 or later)



Scratch

Notes

Depending on the machine data configuration described in /IAM/, IM4, two scratching modes are possible. This subsection describes the variant for MMC and NCK in SW version 6; the following subsection describes the procedure for configurations which correspond to SW 5 or earlier.

Function

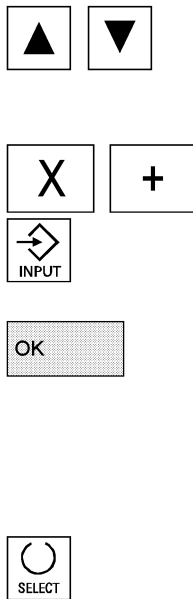
You can determine the zero offset by "scratching" the workpiece, taking an (active) tool and, if necessary, the basic offset into account. A window is available for the "Scratching" function.

Sequence of operations

1. Press softkey "Scratch":
 - The active level is displayed and can be changed.
 - The active NV is displayed and can be changed.
 - The active tool is displayed. No tool is displayed if none is active (message).

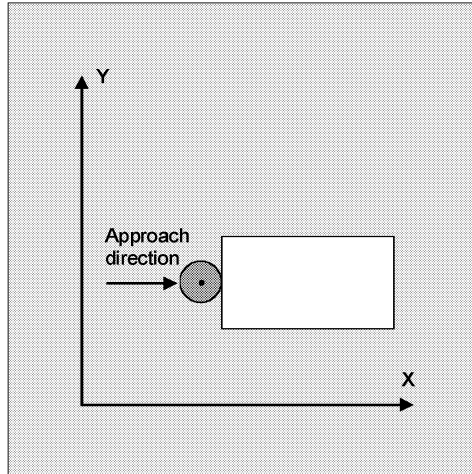
Machine	CHAN1	JOG	\MPF.DIR TEST_ULI.MPF	57 1523216
Channel reset				
Program aborted		ROV		
UCS	Position	Repos offset	Spindle	SZ
X	200.000	mm 0.000	Act. 0.000 rpm	
Y	25.800	mm 0.000	Set 0.000 rpm	
Z	0.000	mm 0.000	Pos 0.000 deg	
B	0.000	deg 0.000	100.000 %	
C	0.000	mm 0.000	Power [%]	
Scratch				
Plane	G17	T no.	3	
Zero offset	G500	Cut edge	1	
Axis	Offset	Setpt. pos.	Geo. + wear	Base
X	234.000	-234.000	+ R 0.000	+ L3 0.000 mm
Y	384.000	-384.000	+ R 0.000	+ L2 0.000 mm
Z	0.000	0.000	L1	L1 mm
C		0.000		mm
				Abort
				OK

Meanings of columns in the "Scratch" window:



- "Offset": Current value of offset to be determined. The coarse offset is displayed; the fine offset is taken into account and remains valid.
 - "Set position": Input of subsequent setpoint position for scratched edge.
 - "Approach direction": selection fields for positive/negative approach direction
2. Use the cursor to select the first axis to be moved in the "Scratch" display.
Selected axes are automatically marked in parallel in the actual value window.
 3. Move the axis up to the workpiece, enter the chosen setpoint position (e.g. "0") and press the "Input" key; the offset is then calculated.
Repeat the process for other axes.
 4. Press "OK" to transfer all values to the selected ZO. The offset is always calculated in relation to the current workpiece coordinate system (WCS).
 5. To ensure that allowance is made in the tool geometry (or the tool base dimension), position the cursor in the "Approach direction" column on the axis to be altered and press the "Toggle key" so that the desired approach direction is visible.

Example 1:



Set "+R" with toggle key.

The figure below shows how system frame \$P_SETFR is displayed as the line "Set work offset".

Parameter	CHAN1	Jog	MPF0	
<input checked="" type="checkbox"/> Channel reset			Program aborted	Axes +
Change active work offsets and compensations				Axes -
Set zero	Coarse	X [mm]	Y [mm]	Z [mm]
	Fine	0.000	0.000	0.000
Ext. WO frame	Coarse	0.000	0.000	0.000
	Fine	0.000	0.000	0.000
Tool holder	Coarse	0.000	0.000	0.000
	Fine	0.000	0.000	0.000
1.global base WO	Coarse	0.000	0.000	0.000
	Fine	0.000	0.000	0.000
1.chan-spec. base WO	Coarse	0.000	0.000	0.000
	Fine	0.000	0.000	0.000
Tool reference	Coarse	0.000	0.000	0.000
	Fine	0.000	0.000	0.000
Work pc. ref.	Coarse	0.000	0.000	0.000
	Fine	0.000	0.000	0.000
Frame cycles	Coarse	0.000	0.000	0.000
	Fine	0.000	0.000	0.000
Decimal no.inp.: only nos, "+", "-" and "~" allowed				
Tool compens.	R variables	Setting data	Work offset	User data
				Active WO + compens.

4.4.7 Scratching (SW 5 or earlier)



Scratch



Notes

Depending on the machine data configuration described in /IAM/, IM4, two scratching modes are possible. This subsection describes the variant for MMC and NCK in SW version 5; the previous subsection describes the procedure for configurations which correspond to SW 6 or later.

Function

You can determine the zero offset by "scratching" the workpiece, taking an (active) tool and, if necessary, the basic offset into account. A window is available for the "Scratching" function.

Sequence of operations

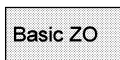
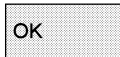
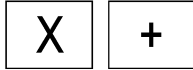
1. Press softkey "Scratch":
 - The active plane is selected (G17).
 - The active ZO is selected. If no ZO is currently active, then the first ZO (G54) is selected.
 - The active tool is displayed. No tool is displayed if none is active (message).

Meanings of columns in the "Scratch" window:

- "Offset": Current value of offset to be determined. The coarse offset is displayed; the fine offset is taken into account and remains valid.
- "Set position": Input of subsequent setpoint position for scratched edge.
- "Geo + Wear": Selection fields for tool offsets (geometry + wear)
- "Base": Display of effective tool base dimension

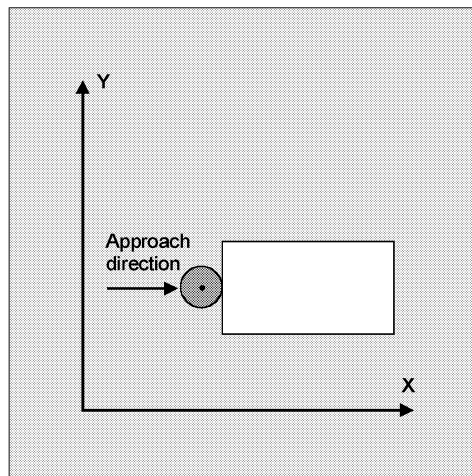
2. Use the cursor to select the first axis to be moved in the "Scratch" display.

Selected axes are automatically marked in parallel in the actual value window.



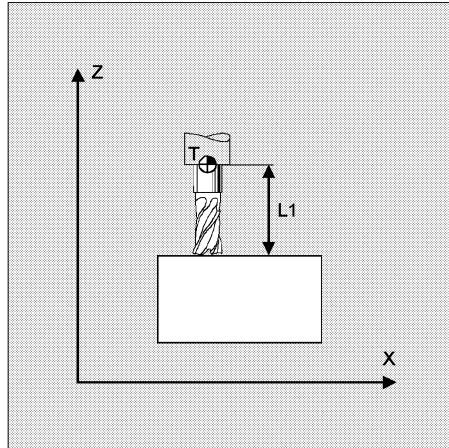
3. Move the axis up to the workpiece, enter the chosen setpoint position (e.g. "0") and press the "Input" key; the offset is then calculated.
Repeat the process for other axes.
4. Press "OK" to transfer all values to the selected ZO. The offset is always calculated in relation to the current workpiece coordinate system (WCS).
5. Press the softkey "Basic ZO" to select the values for the basic frame (G500 is automatically displayed in the "Zero offset" field) or enter G500 in the field "Zero offset".
6. To ensure that allowance is made in the tool geometry (or the tool base dimension), position the cursor in the "Geometry + Wear" column on the axis to be altered and press the "Toggle key" to define how the tool offset must be included in the calculation.

Example 1:



Set "+R" with toggle key.

Example 2:



Set "-L1" with toggle key.

There are three different variants depending on the system configuration:

- Tool selection with D No. only (flat D No.)
- Without tool management
- With tool management

Correspondingly, the "scratching" screen displays

- only the tool edge (D No.O),
- T number and tool edge and
- tool name and Duplo number.

Additional notes

- The "Scratching" function is available if the following setting data = 0 and MD 28082: MM_SYSTEM_FRAME_MASK = 0.
`$SC_MIRROR_TOOL_LENGTH`
`$SC_MIRROR_TOOL_WEAR`
`$SC_WEAR_SIGN_CUTPOS`
`$SC_WEAR_SIGN`
- When a basic offset is active and contains rotation or mirroring, the "scratching" function cannot be used for G54 to G599.
- Any rotation, mirroring or scale modification included in the offset to be measured remains valid.

4.4.8 Display of system frames

If system frames have been activated via MD, they can be displayed via the operating area Parameters, Active ZO + Override, Details.

The following assignment exists:

\$P_SETFR	System frame for set actual value, scratching
\$P_EXTFR	System frame for external zero offset
\$P_PARTFR	System frame for TCARR and PAROT
\$P_TOOLFR	System frame for TOROT and TOFRAME
\$P_WPFR	System frame for workpieces
\$P_CYCFR	System frame for cycles

The (coarse and fine) offsets set via the frames as well as the rotation and mirroring frames also defined via the frames are displayed. They are displayed according to their position in the frame sequence.

You can choose between the system frames and basic frames by scrolling through the desired section.

The following figure shows sample screen contents

Parameter	CHAN1	Jog	MPF0	
Channel reset			Program aborted	Axis +
Change active work offsets and compensations				Axis -
		X [mm]	Y [mm]	Z [mm]
1.chan-spec. base WO	Coarse	0.000	0.000	0.000
	Fine	0.000	0.000	0.000
Tool reference	Coarse	0.000	0.000	0.000
	Fine	0.000	0.000	0.000
Work pc. ref.	Coarse	0.000	0.000	0.000
	Fine	0.000	0.000	0.000
Frame cycles	Coarse	0.000	0.000	0.000
	Fine	0.000	0.000	0.000
Prog. WO	G58/TRANS	0.000	0.000	0.000
	G59/ATRANS	0.000	0.000	0.000
Act. tool	T no.	0	D no.	0 Plane G17
Lengths	Geometry			
	Wear			
	Base			
Radius	Geometry			
	Wear			
Decimal no.inp.: only nos. "*" and "+" allowed				
Tool compens.	R variables	Setting data	Work offset	User data
				Active WO + compens.



Additional notes

SW 6.2 or later:

Set work offset	corresponds to \$P_SETFR
Ext. WO frame	corresponds to \$P_EXTFR
Toolholder	corresponds to \$P_PARTFR)
Tool reference	corresponds to \$P_TOOLFR)

SW 6.3 and higher:

Workpiece ref.	corresponds to \$P_WPFR)
Cycles	corresponds to \$P_CYCFR)

See also the figure at the end of 4.4.6

4.5 MDI mode

4.5.1 Function and basic display

In "MDI" (Manual Data Input) mode, you can write parts programs block by block and execute them. You can transfer the required motions as single parts program blocks to the control using the operator panel.

The control processes the entered program blocks when you press the "NC Start" key.



Caution

The same safety interlocks must be applied as used in fully automatic operation. The same preconditions must be fulfilled as for fully automatic operation.

The automatic functions (traverse blocks) are active in "MDI" mode.

Teach In

The functions associated with "Jog" are active in submode "Teach In" and can be accessed via an MCP key. You can therefore create and store a program in the input and manual modes by alternating between "MDI" and "Teach In".

You can use the editor to edit the program blocks in the MDI window. You can view blocks that have already been processed by paging upwards.

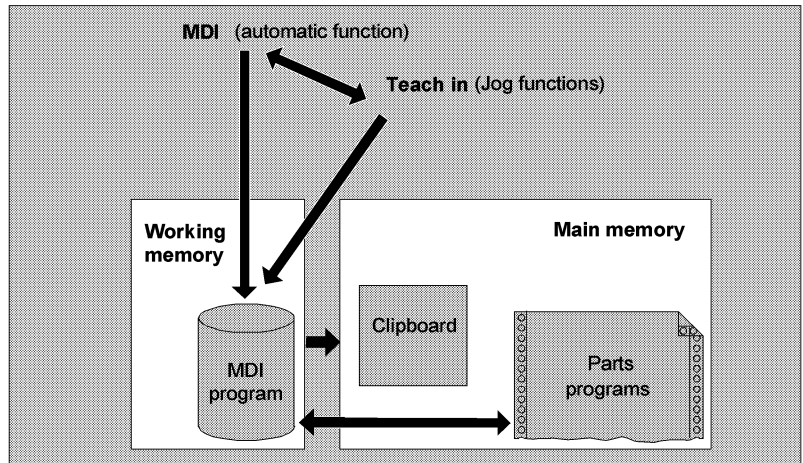
Blocks that have already been executed can only be edited in the Reset state.

Further blocks can be added with "Input".

Blocks can be input and executed only with the channel in the "Channel Reset" or "Channel interrupted" state.

The program generated in MDI mode can be stored

- in the parts programs directory (MPF.DIR)



In the basic display "MDI", position, feedrate, spindle and tool values and the Contents of the MDI buffer are displayed.

Machine	CHAN1	MDI	ISYF.DIR OSTORE1.SYF
Channel active		Program running	
G fct. transf.			
Auxiliary functions			
MCS Position		Dist.-to-go	Master spindle S1
-X	90.301 mm	9.699	Act. + 0.000 rpm
+Y	80.081 mm	9.919	Set 0.000 rpm
+Z	0.000 mm	0.000	Pos 0.000 deg.
+V	0.000 mm	0.000	0.000 %
Power [%]		[Progress bar]	
MDA- program		Feedrate mm/min	
N51 Y0 H55=99		Act. 7000.000 0.000 %	
Y120 F7000		Set 7000.000	
Tool		Act. val. WCS	
preselected tool:		Delete MDI buffer	
G0 G91		MDI file fct.	
Program control		Handwheel Editor function	

Explanation of "MDI" basic display

Like the Jog basic display, the actual value window, spindle window, feedrate window and tool window are output.



Store MDI
program

Editor
functions

MDI
file fct.

Delete MDI
prog.

Softkey "Store MDI program" copies the MDI program to the clipboard.

Press this softkey to access vertical softkeys providing editing functions:

Overwrite, Mark, Copy, Paste, Delete, Find, Find Next, Position.
You can save the MDI buffer contents or copy a parts program to the MDI buffer for editing in MDI or Teach In.

Vertical softkeys

Like the Jog display, the softkeys "G func.+Transf.", "auxiliary functions" and "spindle" (if spindle available) are displayed.

The contents of the MDI program in the NC are erased.

4.5.2 Save program, file function



Function

You can use the MDI file function to

- save the contents of the MDI program to a file or
- read a parts program to the MDI buffer to modify it in MDI or Teach In mode.



Sequence of operations

"MDI" mode is selected in the "Machine" operating area.

The program overview of "Parts programs" is opened, a dialog window also appears.

The new file is then displayed in the program overview.

The program overview of parts programs is displayed. From the list of programs select the one that you wish to copy to the MDI buffer with "Read in MDI". This can now be modified accordingly.

The system will ask you to enter a name for the file to be saved to the MDI buffer.


The program is saved/stored as a parts program (MPF) under the name you have entered in the Parts Programs directory.



MDI
file fct.

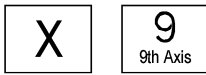
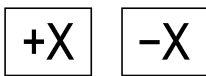


Read in
MDI



Store MDI
program

4.5.3 Teach In

**Function**

With the function "Teach In" parts programs (main programs and subroutines) for motion sequences or simple workpieces can be created, modified and executed by approaching and then storing positions in combination with the function "MDI".

There are two possible methods of writing programs with "Teach In" and "MDI":

1. Manual positioning.
2. Manual input of coordinates and additional information.

Both entry of blocks (input, delete, insert) and automatic insertion by manual approach of positions are possible only at points that have not yet been executed.

1. Manual positioning**Sequence of operations**

"MDI" mode is selected in the "Machine" operating area.

Submode "Teach In" is selected.

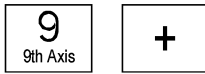
After selection of the function the cursor is located in the first blank line of the "Teach-In program" window.

Turning machine:

Press the "Axis keys".

Milling machine:

Select the axis to be traversed



then press the "+" or "-" key.

The axis name and the path being traversed are displayed continuously on the screen in the "Teach In program" window. The axis name and axis positions are transferred to the MDI program as values referred to the workpiece coordinate system.

Save position values:

Enter the additional functions (e.g. feedrates, miscellaneous functions, etc.) in the program (if permitted).

Using MDI file functions, you can store the MDI program in the "Parts programs" directory (MPF.DIR).

Using the edit functions of the vertical softkeys, you can execute the following program block functions: Overwrite, Mark, Copy, Paste, Delete, Find, Find Next and Position.



When you switch to "MDI" mode and press "NC Start", the corresponding traversing motions and additional function entries are executed as program blocks. While the program is being processed, the blocks traversed by the NC are displayed in the "Current block" window.

2. Manual input of coordinates

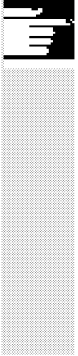
Sequence of operations

"MDI" mode is selected in the "Machine" operating area.

Save position values/additional functions:

Transfer the coordinates of the traversing positions plus any additional functions (preparatory functions, auxiliary functions, etc.) in the program by entering them in the "MDI program" window.





Additional notes

- Any change to the zero offset will cause the axis to carry out compensatory movements when you press NC Start.
- In the case of G64, the end point response will differ when the parts program is executed in "Automatic" mode.
- All the G functions can be used.
- When using special kinematics or robots, the positions of the STAT machine and rotary axis TU are displayed in addition to the positions in the workpiece coordinate system.

4.6 Automatic mode

4.6.1 Function and basic display

You can execute parts programs fully automatically in "Automatic" mode, i.e. this mode is normally used for machining parts.

Preconditions

The following conditions must be fulfilled before you can execute parts programs:

- You have synchronized the measuring system of the control with the machine (reference point approach).
- You have already loaded the parts program to be executed to the control system.
- You have checked the required offset values or entered them, i.e. zero offsets or tool offsets.
- The requisite safety interlocks are already active.

The "Automatic" basic display contains values relating to position, feedrate, spindle and tool as well as the block currently being processed or program pointer.

Machine		Auto		WKS.DIRTEST.MPF	
Channel reset		Program aborted		G fct. transf.	
MCS Position		Dist. -to-go		Master spindle S1	
-X	53.761 mm	-25.761		Act. + 0.000 rpm	Spindles
+Y	30.000 mm	0.000		Set 0.000 rpm	
+Z	112.704 mm	-112.704		Pos 0.000 deg.	Axis feedrate
+	0.000 mm	0.000		0.000 %	
Current block TEST.MPF		Feedrate mm/min		Program blocks	
N51 Y0 H55=99		Act. 7000.000 0.000 %		Zoom act. val.	
Y120 F7000		Set 7000.000		Act. val. WCS	
Tool		preselected tool:		Program level	
G0		G91		Program overview	
Over-store	DRF offset	Program control	Block search	Handwheel	

Explanation of basic display

Like the Jog display, this display contains actual value, spindle window, feedrate and tool windows.



Program overview

Horizontal softkeys

The workpiece or program overview is displayed. In this area it is possible to select programs for execution.

The memory space available on the hard disk and in the NCU is displayed in the footer.

Current block

Vertical softkeys

The "Current block" window is also displayed.

The current block is highlighted while the program is running.

The name of the program to which the blocks on the screen belong is output in the window header.

Program level

When you press softkey "Program level", the window headed "Program level" appears instead of the "Current block" window. The program nesting depth (P = number of passes) is displayed.

Program level

or

Current block

In program operation it is possible to toggle between the "Program level" and "Current block" displays.

Program blocks

Seven program blocks of the current program are displayed together with the current position in the parts program.

Unlike with function "Current block", the program created by the programmer is displayed.

Additional notes

The other softkeys are described in the sections below.

4.6.2 Program overview



Program
overview

Selection

Change
enable

Workpiece

Function

After selection of a workpiece or program overview, individual workpieces or programs can be enabled or disabled for execution.

Sequence of operations

"AUTO" is selected in the "Machine" operating area.
The appropriate channel is selected.
The channel is in reset state.
The workpiece/program to be selected is in the memory.

An overview of all workpiece directories/programs that exist is displayed.

Position the cursor on the desired workpiece/program.

Vertical softkeys

Select the workpiece/program for execution:

The name of the selected workpiece is displayed on the screen in field "Program name" at the top. The program is also loaded and selected for processing.

The selected workpiece/program is enabled (X) or disabled (no "X").

A selection can only be made when the workpiece/program is enabled.

Explanation of other softkeys:

All programs of a particular type are displayed via the horizontal softkeys:

Display of all workpiece directories

Parts programs

Display of all existing parts programs

Sub-routines

Display of all existing subroutines

Standard cycles

Display of all existing standard cycles

User cycles

Display of all existing user cycles

Manufacturer cycles

Display of all existing manufacturer cycles

4.6.3 Loading and unloading the workpiece/parts program



Function

Workpieces and parts programs can be stored in the NC memory ("Load") and erased from this memory again ("Unload") after they have been executed.

In this way, it is possible to prevent the NC memory from becoming unnecessarily overloaded because programs that are no longer required can be erased immediately.



Sequence of operations

The "Machine" operating area is selected.

The "Program overview" is open.
The vertical softkey bar changes.

Program overview

Position the cursor in the directory tree on the workpiece/parts program that you wish to load.

Load
HD->NC

The selected workpiece/parts program is loaded from the hard disk into the memory of the NC.

It is erased from the hard disk.

Enable

If the enable command (X) is set, the workpiece can be machined.



Unload
NC->HD

The highlighted workpiece/parts program is loaded from the NC memory to the hard disk.

The workpiece/program is erased from the NC memory.

Execute fr.
hard disk

If a program requires more memory than the NC can provide for execution, then it can be executed from the hard disk without loading it completely into the NC.



Additional notes

Workpieces/parts programs that have been loaded to the NC memory are marked with ("X") in the "Loaded" column in the workpiece/program overview.

4.6.4 Log: Loading list of the programs



Function

- Error list:
Display of previously loaded programs if errors have occurred during loading.



Program
overview



Log

Sequence of operations

The "Machine" operating area is selected.

The "Program overview" is open.

The softkey bars change.

Select softkey "Log".

The "Job log for program overview" is opened.

4.6.5 Executing programs from the hard disk



Function

If a program requires more memory than the NC can provide for execution, then it can be loaded continuously from the hard disk.



Sequence of operations

The "Machine" operating area is selected.

Program
overview

The "Program overview" is open.

The vertical softkey bar changes. Select the program you wish to execute with the cursor (enable (x) must be set).

Execute fr.
hard disk

The highlighted program is loaded continuously to the NC memory during execution.

↕
Cycle Start

Press "NC Start"Ⓚ



The program remains stored on the hard disk while the "Execute from hard disk" function is in progress.

The "External programs" window is opened.

The "Status" field indicates what percentage of the program has been loaded from the hard disk.



Additional notes

Programs which are executed from the hard disk are marked with "EXT" in the program overview.

The identifier "EXT" does not disappear until a different parts program is selected in this channel.

SW 6.2 or later

If the programs are executed from the hard disk, the 3-block display (softkey program run) or the larger multi-block display (softkey program blocks) can be selected during automatic operation. The setting made applies for the **single block** or **NC stop** status.

4.6.6 Access to external network drive



Function

With the SINDNC software, you can link your control to external network drives or other computers. This makes it possible, for example, to execute parts programs from other computers.

Precondition:

- The SINDNC software is installed.
- The computer or drive which you want to link up to is accessible/enabled.
- A connection to the computer/drive has been established.
- The softkeys for selecting the drive/computer link have been configured in the machine data, see /IAM/ HMI/MMC, Installation and Start-Up Guide, IM4, Section 5.3 Data Transmission
- Separate display machine data which define the access rights to external drives exist for the Machine and Program operating areas, see /IAM/, Installation and Start-Up Guide HMI/MMC, IM4, Section 5.5.



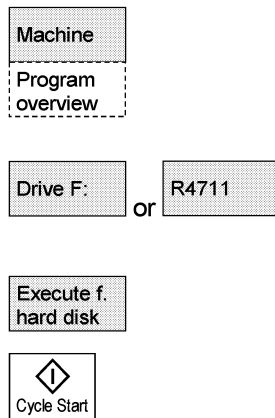
Sequence of operations

You can access the softkeys (configured) for the external drive or computer in the Machine operating area via the softkey "Program overview" and the "Etc." key. Horizontal softkeys 1 to 4 are reserved for this. The vertical softkey bar changes.

When you press a softkey, e.g. "Drive F:" or "R4711" the Explorer appears on the screen with the data of the external drive, e.g. "Drive F" or of computer "R4711".

Select the program that you want to execute with the cursor and then press the softkey "Execute from hard disk".

The program starts when you press "NC Start".





Softkey assignment SW 6.2 or later

Press the "Program overview" softkey in the Machine (and Program) operating area to access the first three softkeys configured for external network drives. If four softkeys have been configured, press the "ETC" key to access the fourth key on the continuation of the bar. If no softkeys are configured for external drives, the softkeys "Standard cycles", "User cycles" and "Manufacturer cycles" can be found in the appropriate places in the horizontal softkey bar.

Program call from the parts program: EXTCALL

It is possible to access files on network drives from a parts program using the command EXTCALL.

The programmer can define the source directory in the part program with setting data SD 42700: EXT_PROG_PATH and the file name for the subprogram to be loaded with the command EXTCALL.



The following secondary conditions must be taken into account with EXTCALL calls:

- Only files marked MPF or SPF can be called from a network drive using EXTCALL.
- DOS naming conventions must be used for the files and paths: A max. of 25 characters for the name, 3 characters for the extension.
- A program is found on a network drive with the command EXTCALL if
 - the search path points to the network drive or a directory on it with SD 42700: EXT_PROG_PATH. The program must be stored directly on that level, no subdirectories are searched.
 - The program is defined directly in the EXTCALL call with a qualified path which can also point to a subdirectory of the network drive and is also located there.
 - The search path is not restricted with SD 42700: EXT_PROG_PATH.



For further information about the EXTCALL command, please refer to:
/PGA/ Programming Guide Advanced, Chapter 2.
/FB/, K1 Description of Functions "Processing programs from external"

4.6.7 Program editing



Function

As soon as a syntax error in the parts program is detected by the control, program processing is interrupted and the syntax error is displayed in the alarm line.

You can switch to the program editor and monitor the program run in a full-screen display. If an error occurs (Stop state), you can edit the program in the program editor.



Sequence of operations

"Auto" is selected in the "Machine" operating area.

It is not possible to overtravel the point of interruption. The feedrate override switch is active.

The program status is "Stopped" or "Reset".

Correct
program

The correction editor is displayed with this softkey, the softkey bars change, the softkey "Edit" is marked.

If an error occurs, the faulty block is highlighted and can be corrected.

Using editor functions "Overwrite", "Highlight block", "Insert block", "Go To..." and "Find...", you can edit the program in the correction block editor.

Program
level +

or

Program
level -

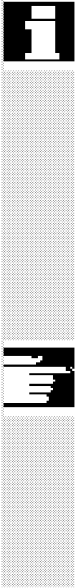
You can toggle between program levels using these softkeys.

Press "Level -" to view the program in which the faulty routine is called.

You can return to the program to be edited by pressing "Level +". This is only possible if the NC has a correction block.

⏏
Cycle Start

After you have corrected the error, you can continue the program run by pressing "NC Start".



- **NC Stop state:**
Only program lines that have not yet been executed can be edited.
- **Reset state:**
All program lines can be edited.

Note:

You cannot use the "Correct program" function if you are running a program from the hard disk.

Additional notes

If the error cannot be corrected in the "Channel aborted" state, an appropriate message is displayed when the "Correct block" softkey is pressed.

In this case, program execution must be aborted with "NC Reset". The parts program can then be edited under Programming.

4.6.8 Block search/setting the search destination



Function

The block search function allows you to run the parts program forward until you reach the block you require.

Four search types are available in SW 6.1 and higher:

1. With calculation on contour:

During block searches with calculation, the same calculations are performed as in normal program mode. The complete destination block is then executed analogously to normal program execution.

2. With calculation at block end point:

During block searches with calculation, the same calculations are performed as in normal program mode. The interpolation mode valid in the destination block is then applied to approach its end point or the next programmed position.

3. Without calculation:

No calculations are performed during the block search.

The values stored in the control remain the same as they were before the block search.

4. Multi-channel block search with calculation (SW 6.1 and higher):

This block search is called SERUPRO in "program test mode".

Single-channel applications are possible for interaction with other functions running in parallel. No axis movements are executed during the block search, however all auxiliary functions are output to the PLC.

The NC automatically starts the selected program in program test mode. If the NC reaches the specified target block in the current channel, it stops at the beginning of the target block and deselects program test mode again. The auxiliary functions of the target block are not output.

You can define the search destination

- by direct positioning or
- indirectly by entering a block number, a label, a program name or any character string.



References

For further information about block search, please refer to:

/FB/ K1, Mode Group, Channels, Program Operation



Block
search

Calculate
contour



Calc. block
end point

without
calculation

Sequence of operations

"AUTO" is selected in the "Machine" operating area.

The channel is in the reset state.

The program in which the block search is to be performed is selected.

Calls up the "Search position" function.

Position the cursor on the destination block.

The block search is started when you press one of the following three softkeys:

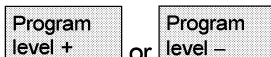
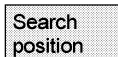
Block search start with calculation on contour

- When you press "NC Start", a compensatory axis motion is initiated between the current actual position and the new position determined by the block search.
- The current position is defined by the program status (all axis positions, active auxiliary functions) at the beginning of the selected block, i.e. after a block search the control stops at the final position of the NC block before the target of the search.

Block search start with calculation at block end point

Block search start without calculation

- If the destination block is found this block becomes the current block. HMI announces "Target of search found" and displays the target block in the current block display.
- Action blocks (e.g. collection of auxiliary functions) are output after NC Start.
- Alarm 10208 is output to indicate that operator interventions such as Overstore or Mode Change after JOG are permissible.
- When you press NC start the program is started and the process resumes at this destination block.



Block search can be aborted with Reset.

Define search destination in program editor:

The current selected program level is displayed.

Position the cursor bar on a destination block of your choice in the parts program.

You can toggle between program levels using these softkeys if necessary.

Defining the search destination in the search pointer:

Once the softkey "search pointer" has been pressed, a screen form with the program pointer is displayed.

It contains input fields for program name, search type (block number, text, ...) and search destination (content). The cursor is positioned in the input field for the "search type".

You must enter your selected search type for the search destination in input field "type".

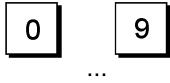
Information about the available search types is displayed in the dialog line.

The following search types can be entered:

Search type (= jump to ...)	Value in search type field
End of program	0
Block number	1
Jump label	2
Any character string	3
Program name	4
Line number	5

Different search types can be specified for different program levels.

You can enter your chosen search destination (according to search type) in the "Search destination" field.



Interrupt.
point

You can enter the corresponding number of program passes in the "P" field (pass counter).

Preset search destination to last program interruption point:

The search pointer is assigned the data of the last program interruption point.

4.6.9 Fast external block search



Function

The block search function allows you to run the parts program forward until you reach the block you require.

You can use the "External without calc." softkey in the "Search position" and "Search pointer" menus to start a fast block search in programs that are processed by an external device (local hard disk or network drive).

You can define the search destination

- by positioning the cursor bar directly on the target block or
- by specifying a block number or a line number.

Block search sequence:

Only programs and program parts required to reach the next specified search destination and subsequent program processing are transferred to the NC. This behavior applies to the function

- "Processing from external source" and
- when executing of EXTCALL statements.

Example:

The following search destinations are entered in the Block search menu:

Search destination 1

Search destination 2

Search destination 3

Programs

```

MAIN1.MPF
1      G90
2      X100 Y200 F2000
3      EXTCALL "SUBPRG1"
4      X200 Y400
5      EXTCALL "SUBPRG2"
6      EXTCALL "SUBPRG3"
7      X0 Y0
Search destination 1 8      EXTCALL "SUBPRG2"
9      X50
10     M30

SUBPRG2.SPF
1      X200 Y300
2      X400 Y500
3      EXTCALL "SUBPRG3"
.
.
Search destination 2 4000 EXTCALL "SUBPRG3"
.
.
5000 M17

SUBPRG3.SPF
1      X20 Y50
2      .
.
Search destination 3 2300 X100 Y450
.
.
5000 M17

```

A distinction is made between the following two cases for block search:

1. The main program MAIN1.MPF is in the NC, the subroutines SUBPRG1.SPF to SUBPRG3.SPF are processed externally.
2. The main program MAIN1.MPF and the subroutines SUBPRG1.SPF to SUBPRG3.SPF are processed externally.

Case 1

- Search destination 1: The NC does not consider the EXTCALL calls in lines 3, 5, and 6 of the main program MAIN1.MPF and jumps to the search destination Line 8.
- Search destination 2: The NC jumps to the search destination at line 4000 of external subprogram SUBPRG2.SPF; the previous statements in the external program are skipped.
- Search destination 3: The NC jumps to the search destination at line 2300 of external subprogram SUBPRG3.SPF; the previous statements are skipped, and the search stops.

On NC Start, the subroutine SUBPRG3.SPF is processed starting at the destination line.

Case 2

- Search destination 1: The NC jumps to the line 8 of the search destination.
- Search destination 2: see above.
- Search destination 3: see above.

On NC Start, the subroutine SUBPRG3.SPF is started and processed starting at the destination line.

Sequence of operations

"AUTO" is selected in the "Machine" operating area.

The channel is in the reset state.

The program in which the block search is to be performed is selected.

Switches to the "Search position" dialog.

Define search destination in program editor:

The current selected program level is displayed.

Searching via the "Search position" is only possible for the programs loaded in the NC, not for programs processed from the hard disk.

Position the cursor bar on a destination block of your choice in the parts program.

If an interrupt point is present in the NC, you can switch between the program levels.

Start of search without calculation for external programs.

Block
search

Search
position

Program
level +

or

Program
level -

External
wo. calc.



Search
pointer

External
wo. calc.

Defining the search destination in the search pointer:

Once the softkey "search pointer" has been pressed, a screen form with the program pointer is displayed.

It contains input fields for program name, search type (block number, text, ...) and search destination (content). The cursor is positioned in the input field for the "search type".

Type 1 (block number) and type 5 (line number) are possible types for the search destinations.

Start of search without calculation for external programs.

4.6.10 Start of search in Program test mode, multi-channel



Function

The NC is operated in Program test mode for this search, so that interactions between channel and synchronous actions and among several channels are possible within an NCU.

The "Block search in Program test mode" enables a program advance up to the desired point of the parts program with the following functions:

During "Block search in Program test mode", all auxiliary functions of the NC are output to the PLC and the parts program commands for channel coordination (WAITE, WAITM, WAITMC, Replace axis, Write variables, etc.) are carried out by the NC.

- This means that the PLC is updated during this block search and
- machining operations that involve several channels are processed correctly within the block search.

You can select the function via the "Prog. test Contour" softkey in the "Search position" and "Search pointer" menus.

You can define the search destination

- by positioning the cursor bar directly on the target block or
- using the input fields "Program name", "Search type" and "Search destination".

Additional notes

The function "Block search in program test mode" requires NC SW 6 or later.



Sequence of operations

Precondition:

- "AUTO" is selected as operating mode.
- The channels are in the Reset state.
- The program in which the block search is to be performed is selected.

The channel in which the destination block is searched for or the program pointer set is the destination channel. Other channels are started according to the setting in the "machine.ini" file.



Block search

Switches to the "Search position" dialog.

Search position

Define search destination in program editor:

The current selected program level is displayed.

Program level +

or

Program level +

Position the cursor bar on a destination block of your choice in the parts program.

You can toggle between program levels using these softkeys if necessary.

Prog. test contour

Start of search in Program test mode.

A dialog message appears as soon as all the channels involved in the current search have exited program test mode. This message indicates that Repos offsets may, depending on the particular parts program, have been produced in the channels involved during the block search and will be retracted by the NC through interpolation the next time it starts. You must acknowledge this message.

The repositioning offsets that occur may be traversed manually in JOG mode before continuing program processing with the "NC Start" key at the program position reached in the block search.

Search pointer

Defining the search destination in the search pointer:

Once the softkey "search pointer" has been pressed, a screen form with the program pointer is displayed.

It contains input fields for program name, search type (block number, text, ...) and search destination (content). The cursor is positioned in the input field for the "search type".



Prog. test
contour

Start of search in Program test mode.

A dialog message appears as soon as all the channels involved in the current search have exited program test mode, see "Search destination in program editor mode" above.

Any repositioning offsets (in the individual channels) can be retracted in sequence via manually controlled traversing movements in JOG mode, before program processing is continued via NC Start at the point in the program reached by the block search.

4.6.11 Overstore



Function

In "AUTO" mode you can overstore technological parameters (auxiliary functions, all programmable instructions ...) in the working memory of the NC. You can also enter and execute any NC block.

Sequence of operations

"AUTO" is selected in the "Machine" operating area.

Overstore with single block:

The program automatically stops at the next block boundary.

The "Overstore" window is opened.

In this window you can now enter the NC blocks that are to be processed.

The blocks you have entered are executed and displayed in the "Current block" window.

The "Overstore" window and associated softkeys are not displayed again until the control switches to the "Channel interrupted", "NC Stop" or "NC Reset" state.

More blocks can be added to the overstore buffer when these blocks have been processed.

Note:

After "Overstore" a subroutine with the content REPOSA is executed. The program is displayed for the operator.



Cycle Stop



Overstore



Cycle Start

**Overstore without single block:**

The basic menu for "AUTO" mode is selected.

Stop the program by pressing "NC Stop".

The "Overstore" window is opened.

In this window you can now enter the NC blocks that are to be processed.

The blocks you have entered are executed, the "Current block" window is opened and the softkeys disappear. The "Overstore" window and associated softkeys are not displayed again until the control switches to the "Channel interrupted", "NC Stop" or "NC Reset" state.

More blocks can be added (to the overstore buffer) when these blocks have been processed.

Additional notes

- You can close the window and exit the Overstore function by pressing the "Recall" key. You have now exited the Overstore function.
- You cannot change operating modes until you have deselected "Overstore" with the "Recall" key.
- The program that was selected before Overstore is now resumed in "AUTO" mode when you press "NC Start" again.
- Overstore does not alter the programs stored in the parts program memory.

4.6.12 Program control



Function

You can use this function to change a program sequence in "AUTO" and "MDI" modes. The following program control functions can be activated or deactivated:

DRY	Dry run feedrate	
ROV	Rapid traverse override	
M01	Programmed stop 1	
M Fct*	Programmed stop 2	HMI SW 6.3, NCK SW 6.4 and higher

DRF	DRF selection
PRT	Program test

SBL1	Single block with stop after each machine function
SBL2	Single block with stop after every block
SBL3	Stop in cycle

Fct* A function associated with M01 with stop at end of cycle.

With horizontal softkeys:

"Skip blocks": SKP blocks

up to 8 skip levels can be activated. Please follow the machine manufacturer's instructions.

"Block display": Block display modes during program execution either all blocks or only motion blocks are displayed.

The display is dependent on the machine data of the operator panel front in which access authorization can be set.

For further information, please refer to Section 2.5,

"Program control display"

References: /FB/, A2, Various Interface Signals, Chapter 4, and /FB/ K1, Mode Groups, Channels, Program Operation

DRY

Program control with DRY

When you select the DRY function "dry run feedrate" in "AUTOMATIC" mode, the feedrate entered in setting data SD 42100: DRY_RUN_FEED is applied in the program instead of the programmed feedrate.

SKP:
Activate skip levels

Blocks that cannot be executed can be skipped (8 skip levels). Blocks which are to be skipped are marked with an oblique "/" in front of the block number. The skip levels in the parts program are specified by "/0" or by "/" to "/7". You can activate/deactivate skip levels in the following window:

Program control	
SKP: Skip blocks	active
<input checked="" type="checkbox"/> <input checked="" type="checkbox"/> Skip /	<input checked="" type="checkbox"/> <input checked="" type="checkbox"/>
<input checked="" type="checkbox"/> Skip /1	<input checked="" type="checkbox"/>
<input checked="" type="checkbox"/> Skip /2	<input checked="" type="checkbox"/>
<input checked="" type="checkbox"/> Skip /2	<input type="checkbox"/>
<input checked="" type="checkbox"/> Skip /3	<input type="checkbox"/>
<input checked="" type="checkbox"/> Skip /4	<input type="checkbox"/>
<input checked="" type="checkbox"/> Skip /5	<input type="checkbox"/>
<input checked="" type="checkbox"/> Skip /6	<input type="checkbox"/>
<input checked="" type="checkbox"/> Skip /7	<input type="checkbox"/>

This program control window is displayed only if more than 1 skip level was set via machine data. Only the set skip levels are displayed. Changes to machine data are described in Section 9.1.

Display activated
skip levels

If more than one skip level is active, the right column indicates whether the PLC has transmitted the skip level selection to the NC.

This allows you to see from the top window which level

- is selected but not yet active,
- selected and active,
- deselected and active,
- deselected and not active.



Sequence of operations

"AUTO" or "MDI" mode is selected in the "Machine" operating area.

Press the horizontal softkey "Program control"

Program control

Another window pane appears in the "program control" window. The softkey menus change and the softkeys of the skip blocks and block display appear in the vertical menu.



Each actuation of the "toggle key" activates or deactivates the selected program control function.

Skip blocks, SW 6.3 and higher

The block display in software versions 6.3 and higher appears in an extended "program control" window pane and can be selected with the vertical softkey "Block display".

Skip blocks

Press the vertical softkey "Skip blocks"



The vertical softkey "Skip blocks" appears only if more than 1 skip level was set in MD 9423: MA_MAX_SKP_LEVEL. The levels to be skipped can only be changed when the control is in the Stop/Reset state.

Place the cursor at the required position.



Each actuation of the "toggle key" activates or deactivates the selected skip level in the "program control" window.

Additional notes

The entries influence the program control display (see Section 2.5). The program control display is output irrespective of the selected menu.

For further information about how to program skip levels, please see:

References: /PG/, Programming Guide, Fundamentals

Block display during program execution

The block display during program execution in software versions 6.3 and higher appears in an extended "program control" window pane and can be selected with the vertical softkey "Block display".



Block display

Press the vertical softkey "Block display"

Program execution modes

A separate "program control" window pane with the previous block display appears in the main screen. You can select the following while the program is running:

Block display during program execution

- Display all blocks
- Display traverse blocks only

Current block

The last current block is shown in the block display. If you want to display all current blocks, you can use the selection menu to choose the desired block.

**M Fct* (SW 6.3 and higher)**

When this function is active, program execution on the NC is interrupted on request by the PLC at blocks in which the additional *associated M function* is programmed for a conditional stop 2.

The M Fct* number displayed here can be changed to the desired value in the Start-Up operating area under Machine Data/Channel MD by selecting MD 22256: AUXFO_ASSOC_M1_VALUE. The number stored in this machine data is then displayed. The value of this miscellaneous M function number must match the programmed stop 2.

4.6.13 DRF offset



Switching on and off

Function

The DRF (differential resolver function) offset allows you to apply an additional incremental zero offset (DRF offset) via a handwheel. It is active in the workpiece coordinate system for geometry axes and special axes. With this function it is possible, for example, to correct tool wear within a programmed block.

The DRF offset can be switched on and off for specific channels by means of the "Program control" function.

It remains stored until

- Power ON for all axes
- DRFOF (deselection of DRF via parts program)
- PRESETON (alteration of actual value by means of Preset)

Alter DRF

You can alter the DRF offset by traversing the appropriate machine axis using the handwheel (the actual value display does not change).



Sequence of operations

"Auto" is selected in the "Machine" operating area.

The standard axis assignment is defined.

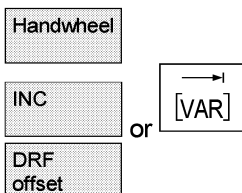
Enter the handwheel of your choice or select via the MCP.

Enter the increment size of your choice or select via the MCP.

The window "DRF offset" is displayed.

Traverse the required axes using the handwheel.

Using the same operating sequence, you can also return the DRF offset to the value "0".



4.6 Automatic mode

"Parameters" Operating Area

5.1	Tool data	5-179
5.1.1	Structure of tool compensation	5-179
5.1.2	Tool types and tool parameters	5-179
5.2	Tool offset	5-196
5.2.1	Function and basic display of tool offset	5-196
5.2.2	New tool	5-198
5.2.3	Display tool	5-200
5.2.4	Delete tool	5-201
5.2.5	New tool edge	5-201
5.2.6	Delete tool edge	5-202
5.2.7	Determine tool offsets	5-203
5.2.8	Immediate activation of tool offset	5-204
5.3	Tool management	5-205
5.3.1	Basic functions of tool management	5-207
5.3.2	Display/change tool data	5-215
5.3.3	Changing the meaning/display of wear data for tools	5-219
5.3.4	Grinding data extension in SW 6.2 or later	5-221
5.3.5	Loading	5-224
5.3.6	Unloading	5-229
5.3.7	Relocation	5-230
5.3.8	Tool master data in tool catalog	5-232
5.3.9	Tool offset data in the tool cabinet	5-235
5.3.10	Job processing for tools	5-238
5.4	R variables (arithmetic variables)	5-246
5.4.1	Function	5-246
5.4.2	Editing/deleting/finding R variables	5-246
5.5	Setting data	5-248
5.5.1	Working area limitation	5-248
5.5.2	Jog data	5-249
5.5.3	Spindle data	5-250
5.5.4	Dry run feedrate for DRY mode	5-251
5.5.5	Starting angle for thread cutting	5-252
5.5.6	Other types of setting data	5-253
5.5.7	Protection zones	5-254
5.5.8	Electronic gear (SW 6.3 and higher)	5-255
5.6	Work offset	5-256
5.6.1	Function	5-256

5.6.2	Changing the settable work offset (G54 ...)	5-258
5.6.3	Global work offset/frame	5-259
5.6.4	Display active settable work offsets	5-262
5.6.5	Display active programmable work offsets	5-263
5.6.6	Display active external work offsets	5-264
5.6.7	Display the sum of the active work offsets	5-265
5.6.8	Activate work offset and basic frame immediately	5-265
5.6.9	Actual value display: settable work offset system, SZS	5-266
5.7	User data/user variables (GUD, PUD, LUD)	5-267
5.7.1	General information	5-267
5.7.2	Changing/finding user data/user variables	5-268
5.8	Displaying system variables	5-270
5.8.1	Processing/creating variable views	5-271
5.8.2	Managing variable views	5-272
5.8.3	Logging system variables	5-273

5.1 Tool data

5.1.1 Structure of tool compensation

		<p>A tool is selected in the program by means of the T function. Numbers T0 to T32000 can be assigned to tools. Each tool can have up to 9 cutting edges: D1–D9. D1 to D9 activates the tool offset of a cutting edge for the active tool.</p> <p>The tool length compensation is applied with the first traversing motion (linear or polynomial interpolation) of the axis.</p> <p>A tool radius compensation is activated by programming of G41/42 in the active plane (G17, 18, 19) and in a program block with G0 or G1.</p>
	Tool wear	<p>Allowances for changes in the active tool shape can be made in the tool length (tool parameters 12 – 14) and tool radius (tool parameters 15 – 20).</p>
	Tool offset with D numbers only	<p>The so-called "flat D numbers" can be activated by means of NC machine data. In this case, the tool management is not available. The D number range increases to 1–32000. A D number can be assigned only once for each tool, i.e. each D number represents precisely one tool offset data record.</p>

5.1.2 Tool types and tool parameters

	Entries	<p>T no. Number of the tool</p> <p>D no. Number of the cutting edge</p> <p>Every data field (offset memory) that can be called with a D number contains not only the geometric information for the tool but also further entries, i.e. the tool type (drill, milling cutter, turning tools with cutting edge position etc.).</p>
--	----------------	--

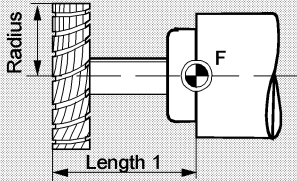
Tool types

Tool type classification:

- **Group with type 1xy (milling cutters):**

- 100 Cutter acc. to CLDATA
- 110 Ballhead cutter
- 120 End mill (without corner rounding)
- 121 End mill (with corner rounding)
- 130 Angle head cutter (without corner rounding)
- 131 Angle head cutter (with corner rounding)
- 140 Facing tool
- 145 Thread cutter
- 150 Side mill
- 151 Saw
- 155 Bevel cutter
- 160 Drill and thread milling cutter

Required offset values for a milling cutter

Entries in tool parameters										
DP1	1xy									
DP3	Length 1									
DP6	Radius									
Wear values according to requirement		<table border="1"> <thead> <tr> <th colspan="2">Effect</th> </tr> </thead> <tbody> <tr> <td>G17:</td> <td>Length 1 in Z Radius in X/Y</td> </tr> <tr> <td>G18:</td> <td>Length 1 in Y Radius in Z/X</td> </tr> <tr> <td>G19:</td> <td>Length 1 in X Radius in Y/Z</td> </tr> </tbody> </table>	Effect		G17:	Length 1 in Z Radius in X/Y	G18:	Length 1 in Y Radius in Z/X	G19:	Length 1 in X Radius in Y/Z
Effect										
G17:	Length 1 in Z Radius in X/Y									
G18:	Length 1 in Y Radius in Z/X									
G19:	Length 1 in X Radius in Y/Z									
Other values must be set to zero		F: Toolholder reference point								

Milling tool with adapter

Entries in tool parameters										
DP1	1xy									
DP3	Length 1 -Geometry									
DP6	Radius -Geometry									
DP21	Length -Adapter									
Wear values according to requirement		<table border="1"> <tr> <th colspan="2">Effect</th> </tr> <tr> <td>G17:</td> <td>Length 1 in Z Radius in X/Y</td> </tr> <tr> <td>G18:</td> <td>Length 1 in Y Radius in Z/X</td> </tr> <tr> <td>G19:</td> <td>Length 1 in X Radius in Y/Z</td> </tr> </table>	Effect		G17:	Length 1 in Z Radius in X/Y	G18:	Length 1 in Y Radius in Z/X	G19:	Length 1 in X Radius in Y/Z
Effect										
G17:	Length 1 in Z Radius in X/Y									
G18:	Length 1 in Y Radius in Z/X									
G19:	Length 1 in X Radius in Y/Z									
Other values must be set to zero		<p>F: Adapter reference point (when tool is inserted = toolholder reference point)</p> <p>F': Toolholder reference point</p>								

• **Group with type 2xy (drills):**

- 200 Twist drill
- 205 Solid drill
- 210 Boring bar
- 220 Center drill
- 230 Countersink
- 231 Counterbore
- 240 Tap, regular
- 241 Tap, fine
- 242 Tap, Whitworth
- 250 Reamer

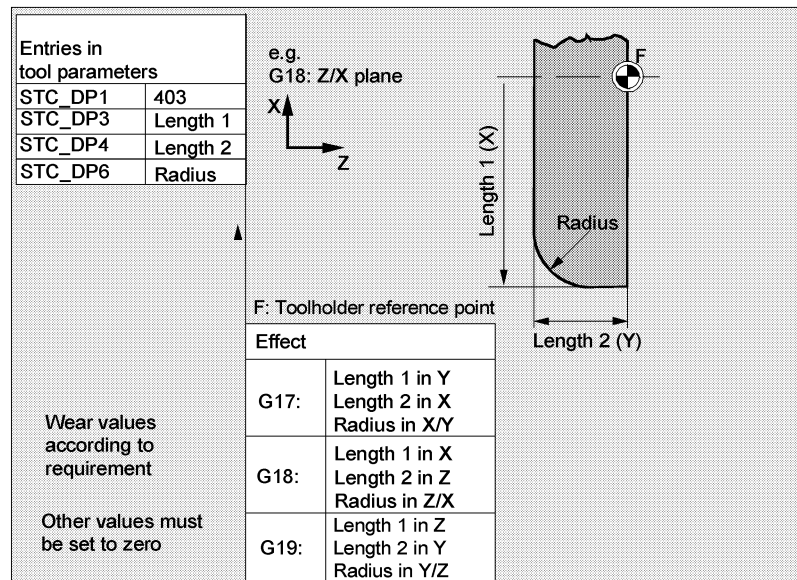
Required offset values for a drill

Entries in tool parameters										
DP1	2xy									
DP3	Length 1									
Wear values according to requirement		<table border="1"> <tr> <th colspan="2">Effect</th> </tr> <tr> <td>G17:</td> <td>Length 1 in Z</td> </tr> <tr> <td>G18:</td> <td>Length 1 in Y</td> </tr> <tr> <td>G19:</td> <td>Length 1 in X</td> </tr> </table>	Effect		G17:	Length 1 in Z	G18:	Length 1 in Y	G19:	Length 1 in X
Effect										
G17:	Length 1 in Z									
G18:	Length 1 in Y									
G19:	Length 1 in X									
Other values must be set to zero		<p>F: Toolholder reference point</p>								

- **Group type 4xy (grinding tools):**

- 400 Surface grinding wheel
- 401 Surface grinding wheel with monitoring
- 403 Surface grinding wheel with monitoring without base dimension for grinding wheel peripheral speed
- 410 Facing wheel
- 411 Facing wheel with monitoring
- 413 Facing wheel with monitoring without base dimension for grinding wheel peripheral speed
- 490 Dresser

Required offset values for a surface grinding wheel



Required offset values for inclined grinding wheel with implicit monitoring selection

Entries in tool parameters		STC_TPG1	Spindle number	
STC_DP1		403	STC_TPG2	Chaining rule
STC_DP3	Length 1	STC_TPG3	Minimum wheel radius	
STC_DP4	Length 2	STC_TPG4	Minimum wheel width	
STC_DP6	Radius	STC_TPG5	Current wheel width	
Wear values according to requirement		STC_TPG6	Maximum speed	
		STC_TPG7	Max. surface speed	
		STC_TPG8	Angle of the inclined wheel	
Other values must be set to zero		STC_TPG9	Parameter no. for radius calculation	
Effect		F: Toolholder reference point		
G17:	Length 1 in Y Length 2 in X Radius in X/Y			
G18:	Length 1 in X Length 2 in Z Radius in Z/X			
G19:	Length 1 in Z Length 2 in Y Radius in Y/Z			

Required offset values for inclined grinding wheel with implicit monitoring selection

Entries in tool parameters		STC_TPG1	Spindle number	
STC_DP1		403	STC_TPG2	Chaining rule
STC_DP3	Length 1	STC_TPG3	Minimum wheel radius	
STC_DP4	Length 2	STC_TPG4	Minimum wheel width	
STC_DP6	Radius	STC_TPG5	Current wheel width	
Wear values according to requirement		STC_TPG6	Maximum speed	
		STC_TPG7	Max. surface speed	
		STC_TPG8	Angle of the inclined wheel	
Other values must be set to zero		STC_TPG9	Parameter no. for radius calculation	
Effect		F: Toolholder reference point		
G17:	Length 1 in Y Length 2 in X Radius in X/Y			
G18:	Length 1 in X Length 2 in Z Radius in Z/X			
G19:	Length 1 in Z Length 2 in Y Radius in Y/Z			

5.1 Tool data

Required offset values for a surface grinding wheel without basic dimension for grinding wheel surface speed

Entries in tool parameters		STC_TPG1	Spindle number	
STC_DP1		403	STC_TPG2	Chaining rule
STC_DP3	Length 1	STC_TPG3	Minimum wheel radius	
STC_DP4	Length 2	STC_TPG4	Minimum wheel width	
STC_DP6	Radius	STC_TPG5	Current wheel width	
STC_DP21	L1 base	STC_TPG6	Maximum speed	
STC_DP22	L2 base	STC_TPG7	Max. surface speed	
		STC_TPG8	Angle of the inclined wheel	
		STC_TPG9	Parameter no. for radius calculation	
Wear values according to requirement				
Other values must be set to zero				
Effect				
G17:	Length 1 in Y Length 2 in X Radius in X/Y			
G18:	Length 1 in X Length 2 in Z Radius in Z/X			
G19:	Length 1 in Z Length 2 in Y Radius in Y/Z			

F: Toolholder reference point

Required offset values for a facing wheel with monitoring parameters

Entries in tool parameters		STC_TPG1	Spindle number	
STC_DP1		403	STC_TPG2	Chaining rule
STC_DP3	Length 1	STC_TPG3	Minimum wheel radius	
STC_DP4	Length 2	STC_TPG4	Minimum wheel width	
STC_DP6	Radius	STC_TPG5	Current wheel width	
		STC_TPG6	Maximum speed	
		STC_TPG7	Max. surface speed	
		STC_TPG8	Angle of the inclined wheel	
		STC_TPG9	Parameter no. for radius calculation	
Wear values according to requirement				
Other values must be set to zero				
Effect				
G17:	Length 1 in Y Length 2 in X Radius in X/Y			
G18:	Length 1 in X Length 2 in Z Radius in Z/X			
G19:	Length 1 in Z Length 2 in Y Radius in Y/Z			

F: Toolholder reference point

e.g.
G18: Z/X plane

Assignment of tool-specific parameters

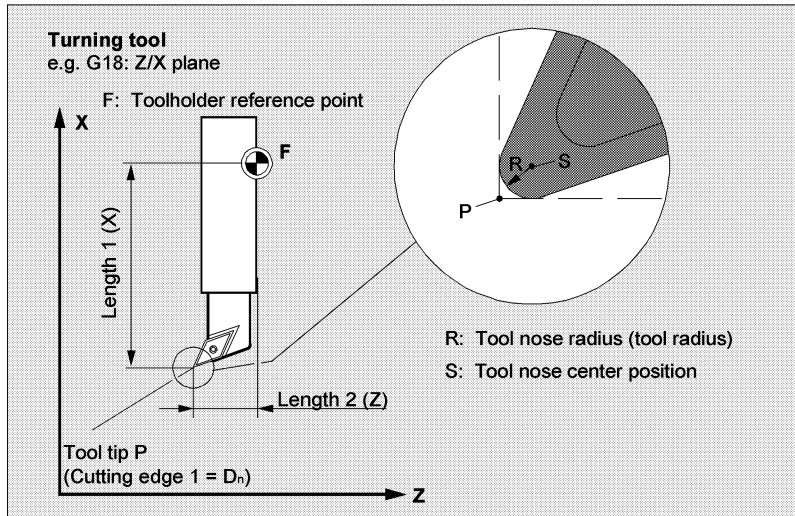
Parameters	Meaning	Data type
Tool-specific parameters		
\$TC_TPG1	Spindle number	Integer
\$TC_TPG2	Chaining rule	Integer
\$TC_TPG3	Minimum wheel radius	Real
\$TC_TPG4	Minimum wheel width	Real
\$TC_TPG5	Current wheel width	Real
\$TC_TPG6	Maximum speed	Real
\$TC_TPG7	Maximum surface speed	Real
\$TC_TPG8	Angle of the inclined wheel	Real
\$TC_TPG9	Param. no. for radius calc.	Integer
Additional parameters		
\$TC_TPC1	Angle of the inclined wheel	Real
to		
\$TC_TPC10		Real

- **Group type 5xy (turning tools):**

- 500 Roughing tool
- 510 Finishing tool
- 520 Recessing tool
- 530 Parting tool
- 540 Threading tool
- 550 Mushroom tool/form tool
- 580 Probe with cutting edge position parameter

5.1 Tool data

Required offset values for a turning tool with tool radius compensation



Required offset values for a turning tool with tool radius compensation

Tool parameter DP2 defines the tool nose position. Any value between 1 and 9 can be entered.

X Tool nose position DP2

Note:
Length 1, length 2 refer to point P for edge positions 1-8;
but in case of 9 to S ($S = P$)

Entries in tool parameters		Wear values according to requirement	Effect	
DP1	5xy		Other values must be set to zero	G17:
DP2	1...9	G18:		Length 1 in X Length 2 in Z
DP3	Length 1	G19:		Length 1 in Z Length 2 in Y
DP4	Length 2			
DP6	Radius			

- Group type 7xy (special tools)

- 700 Slotting saw

- 710 3D probe

- 730 Stop

Required offset values for slotting saw

Type 700 (slotting saw)

Entries in tool parameters			
DP3 Length 1 - Base			
DP4 Length 2 - Base			
DP6 Diameter - Geometry			
DP7 Zero width - Geometry			
DP8 Projection - Geometry	Wear values according to requirement		
Other values must be set to zero $L1 = DP3 + DP6/2$ $L2 = DP4 + DP7/2 - DP8$ $R = DP7/2$	Effect		
	G17:	Semi-diameter (L1) in Z Projection in (L2) Y Saw blade in (R) X/Y	Plane selection 1st-2nd axis (X-Y)
	G18:	Semi-diameter (L1) in Y Projection in (L2) X Saw blade in (R) Z/X	Plane selection 1st-3rd axis (X-Z)
	G19:	Semi-diameter (L1) in Z Projection in (L2) Z Saw blade in (R) Y/Z	Plane selection 2nd-3rd axis (Y-Z)

The offset data (TOA data) you can enter for tool type 700 "slotting saw" are as follows:

	Geometry	Wear	Base	
Tool length compensation				
Length 1	\$TC_DP3	\$TC_DP12	\$TC_DP21	mm
Length 2	\$TC_DP4	\$TC_DP13	\$TC_DP22	mm
Length 3	\$TC_DP5	\$TC_DP14	\$TC_DP23	mm
Radius offset				
Diameter	\$TC_DP6	\$TC_DP15		mm
Slot width b	\$TC_DP7	\$TC_DP16		mm
Projection k	\$TC_DP8	\$TC_DP17		mm

5.1 Tool data**3D probe**

Type 710

See /BNM/, User's Guide Measuring Cycles

Stop

Type 730

The stop is used to position the material in turning machines with bar feed. Only tool length compensation is important.

Calculating the tool parameters

Types 1xy (milling cutters), 2xy (drills) and 5xy (turning tools) are calculated according to the same scheme.

For the geometric values (e.g. length 1 or radius), there are several entry components. These are added to form a resulting quantity (e.g. total length 1, total radius) that then applies.

Tool parameter Number (P)	Meaning	Comment
1	Tool type	For overview see list
2	Cutting edge position	For turning tools only
Geometry		Tool length compensation
3	Length 1	Calculation acc. to type and plane
4	Length 2	
5	Length 3	
Geometry		Radius
6	Radius	Does not apply to drills
7	Reserved	
8	Reserved	
9	Reserved	
10	Reserved	
11	Reserved	
Wear		Length and radius compensation
12	Length 1	
13	Length 2	
14	Length 3	
15	Radius	
16	Reserved	
17	Reserved	
18	Reserved	
19	Reserved	
20	Reserved	

5.1 Tool data

Base dim./adapter	Length compensations	
21	Length 1	
22	Length 2	
23	Length 3	
Technology		
24	Clearance angle	For turning tools
25	Clearance angle	

Offsets that are not required must be set to zero (= default when the offset memory is set up).

The individual values of the offset memory (P1 to P25) can be read and written by the program via system variables.

The tool offsets can be entered not only via the operator panel front but also via the data input interface.



Calculation of tool base dimensions for two-dimensional – millhead

Entries in tool parameters DP1 5xy DP3 Length 1 - Geometry DP6 Radius - Geometry DP21 Length 1 - Base DP22 Length 2 - Base DP23 Length 3 - Base		<p>F': Tool base reference point F: Toolholder reference point</p>
Effect		
Wear values according to requirement Other values must be set to zero	G17: Length 1 in Z Length 2 in Y Length 3 in X Radius in Y/Z	
	G18: Length 1 in Y Length 2 in X Length 3 in Z Radius in X/Y	
	G19: Length 1 in X Length 2 in Z Length 3 in Y Radius in Z/X	

Calculation of tool base dimensions for three-dimensional millhead:

Entries in tool parameters		<p>F': Tool base reference point F: Toolholder reference point</p>
DP1	5xy	
DP3	Length 1 - Geometry	
DP6	Radius - Geometry	
DP21	Length 1 - Base	
DP22	Length 2 - Base	<p>Effect</p> <p>G17: Length 1 in Z Length 2 in Y Length 3 in X Radius in Y/Z</p>
DP23	Length 3 - Base	
Wear values according to requirement		
Other values must be set to zero		<p>G18: Length 1 in Y Length 2 in X Length 3 in Z Radius in X/Y</p>
		<p>G19: Length 1 in X Length 2 in Z Length 3 in Y Radius in Z/X</p>

Required length compensation values for turning tools:

Entries in tool parameters		<p>Turning machine e.g. G18: Z/X plane</p> <p>F: Toolholder reference point</p>
DP1	5xy	
DP3	Length 1	
DP4	Length 2	
Wear values according to requirement		<p>Effect</p> <p>G17: Length 1 in Y Length 2 in X</p>
Other values must be set to zero		<p>G18: Length 1 in X Length 2 in Z</p>
		<p>G19: Length 1 in Z Length 2 in Y</p>

5.1 Tool data

Turning tool with several cutting edges – length compensation:

Entries in tool parameters		Grooving tool e.g. G18: Z/X plane
DP1	5xy	
DP3	Length 1	
DP4	Length 2	
Entries in tool parameters		Grooving tool e.g. G18: Z/X plane
DP1	5xy	
DP3	Length 1	
DP4	Length 2	
Wear values according to requirement	Effect	
	G17:	Length 1 in Y Length 2 in X
	G18:	Length 1 in X Length 2 in Z
Other values must be set to zero	G19:	Length 1 in Z Length 2 in Y

Calculation of tool base dimensions for turning machine:

Entries in tool parameters		Turning tool e.g. G18 Z/X plane
DP1	5xy	
DP3	Length 1-Geometry	
DP4	Length 2-Geometry	
Entries in tool parameters		Turning tool e.g. G18 Z/X plane
DP21	Length 1 - Base	
DP22	Length 2 - Base	
Entries in tool parameters		
DP21	Length 1 - Base	
DP22	Length 2 - Base	
Entries in tool parameters		Turning tool e.g. G18 Z/X plane
DP21	Length 1 - Base	
DP22	Length 2 - Base	
Wear values according to requirement	Effect	
	G17:	Length 1 in Y Length 2 in X
	G18:	Length 1 in X Length 2 in Z
Other values must be set to zero	G19:	Length 1 in Z Length 2 in Y

Tool type 4xy (grinding tools) is calculated separately.

For the geometric values (e.g. length or radius), there are several entry components.

Parameters	Grinding wheel-comp. left	Grinding wheel-comp. right	Dresser left	Dresser right
Tool-specific parameters				
\$TC_DP1	Tool type	$*(2^0=1)$	Tool type	Tool type
\$TC_DP2	Cutting edge position	Cutting edge position	Cutting edge position	Cutting edge position
Geometry tool length compensation				
\$TC_DP3	Length 1	$*(2^2=4)$	Length 1	Length 1
\$TC_DP4	Length 2	$*(2^3=8)$	Length 2	Length 2
\$TC_DP5	Length 3	$*(2^4=16)$	Length 3	Length 3
\$TC_DP6	Radius	Radius	Radius	Radius
\$TC_DP7 to \$TC_DP11	Reserved	Reserved	Reserved	Reserved
Wear tool length compensation				
\$TC_DP12	Length 1	$*(2^{11}=2048)$	Length 1	Length 1
\$TC_DP13	Length 2	$*(2^{12}=4096)$	Length 2	Length 2
\$TC_DP14	Length 3	$*(2^{13}=8192)$	Length 3	Length 3
\$TC_DP15	Radius	Radius	Radius	Radius
\$TC_DP16 to \$TC_DP20	Reserved	Reserved	Reserved	Reserved
Tool base dimensions/adaptor dimension tool length compensation				
\$TC_DP21	Base length 1	$*(2^{20}=1048576)$	Base length 1	Base length 1
\$TC_DP22	Base length 2	$*(2^{21}=2097152)$	Base length 2	Base length 2
\$TC_DP23	Base length 3	$*(2^{22}=4194304)$	Base length 3	Base length 3

Technology				
\$TC_DP24	Reserved	Reserved	Reserved	Reserved
\$TC_DP25	Reserved	Reserved	Reserved	Reserved
Additional parameters				
\$TC_DPC1				
to				
\$TC_DPC10				

* Value of the chaining parameter if the compensation parameter is to be chained.

Parameter no. for radius calculation

\$TC_TPG9

With this parameter it is possible to define which offset value is used for grinding wheel surface speed, tool monitoring and centerless grinding. The value always refers to cutting edge D1.

\$TC_TPG9 = 3	Length 1 (geometry + wear + base, depending on tool type)
\$TC_TPG9 = 4	Length 2 (geometry + wear + base, depending on tool type)
\$TC_TPG9 = 5	Length 3 (geometry + wear + base, depending on tool type)
\$TC_TPG9 = 6	Radius

*: The tool parameter of cutting edge 2 is chained to the parameter of cutting edge 1 (see tool-specific grinding data \$TC_TPG2, chain rule). Here, typical chains are shown and the associated place value is specified in brackets.

Spindle number

\$TC_TPG1

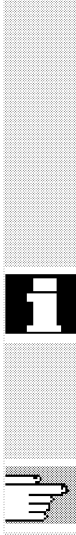
This parameter contains the number of the spindle to which the monitoring data and GWPS refer.

Chain rule

\$TC_TPG2

This parameter defines which tool parameters of the right wheel edge (D2) and left wheel edge (D1) must be chained (see TOA data). If the value of one of the chained parameters is changed, it is then automatically included in the chained parameter.

With an inclined grinding wheel it is important that the minimum grinding wheel radius be specified in the Cartesian coordinate system. The length compensations always specify the distances between the tool holder reference point and the tool tip in Cartesian coordinates.



The monitoring data apply both to the left and to the right cutting edge of the grinding wheel.

The tool lengths are not compensated automatically if the angle changes.

With inclined axis machines, the same angle must be used for the inclined axis and the inclined grinding wheel.

Offsets that are not required must be assigned the value 0 (= default when the offset memory is set up).

The tool offsets can be entered not only via the operator panel front but also via the data input interface.

For programming offset data see
/PG/, Programming Guide, Fundamentals

5.2 Tool offset

5.2.1 Function and basic display of tool offset

Tool offset data consist of data which describe the geometry, wear, identification, tool type and the assignment to parameter numbers.

The unit used for the dimensions of the tool is displayed.

The input field is highlighted.

Parameter	CHAN1	AUTO	MPF.DIR TEST.MPF
<input checked="" type="checkbox"/> Channel active			Program running
			T No. +
			T No. -
Tool offset data			D No. +
T number	1	D number	1
Tool type	100	No. of c. edges	1
C. edge pos.	1		
			D No. -
			Delete...
			Go to...
			Overview...
			New...
			Determine compens...
Tool offset	R variables	Setting data	Zero offset
			User data

	Geometry	Wear	Base	
Tool length comp.				
Length 1	:	0.000	0.000	mm
Length 2	:	0.000	0.000	mm
Length 3	:	0.000	0.000	mm
Radius comp.				
Radius	:	0.000	0.000	mm
DP7.18.res	:	0.000	0.000	
DP9.18.res	:	0.000	0.000	
DP10.19.res	:	0.000	0.000	
DP11.20.res	:	0.000	0.000	
Length compensation				
Clear angle	:	0.000	Deg.	
DP25.res	:	0.000		

Every offset number contains up to 25 parameters depending on the tool type.

The number of parameters shown in the window is that for the tool type.


Tool wear: fine compensation

If you have the authorization level set in MD 9202:

USER_CLASS_TOA_WEAR or higher, you can alter the values for tool fine compensation incrementally. The difference between the old and new values must not exceed the limit stored in MD 9450: WRITE_TOA_FINE_LIMIT.

The maximum number of offset parameters (T and D numbers) can be set by means of machine data.





Tool
offset

Horizontal softkeys

You can select different data types with the horizontal softkeys:

Selection of "Tool offset" menu

R
variables

Selection of "R variables" menu

Setting
data

Selection of "Setting data" menu

Zero
offset

Selection of "Zero offset" menu

User
data

Selection of "User data" menu

Determine
compens...

Support in determining tool offsets. This softkey is not required if the tool management function is available.

Vertical softkeys

The vertical softkeys support data input:

T No.
+

Selection of the next tool

T No.
-

Selection of the previous tool

D No.
+

Selection of next higher offset number (cutting edge)

D No.
-

Selection of next lower offset number (cutting edge)

Delete...

Deletion of a tool or cutting edge

Go to...

Find any tool or the active tool

Overview

List of all available tools

New...

New cutting edge or a new tool

5.2.2 New tool



Tool
offset

New...

New tool

Abort

OK

OK + new
tool

Function

If you create a new tool, the relevant tool types are automatically preselected as input support when you select the tool group.

Sequence of operations

The "Tool offset" window is displayed automatically.

Select softkey "New tool".

The "New tool" window appears on the screen.

As soon as you enter the first characters in the string for the tool group, for example

- 5xy turning tools,
all available tool types in the 5xy group are automatically displayed for you to select, i.e.,
- 500 Roughing tool
- 510 Finishing tool
- 520 Recessing tool
- 530 Parting tool
- 540 Threading tool.
- 550 Mushroom tool/form tool
- 580 Probe with cutting edge position parameter.

Enter the digits for your selections via the alphanumeric keypad or select them from the displayed list.

No new tool is set up.

A new tool is set up.
The window is closed.

Sets up another tool.
The window remains open so that you can set up other new tools.



OK + new
cutting edge

Sets up another cutting edge for the tool you have just set up.
The window remains open.

5.2.3 Display tool



Tool
offset



T No.
+

T No.
-



Function

You can select tools that you have set up and access their tool compensation data.

Sequence of operations

The "Tool offset" window is displayed automatically.

If the "Parameter" area has already been selected, the window and the last tool selected when the area was exited are displayed.

The tool offset data of the current tool are displayed immediately. If no tool has yet been selected, the data of the first tool are shown together with its first D number.

If no tools are available in the area, a message is output.

Select the set up tools.

Additional notes

Input of the geometry and wear data of the tool can be disabled using the keyswitch.

5.2.4 Delete tool



Tool
offset

T No. +/-
D No. +/-

Delete
tool

Delete
tool

Function

The tool is deleted together with all its cutting edges and the tool list updated accordingly.

Sequence of operations

The "Tool offset data" window is displayed automatically.

Scroll the screen contents until you reach the tool to be deleted.

The vertical softkey bar changes when softkey "Delete" is selected.

Select softkey "Delete tool".

The tool and all its edges are deleted. The tool offsets of the tool number preceding the deleted tool are displayed.

5.2.5 New tool edge



Tool
offset

Function

To help you to select a new tool edge, the associated tool types are displayed automatically when you select a tool group.

Sequence of operations

The "Tool offset" window is displayed automatically.



New...



New
edge

Press softkeys "New ..." and "New edge".

The window "New edge" is displayed.

As soon as you enter the first characters in the string for the tool group, for example

- 5xx turning tools

all available tool types in the 5xx group are automatically displayed for you to select, i.e.,

- 500 Roughing tool
- 510 Finishing tool
- 520 Recessing tool
- 530 Parting tool
- 540 Threading tool.
- 550 Mushroom tool/form tool

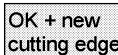
580 Probe with cutting edge position parameter.



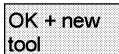

Abort



OK



OK + new
cutting edge



OK + new
tool

The possible tool point directions are displayed automatically for some tool types (e.g. for special tools, turning tools, grinding tools, etc.).

Enter the appropriate digit on the alphanumeric keyboard.

Creation of a new cutting edge is aborted.

A new cutting edge is set up.

A new cutting edge is set up.

Another new cutting edge can be set up.

A new cutting edge is set up.

Another new tool can be set up.

5.2.6 Delete tool edge




Delete



Delete
edge

Function

You can delete one or several edges of a tool. The tool list is updated automatically.

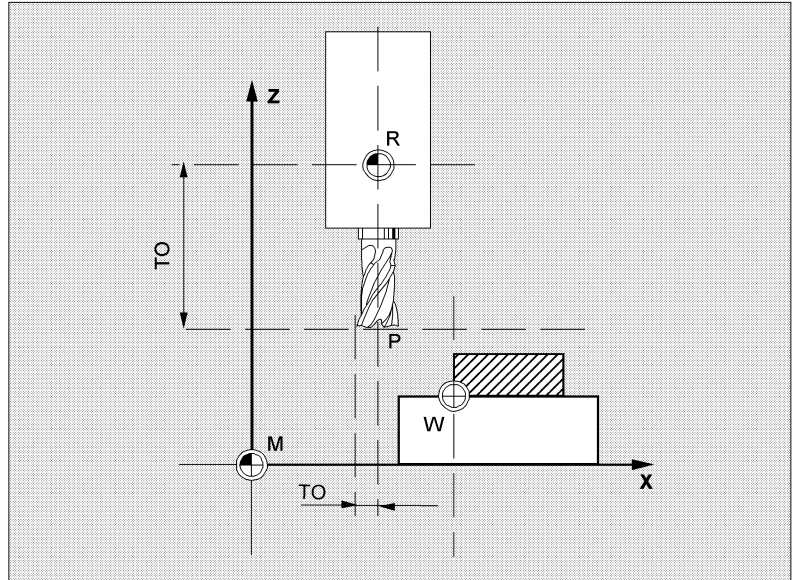
Select the edge of a tool, press softkey "Delete" and then softkey "Delete edge".

5.2.7 Determine tool offsets



Function

The "Determine tool offsets" function allows you to change the reference values of different axes and then to calculate them.



- TO** Tool offset, absolute dimension
- R** Tool mounting point
- M** Machine zero
- W** Workpiece zero

Sequence of operations

The "Tool offset" window is displayed.

Position the cursor on the tool parameter you wish to change.

The "Absolute dimension" window opens.

Use the "Toggle key" to select the appropriate axis. Alter the reference value if necessary using the numeric keypad.



Tool
offset

Determine
compens...

SELECT


 OK

When you press softkey "OK", the current position and corresponding reference value for the selected tool parameter are calculated.

The following equation applies:

Position – reference value = input value

The window is closed.


 Calculate

Position – reference value is entered in the input field. The window remains open.

If "Jog" mode is selected, it is also possible to change the position by traversing the axes.

The control automatically calculates the value from the reference value and the new position.

5.2.8 Immediate activation of tool offset



Function

Machine data can be set to ensure that the active tool offset is activated as soon as the parts program is in the "Reset" or "Stop" state.



Additional notes

If the function is used in the Reset state, machine data \$MC_RESET_MODE_MASK must be set such that the tool offset is not reset on a Reset.



/FB/, Description of Functions, Fundamentals, K2 Axes, Coordinate Systems...



Caution

The offset is applied after NC Start of the Reset in response to the next programmed axis motion in the parts program.

5.3 Tool management

The tool management system is organized by means of various configurable lists which show different views of the tools used.

Magazine list

In the "Magazine list", the tools of a magazine are displayed in order of ascending magazine location numbers.

You can find, display and, in the majority of cases, modify the data. A function is also provided for checking the D numbers and subsequently activating tools.



This list is mainly used to load and unload tools during setup, and to move tools between magazines.

Tool list

In the "Tool list", the tools are displayed in the order of ascending T numbers.



You use this list if you are working with small tool magazines and know the exact location of each tool in each of the magazines.

Working offset list

The cutting edges of active replacement tools are displayed in the "Working offset list". They are sorted in order of ascending D numbers.

You can search (according to D number/DL number), display and modify the data.



Use this list to modify and monitor sum offsets (location-dependent offsets), quantities and cutting edge parameters during the machining process. You can configure up to three different views for the working offset list.

Tool catalog and tool cabinet

The **tool catalog** contains only "ideal" tools.

"Ideal" tools are characterized by the relevant tool "master data" (i.e. with ideal tool dimensions, no wear, etc.). An "ideal" tool is uniquely defined by its "tool name".

The **tool cabinet** contains only "real" tools.

"Real" tools (i.e. real tool dimensions, with wear, etc.) are characterized by the relevant tool "offset data". A "real" tool is uniquely defined by its "tool name" and the associated "Duplo number". It is the "Duplo number" that assigns actual data to a "real" tool.

If a connection to a host computer exists, a message is automatically output to this host computer when a tool is loaded (but not relocated), unloaded or deleted, and the corresponding data block is transferred. The data are then still available on the host computer even after the tool has been deleted.

Machine manufacturer

Please refer to the information supplied by your machine manufacturer to see which functions are included in your tool management.

See /FBW/ Description of Functions Tool Management or
/FBSP/ Description of Functions ShopMill



5.3.1 Basic functions of tool management

The tool management system offers various tools for selection. You can assign geometric and technological data to the tool types in order to set up your master tool data. Several versions of each tool can exist. You can assign the actual data of the tool used (particular tool data) to these versions.

You start the tool management from the operating area "Parameters" by pressing the softkey.

The machine manufacturer configures which list is to be displayed when the tool management system is called up. In the example shown, the "Magazine list" is displayed.

Important

The structure of the table is **freely** programmable (configured by the machine manufacturer).

The **example** shows only one possible case:

Tool
managemt.



Parameter	CHAN1	AUTO	MPF DIR LEER.MPF																																			
<input checked="" type="checkbox"/> Channel RESET																			Magazine list 1																			
																			Magazine list 2																			
Magazine list 1																																						
Magazin: 2 - Kette20 Plätze: 20 AVB: 0																																						
Pl	PP	PTP	WerkzeugID	Dupl	TNr	PTT	WW	WWW	WWW	PV	WTyp	xGeo-L1	xGeo-L2						Magazine list 3																			
1	-	-	1	newRack860	1	1	1	-	F	G	M	V	-	0	900	11.0000	11.0000		Tool data																			
2	-	F	1	Wzg1	1	76	1	-	F	-	-	-	-	0	900	0.0000	0.0000		Activate D check																			
3	-	F	1											0					Buffer locations																			
4	-	-	1											0					Search and position																			
5	-	-	1	Wzg2	1	85	1	-	F	-	-	-	E	0	900	0.0000	0.0000		Next magazine																			
6	-	F	1	Wzg3	1	7	1	-	F	-	-	-	-	0	900	0.0000	0.0000																					
7	-	F	1											0																								
8	-	F	1											0																								
9	-	F	1											0																								
10	-	F	1											0																								
11	-	F	1											0																								
12	-	F	1											0																								
13	-	F	1											0																								
14	-	F	1											0																								
<table border="1"> <tr> <td>Magazine list</td> <td>Tool list</td> <td>Load</td> <td>Unload</td> <td>Relocate</td> <td>Working offset list</td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> </tr> </table>																			Magazine list	Tool list	Load	Unload	Relocate	Working offset list														
Magazine list	Tool list	Load	Unload	Relocate	Working offset list																																	

Location (PI)	Location number
Location state (P)	Location state (one column is provided for each state) e.g. F = location free D = location disabled
PTP	Type assigned to the location
ToolID	Name of the tool
Dupl	Number of replacement tool
TNo	Internal T numbers which may be needed for reloading tool data.
PTT	Type of location assigned to the tool
W (8x)	Tool status (one column is provided for each status) No display = Replacement tool A = Active tool F = Tool enabled G = Tool disabled M = Measured tool V = Warning limit reached W = Tool is being changed P = Fixed-location-coded tool E = Tool was in use
PV	Wear group assigned to the tool.
Tool type	Tool type Only some of the tool offsets are enabled for input, depending on the tool type. All other tool types are preset to a value of 0.
Geo - L1 ...	Tool offsets such as length, radius, wear, monitoring data, etc.
Radius ...	













Special situation:

Horizontal softkeys

The first or most recently displayed magazine is shown in the "Magazine list" together with all the tools which have been loaded. You can access the next magazine by pressing the corresponding vertical softkey.

All tools which are stored as a set of data on the NC are displayed (irrespective of whether they have been assigned to a magazine location).

A magazine location is assigned to the tool.

The tool is deleted from the current magazine location.

The tool is moved from the current magazine location to another location.

The cutting edges of active replacement tools are displayed. They are sorted in order of ascending D numbers.

Further softkeys are displayed when you press the "Etc." key:

You can create new tool master data ("ideal" tools) and modify the existing data.

You can create new tool offset data and tool operating data ("real" tools) and modify the existing data.

This softkey allows you to display the tool data as transformed data (adapter data are calculated) or non-transformed data.

This conversion feature is only available in the magazine list display. The data are always displayed as non-transformed data in the tool list and as transformed data in the working offset list.

If you display transformed data and want to create a new tool in the magazine list (loading mode), the transformed data display is activated exclusively for input of this data block.

(Names assigned by user)

Magazine list 1

Magazine list 2

Magazine list 3

Tool data

Activate D check

Vertical softkeys (magazine list)

Selection of user-specific display windows (if configured), e.g.

- General data
- Geometry data
- Wear data

You can display and edit the complete data of a tool.

Call up the start screen for updating the tool data. You can display and normally also modify all the data of the tool, its cutting edges and its working offsets in this screen and subsidiary screens. The vertical softkey bar changes.

The exact procedure is described under "Displaying/modifying tool data" (on page 215 ff).

Two functions are implemented with this softkey:

- Verify unique D number assignment
Duplication can occur when assigning D numbers to the cutting edges of the individual tools. This function checks all numbers within the current magazine or all allocated D numbers within a TO unit (configured by parameter).
- Activating the tools
If unique D numbers were assigned, a wear group is activated. A tool is subsequently activated from each replacement tool group in the TOA of the current channel. Allowance is made for the active wear group.

Only automatically accessible magazines are considered for processing. Any previously active tools are disabled by the activation of tools, especially if the wear group is changed.





Buffer locations

Display and skip the buffer window in the magazine list. The display shows spindles, grippers, etc., i.e. locations which can accommodate tools but which are not magazine locations. If no buffers are defined, this key has no function.

Search and position

The "Find tool/location" window appears.

- Search
 - Tool:
Enter the tool name and Duplo number and start the search with OK.
The cursor is positioned on the tool found.
 - Location:
Enter the magazine and location number and start the search with OK.
The cursor is positioned on the tool found.
- Position
Select softkey "Position",
tool/location is moved to loading point. If there is more than one possible loading point, a window is opened. You can select the correct one with the cursor.

Next magazine

The locations of the next magazine are displayed in the magazine list.

You can only scroll forwards. When you reach the last magazine, the display returns to the first magazine.

(Names assigned by user)

Tool list 1

Tool list 2

Tool list 3

Tool details

Tool from CC

Tool in cabinet

Delete tool

Tool in cabinet

Data on CC

Abort

OK

Vertical softkeys (tool list)

Selection of a table structure configured by the user (if configured), e.g.:

- General data
- Geometry data
- Wear data

You can display and edit the complete data of a tool (as in the magazine list).

The tool data are read from a code carrier and entered in the tool list (you can then edit the data).

Select the tool in the tool cabinet.

The tool data are read from there and entered in the tool list.

A list display appears for which you can parameterize a filter for the tool number, duplo number and tool type. The list displays all the tools that fulfill the filter criteria. You can select the tool you require from the list.

The selected tool is removed from the list. Use the vertical softkeys to determine whether to save the tool data.

The data are copied into the tool cabinet. The tool can be loaded later with the same data.

If a code carrier is installed, the tool data are stored on the carrier. The tool can then be loaded later with the same data.

The operation is canceled. The tool is not removed from the list.

The tool is deleted from the list. The tool data are no longer available.



New tool

<<

Each time you press this softkey, a tool is created **immediately**. The screen form for entering the tool data (tool details) and the associated softkey bar (same as tool details) appear. The values are initialized with the default settings (configured in the INI file) and can be modified here (e.g. you can change the name of the tool).

Use the vertical softkey bar to display the tables for cutting edge data and offsets with the default settings. If necessary, enter any changes in the individual views.

Terminates input of the tool data and switches to the tool list display. The new tool appears in the table and is available for loading.

Once you have set up a new tool, the cursor automatically jumps to the line containing the new tool when you return to the tool list. This gives you feedback about your operation.

Note

The data you enter are always updated immediately (without prompting). You can change the properties of the new tool by pressing the "Tool details" softkey.

It is not possible to modify the data directly in the table.

You can only change the name and type of a tool by selecting New-Add, not via Tool details.

To change a name, you have to create a new tool and delete the old one.

(Names assigned by user)

W offset
list 1

W offset
list 2

W offset
list 3

Tool data

Find
D number

Current
D number

Vertical softkeys (working offset list)

Selection of a table structure configured by the user (if configured), e.g.:

- General data
- Geometry data
- Wear data

You can display and edit the complete data of a tool (as in the magazine list).

Find an entry with a specific D number/DL number.

- Enter the D number and DL number you want to find in the search window.
- Confirm with "OK". If a matching entry is found, the cursor jumps to the corresponding line. If you have not specified a DL number, the cursor is positioned on the first line of the matching tool.

The D number of the current tool is determined and displayed.

5.3.2 Display/change tool data



Tool ma-
nagement

Magazine
list

Tool list

Working
offset list

Tool
details

Function

You can view and edit the tool data of the tool selected in the lists.

You can edit the following cutting edge data:

- Offset values
- Monitoring data
- User data

Sequence of operations

Select softkey "Tool management".

The list configured by the machine manufacturer appears (e.g. the magazine list). The horizontal and vertical softkey bars change.

Select the appropriate list via softkey:

- "Magazine list"
- "Tool list"
- "Working offset list"

Position the cursor bar on the appropriate tool. The tool is selected.

Select softkey "Tool details".

The input screen form for "Tool details" appears.

The vertical softkey bar changes again.

The following functions are available:

- Create new cutting edges
- Modify cutting edge data
- Modify monitoring data
- Modify location-dependent offsets (DL numbers)
- Delete cutting edges

You can modify the following data in the input screen form:

- Type of location
- Tool location coding
- Monitoring type
- Status (available, disabled, measured, etc.)
- Tool user data (OEM_Tx; x = 1...10)
- D numbers
- Tool name
- Duplo number
- Tool type, only in magazine list and tool list

**Note**

The tool data Name, Duplo number and Type can only be changed if the option has been enabled by the machine manufacturer. If the option is not activated you cannot make any changes. These data are permanently defined when a new tool is created.

New edge

New cutting edges are created for the displayed tool. A cutting edge number to which no D number has yet been assigned is automatically selected in the table.

When you have allocated a D number, the value is displayed in red (the cutting edge has not yet been created). Select "Cutting edge data" in the vertical softkey bar. The selected cutting edge is created.

Default values are assigned to the cutting edge data and the corresponding table is displayed. Enter any necessary changes.



You can use the "Cutting edge +" and "Cutting edge –" keys to display and, if necessary, edit the data of the other cutting edges.

The data are updated immediately.

Press the "<<" softkey to return to the Tool details input screen form.

The new cutting edge is defined. The display color changes.



Edit the tool data



Cutting
edge data

Additional notes

If nine cutting edges have already been defined for the tool, you must delete one of the cutting edges (via softkey) before you can create a new one.

A new cutting edge can be added to a tool at any time (even if the tool is already located in the magazine). Select "New cutting edge" and enter the cutting edge data.

The softkeys "Cutting edge data", "Monitoring data" and "Location-dependent offsets" display tables for editing the data. You can move freely between the individual tables. The name, Duplo number and type of tool and all defined cutting edges (#1...#9) are displayed. Use the softkeys "Cutting edge +" and "Cutting edge –" to change between the cutting edges. Press the "<<" softkey to return to the Tool details input screen form.

Note

When you edit the data, the new data is saved immediately. The "<<" softkey only changes the display.

The cutting edge data and tool offsets of the currently selected cutting edge are displayed and can be edited.

You can edit the following data:

- Cutting edge user data (OEM_Sx; x = 1...10)
- Tool offsets
 - Geometry
 - Wear
 - Basic offset
 - Tool point direction (for turning tools)
 - Tool clearance angle (for turning tools)

You can define the value of Length1, Length2, Length3 and Radius1 for every parameter.

Monitoring
data

The monitoring data of the currently selected cutting edge are displayed and can be edited.

When you have selected a cutting edge, define the actual value, setpoint and prewarning limit monitoring data for the following parameters:

- Quantity
- Service life
- Wear

Loc-dep.
offsets

The tool offsets (identical to the cutting edge data table) and the location-dependent offsets of the currently selected cutting edge are displayed and can be edited.

You can change the following settings:

- You can define the value of Length1, Length2, Length3 and Radius1 for every parameter.
- Up to six location-dependent offsets (DL1...DL6) are possible (according to the machine manufacturer's configuration).
- Setup values and wear values can be defined for each offset.

Edge +

In the individual tables, this softkey displays the data/offsets of the **next** cutting edge for editing.

Edge -

The data/offsets of the **previous** cutting edge are displayed and can be edited.

Delete
edge

In the table, select the cutting edge you want to delete (by positioning the cursor).

The cutting edge is deleted when you activate the softkey.

Notice! There is no separate prompt.

The D number is removed from the table and can be reallocated.

<<

Terminates input of the tool data and returns to the table displayed previously.



Note

You can enter individual tool data directly in the list, provided you have the necessary access authorization and the data are displayed in the list (the list structure is configurable).

Select the value to be edited and enter the required data. The system automatically switches to edit mode.

5.3.3 Changing the meaning/display of wear data for tools

In the tool list screens, additional symbols indicate whether special features have to be taken into account for the current tool according to G code 56 and setting data SD 42935: WEAR_TRANSFORM.

Symbols

TRANSFORMIERT: Arbeitskorrekturen Liste 1

Zeil	Aktuell	Werkzeug-Id	Dupl	DNo	Mag
1	REV2	REI1	1	10	
2		REI1	1	11	
3		REI1	1	12	
4		REI1	1	13	
5		REI2	1	20	
6		REI2	1	21	
7		REI2	1	22	
8		REI2	1	23	
9	HAND	REI3	1	30	
10		REI3	1	31	
11		REI3	1	32	
12		REI3	1	33	
13		REI4	1	40	
14		REI4	1	41	
15		REI4	1	42	
16		REI4	1	43	




Arbeitskorrekturen R-Parameter Setting-daten Nullpa-verschl

The G56 reset value is TOWSTD and at least 1 bit is set in \$SC_WEAR_TRANSFORM.


The current tool for channel "REV2" is indicated in row 1. The current value of G56 on channel "REV2" differs from the G56 reset value (). Row 9 displays the current tool for channel "HAND". Here the current value of G56 is the same as the reset value ().



Symbols 

TRANSFORMIERT: Arbeitskorrekturen Liste 1

Zeil	Aktuell	Werkzeug-Id	Dupl	DNo	Mag
1		REI1	1	10	4
2	 REV2	REI1	1	11	4
3		REI1	1	12	4
4		REI1	1	13	4
5		REI2	1	20	
6		REI2	1	21	4
7		REI2	1	22	4
8		REI2	1	23	4
9		REI3	1	30	5
10		REI3	1	31	5
11	 HAND	REI3	1	32	5
12		REI3	1	33	5
13		REI4	1	40	5
14		REI4	1	41	5
15		REI4	1	42	5
16		REI4	1	43	5




Arbeitskorrekturen Werkzeug-Liste

The G56 reset value is TOWMCS  and \$SSC_WEAR_TRANSFORM is set.


The current tool for channel "REV2" is indicated in row 2. The current value of G56 on channel "REV2" differs from the G56 reset value (). Row 11 displays the current tool for channel "HAND". Here the current value of G56 is the same as the reset value ().


Symbols 

TRANSFORMIERT: Arbeitskorrekturen Liste 1


Zeil	Aktuell	Werkzeug-Id	Dupl	DNo	Mag
1		RE11	1	10	
2		RE11	1	11	
3		RE11	1	12	
4		RE11	1	13	
5		RE12	1	20	
6	 REV2	RE12	1	21	
7		RE12	1	22	
8		RE12	1	23	
9		RE13	1	30	
10		RE13	1	31	
11		RE13	1	32	
12		RE13	1	33	
13		RE14	1	40	
14		RE14	1	41	
15	 HAND	RE14	1	42	
16		RE14	1	43	

Arbeitskorrekturen Werkzeug-Liste

The G56 reset value is TOWWCS  and \$SC_WEAR_TRANSFORM is set.

The current tool for channel "REV2" is indicated in row 6. The current value of G56 on channel "REV2" is the same as the reset value (.

Row 15 displays the current tool for channel "HAND".

Here, the current value of G56 differs from the G56 reset value (.

Additional notes

Additional information can be found in /FBW/, Description of Functions, Tool Management.

5.3.4 Grinding data extension in SW 6.2 or later



Function

If the selected tool is a grinding tool, a vertical softkey 6 "Grinding data" is provided in the following:

- Tool details basic display
- Tool details cutting edge data (secondary) display
- Tool details monitoring data (secondary) display

When this softkey is pressed, a display appears where grinding data can be:

- Displayed
- Modified.

Werkzeug Schleif-Daten			
Name:	SCHLEIFER220	Duplo:	2
Typ:	403 Umf-Schleifscheibe mU oB		
Schneiden:	#1	#2	#3
D	1		
Maximale Drehzahl	10000.000	[U/min]	
Maximale Umfangsgeschwindigkeit	130.000	[m/s]	
Minimaler Scheibradius	220.000	[mm]	
Minimale Scheibenbreite	140.00	[mm]	
Aktuelle Scheibenbreite	160.000	[mm]	
Winkel der schrägen Scheibe	30.000	[grad]	
Spindel-Nummer	1		
Parameter-Nummer für Radiusrechnung	3		
Verkettungsvorschrift	9		
OEM_T1 [mm]	0.000	OEM_T2 [mm]	0.00
OEM_T3 [mm]	0.000	OEM_T4 [mm]	0.0000
OEM_T5 [mm]	0.00000	OEM_T6 [m/s2]	0.000
OEM_T7 [U/s2]	0.000	OEM_T8 [m/s3]	0.000

Arbeitskorrekturen Werkzeug-Liste Magazin-Liste

This display contains the following:

Top section:

- Tool name
- Duplo number
- Type
- Cutting edge navigation bar
as in the "Cutting edge data" display for tool details
(The data is only displayed and cannot be modified)

Middle section:

The data correspond to the system variables entered for grinding.

Variable for	Unit	Identifier
Spindle number	-	\$TC_TPG1
Chaining rule	-	\$TC_TPG2
Minimum wheel radius	[mm, in]	\$TC_TPG3
Minimum wheel width	[mm, in]	\$TC_TPG4
Current wheel width	[mm, in]	\$TC_TPG5
Maximum speed	[rpm]	\$TC_TPG6
Maximum peripheral speed	[m/s, ft/s]	\$TC_TPG7
Angle of the inclined wheel	[degrees]	\$TC_TPG8
Parameter number for radius calculation/compensation parameter for GWPS	-	\$TC_TPG9



Scroll bar

Additional notes

For information on NC variables, please refer to the Description of Functions

/FB/, W4 Grinding-Specific Tool Offset and Monitoring.

Bottom section:

Tool OEM data are displayed as in the basic display for tool details. The tool OEM data can be modified in both the grinding data display and the basic display for tool details. The designation and the unit in the OEM data can be parameterized in the relevant language. The bottom section of the display is only available if tool OEM data are available on the NC.

A scroll bar appears next to the middle and bottom sections if the bottom section is displayed.

The grinding data display and the softkeys are only displayed if the tool is a grinding tool (tool type between 400 and 499).

Access rights are checked for grinding data.

Default access is "All".

All cutting edges on a tool are of the same type. The same applies to the dresser cutting edges of grinding tools.

Tool types

The tool types for grinders are 4xy in accordance with /FB/, W4.

They are available in displays/functions for:

- Tool cabinet
- Tool catalog
- Cutting edge parameters in tool details
- Lists (magazine list, tool list, working offset list)

Supplementary conditions

- The tool-specific grinding data cannot be displayed in the list displays.
- They are not saved in the tool cabinet/tool catalog.
- They are not transferred via code carrier/SINCOM.
- The extension is available in HMI Advanced Version 6.2 or later.

5.3.5 Loading



Function

You can load a tool in one of the following ways:

- You can enter the individual tool data directly in the list.
- You can import tool data from existing tools.

You can load a tool from the "Magazine list" or the "Tool list".

- **Loading a tool from the "Magazine list"**

To edit tool data directly in the list, you must first find a suitable empty location for the tool (use the softkeys). You can then enter the data directly in the list.

You can also load all existing tools into the magazine.

You load the associated tool data:

- from the master data catalog
- from the tool cabinet
- from the code carrier (if one exists) or
- from the host computer (if one is connected).

In this case, the system automatically tries to find a suitable empty location for the selected tool.

- **Loading a tool from the "Tool list"**

You can load magazines whose data are already stored in the TO memory.

The magazine location is selected either by searching for an empty location or by entering a magazine number and location number in the corresponding columns of the list.



Tool ma-
nagement

Magazine
list

Load

Manual entry of data (with search for empty location)



Additional notes

You can parameterize a filter for loading tools into the lists. A list display appears for which you can parameterize a filter for the tool number, duplo number and tool type. The list then displays the tools in the tool cabinet that meet the filter criteria. By selecting it from the list, you accept the tool.

Sequence of operations (loading from the "Magazine list")

Select softkey "Tool management".

The "Magazine list" is displayed.

The horizontal and vertical softkey bars change.

The "Magazine list" is selected.

The appropriate magazine is selected.

Select the "Load" softkey.

The vertical softkey bar changes again.

If you want to enter data directly in the table, you must first find a suitable empty location in the magazine.

There are four methods by which you can find an empty location for different tool sizes in combination with location types:

1. Directly in the magazine list ("Manually").
2. Via a user-defined location
e.g. "oversize" (name configured by machine manufacturer)
3. Via the "Find empty location" softkey
4. Via the "To loading point" softkey

Additional notes

Values are checked for validity when you enter a tool type in the lists. Only known tool types are permitted.

Searching directly in the magazine list

Position the cursor on the location of your choice in the magazine list.

normal

large

oversize

normal and
heavy

Find empty
location

To loading
point

Tool details

Search via user-defined location (example)

The assignment of the softkeys is configured by the machine manufacturer.

- "normal" (name configured by machine manufacturer)
- "large" (name configured by machine manufacturer)
- "oversize" (name configured by machine manufacturer)
- "normal and heavy" (name configured by machine manufacturer)

The system searches for a suitable empty location.

The cursor bar is automatically positioned on the magazine location found in the "Magazine list".

Search via the "Find empty location" softkey

Enter the "Tool size" and "Location type" in the dialog box.

If more than one loading point is configured, select the desired loading point from a query window.

The system searches for a suitable empty location.

The cursor bar is automatically positioned on the magazine location found in the "Magazine list".

Search via the "To loading point" softkey

You have found an empty location in front of the current loading point.

When you press "To loading point" the cursor is automatically positioned at this location.

Enter data

If the desired empty location is found after a search operation, the system switches to edit mode and the softkey bar changes. When you search for an empty location manually, the system switches to edit mode as soon as you start entering the data on the keyboard.

You can use the "Tool details" display to edit the data of the tool to be loaded (if necessary).

If the tool has not yet been created, it is created automatically when you call up the input screen form.

Abort

Load/input mode is canceled.

A tool created via "Tool details" or "Start" is deleted. You can search for another empty location.

Start

The loading operation is initiated. If the tool has not yet been created, it is created automatically.



Importing tool data

If data are still missing for loading, the input screen form for the tool details appears. The missing data are initialized with default values and can be edited if necessary. Start the loading operation again afterwards.

In addition to direct data entry, there are various ways of importing and loading tool data belonging to tools which have already been defined:

Data from
CC

1. Read the data in from a code carrier (if one is installed)

Data from
host

2. Read the data in from a host computer (if one is installed)

Tool from
cabinet

3. Select the "Tool from cabinet" menu.

Select the tool in the tool cabinet. The tool data are read from there.

If not all of the softkeys are visible, change the display by pressing the appropriate key on the operator panel front.

When you have defined the source for loading, the system automatically tries to find a suitable empty location for the tool to be imported. The vertical softkey bar changes.

If no location is found, an error message is displayed.

Tool
details

You can use the "Tool details" display to edit the data of the tool to be loaded (if necessary).

If the tool has not yet been created, it is created automatically when you call up the input screen form.

Abort

Load/input mode is canceled.

A tool created via "Tool details" or "Start" is deleted. You can search for another empty location.



Start

The loading operation is initiated. If the tool has not yet been created, it is created automatically.



"Load directly to spindle" is possible if the cursor is positioned on the spindle buffer location.



Sequence of operations (loading from the "Tool list")



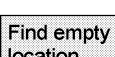
Tool list

The "Tool list" is selected.
The appropriate tool is selected.



Load

Select the "Load" softkey.
The vertical softkey bar changes.



Find empty location

Search for an empty location for a tool which has already been created or enter the desired location and magazine number in the list.

The location found is entered under the magazine/location number.



Abort

The loading operation is not initiated.
The basic display appears.



Start

The loading operation is initiated.

5.3.6 Unloading



Tool management

Magazine list

Tool list

Unload

Tool in cabinet

Data on CC

Function

This function allows you to unload a selected tool and to save its data.

Sequence of operations

Select softkey "Tool management".

The "Magazine list" is displayed.

The horizontal and vertical softkey bars change.

You can unload a tool from the "Magazine list" or the "Tool list".

The sequence of operations is the same for both methods.

Select the appropriate list via softkey:

- "Magazine list"
A tool is to be physically removed from a magazine location. You can configure whether the corresponding NC block is also removed from the TO memory.
Select the magazine and the tool to be unloaded (select the tool with the cursor).

or

- "Tool list"
The NC block is to be unloaded from the memory. Select the tool to be unloaded (select the tool with the cursor).

Select softkey "Unload".

The vertical softkey bar changes.

The tool data of the selected tool are stored in the tool cabinet on the hard disk. This allows you to load the tool with the same data again at a later point.

If a code carrier is installed, the tool data are automatically stored there. This allows you to load the tool with the same data again at a later point.



From
magazine

The selected tool is unloaded.

The corresponding line is deleted from the magazine list.

The entries in the magazine number and location number columns are removed from the tool list.

"Unload directly from spindle" is possible only if the buffer is selected and the cursor is positioned on the location of the spindle.

To quit the display **without** unloading the tool, select another display in the vertical softkey bar **before** activating "Start".



Delete
tool

The tool data of the selected tool are deleted from the TO memory. If the tool is in a magazine location, it is unloaded and deleted.

If a host computer is connected, the data are transferred to the host computer each time a tool is deleted or unloaded.

5.3.7 Relocation

Function

This function allows you to move a selected tool from one location to another.

Sequence of operations

Select softkey "Tool management".

The "Magazine list" is displayed.

The horizontal and vertical softkey bars change.

You can relocate a tool from the magazine list or the tool list. The sequence of operations is the same for both methods.



Tool ma-
nagement

Magazine list

Tool list

Relocate

Abort

Start



Select the appropriate list via softkey:

- "Magazine list"
Select the magazine and the tool to be relocated (position the cursor on the magazine location containing the tool).

or

- "Tool list"
Select the tool to be relocated (position the cursor on the tool).
A tool must already have been loaded (entry in the magazine number and location number columns).

When you select the "Relocate" key, the "Relocate tool" window opens.

There are 2 methods by which you can select the new location for the tool:

- Enter the magazine and location numbers in the "Relocate tool" window.
- or
- Select softkey "Find empty location" and select the desired data in the window.

The tool is not relocated.

The tool is relocated to the new empty location.

Use magazine number 9998 to move a tool to or from a spindle location.

When relocating from the buffer, the previous location information is entered by default. This applies to fixed-location-coded and variable tools.

5.3.8 Tool master data in tool catalog



Function

You can create tool master data in the tool catalog. A set of data can be created for every tool in use.

Advantage

Master data which apply to the tool, regardless of which cutting edge you use, do not have to be entered again for every new tool you create, but can be copied from the tool catalog (in the tool cabinet) for each tool you use.

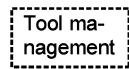


Ideal tools

The **tool catalog** contains only "ideal" tools. "Ideal" tools are characterized by the relevant tool "master data" (i.e. with ideal tool dimensions, no wear, etc.). An "ideal" tool is uniquely defined by its "Tool name".



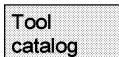
Sequence of operations



Select softkey "Tool management".
The horizontal and vertical softkey bars change.



The horizontal softkey bar is expanded.



Select softkey "Tool catalog".
The vertical softkey bar changes again. The tool details of the tool catalog are displayed.

You can use the list fields to display the available standard tools and tools which have already been defined or to create new tools.

Creating tool data

Proceed as follows to create the tool data:

- Select the desired technology in the appropriate list field (e.g. drilling tools, milling tools).
- Define the tool type in the second list field (e.g. twist drill).



- Activate this softkey to create a new tool.
You can edit the field for the tool name.
- Enter a tool name.
- In the open window "Tool details", define the tool properties (in "Tool size" you define the total number of tool half-locations occupied by the tool).



- Select "Abort" to discard the settings.
The tool is not created.



- Save your data with OK.
The new tool is created.

Displaying/changing tool data

In addition to the tool master data already defined, you can enter default settings in the tool catalog for all the other tool data (e.g. cutting edge data, user data). You can change this data later. Duplo number 0 is allocated to the tool.



Tools in the tool catalog are used as a basis for real tools. It is recommended to define only data actually required in that exact form for several real tools. This helps to minimize the amount of changes required later.

Tool data are displayed and edited as follows:

- Tool offset data (cutting edge data)
The tool offset data window is displayed. The data of the first cutting edge are listed in a table. The vertical softkey bar changes. Enter the required settings.

The following functions are available for editing the cutting edge data:



The cutting edge data of the next defined cutting edge are displayed in a table.



The data of the previous cutting edge are listed.



A new cutting edge is created for the tool.



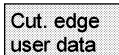
The current cutting edge and all the data defined for the cutting edge are deleted after a prompt.



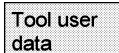
Select "Abort" to discard the changes.
No new cutting edge is created.



Save the cutting edge data with "OK".
A new cutting edge is created (if defined).



- Cutting edge user data (if configured)
Switches to the input screen form "Cutting edge user data".
Up to 10 user-specific cutting edge data are displayed here.
Enter the required settings in the table.



- Tool user data (if configured)
Switches to the input screen form "Tool user data".
Up to 10 user-specific tool data are displayed here.
Enter the required settings in the table.

Additional functions

The following functions are also available in the tool catalog:



The data of the tool are copied and a new tool is created with identical data. You are prompted to enter a name for the new tool.



The currently selected tool is deleted after a prompt. All data of the tool are lost.

Additional notes

The "Tool offset data", "Cutting edge user data" and "Tool user data" softkeys are always displayed when you edit the tool details. This allows you to move between the individual tables as required.

The tool data for tools of the catalog can be edited at any time.

5.3.9 Tool offset data in the tool cabinet



Function

You can create tool offset data in the tool cabinet. A set of data can be created for every tool in use.

The "ideal" master data defined in the tool catalog can be copied to the tool cabinet.

Advantage

Tools which have already been used can be stored in the tool cabinet before they are unloaded from the magazine. The current data, such as the remaining tool life, remain stored and can be accessed again the next time the tool is loaded.

You can also enter the tool data of tools which you plan to use in future (like the tools in a real tool cabinet).

Real tools

The **tool cabinet** contains only "real" tools.

"Real" tools (i.e. real tool dimensions, with wear, etc.) are characterized by the relevant tool "offset data".

A "real" tool is uniquely defined by its "tool name" and the associated "Duplo number". It is the "Duplo number" that assigns actual data to a "real" tool.



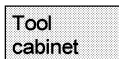
Sequence of operations

Select softkey "Tool management".

The horizontal and vertical softkey bars change.



The horizontal softkey bar is expanded.



Select softkey "Tool cabinet".

The vertical softkey bar changes again.

Create tool offset data

In order to add a tool to the tool cabinet, it must first be created in the tool catalog.

You create a real tool by defining a new Duplo number in the tool cabinet.

Proceed as follows:

- Select the desired technology, tool type and tool successively in the appropriate list fields.
- Define the Duplo number.
The tool master data are loaded into the tool cabinet. The editing functions are available.
- Use the vertical softkeys to make all the necessary changes to the cutting edge data and user data.
- Select "Abort" to discard the settings.
The tool is not created.
- Select "OK" to save the changes.
The tool is created with the current offset data.

Abort

OK



The defined tool data can be edited at any time. You can overwrite the data for the current tool or allocate a new Duplo number to create a replacement tool.

Displaying/changing tool data

The vertical softkeys can be used to display and edit the data for tools already stored in the cabinet:

Offsets

- Tool offset data (cutting edge data)
Enter the required offset values. The design of the vertical softkeys is identical to the tool catalog softkeys (see previous section under offsets).



A new cutting edge can be added to a tool at any time (even if the tool is already located in the magazine).

Cut. edge
user data

- Cutting edge user data (if configured)
Up to 10 user-specific cutting edge data are displayed here. Enter the required settings in the table.

Tool user
data

- Tool user data (if configured)
Up to 10 user-specific tool data are displayed here. Enter the required settings in the table.

Abort

- Select "Abort" to discard the changes.
The data retain their old values.

OK

- Select "OK" to save the changes.
The data are updated.

Additional functions

The "Delete" function is also available in the tool cabinet. It is not possible to copy or create a new ideal tool here (only in the tool catalog).

Delete

The currently selected tool is deleted from the tool cabinet after a prompt.

All data of the tool with this Duplo number are lost. The master data in the tool catalog are not affected (tool with Duplo number 0).

Additional notes

The "Tool offset data", "Cutting edge user data" and "Tool user data" softkeys are always displayed when you edit the tool details. This allows you to move between the individual tables as required.

A tool entered in the cabinet can be loaded to a magazine location via softkey "Tool from cabinet".

5.3.10 Job processing for tools



Area of application

Function

The function "Job processing for tools" (batch) enables the operator to

- load and unload, delete and store tools in the cabinet in one common job for several tools.
- to monitor the progress of execution.
- utilize the "Reactivate tools" function.

Parameterizable filters are used to select the tools. These enable you to create a snapshot of the tool data inventory of the NC containing all tools with the properties specified in the filter definition, e.g. all tools with particular tool status bits set, with a particular tool type, with a certain length, with particular OEM data, etc.

The search is carried out exclusively in the NC.

Job processing for tools can be initiated via the operator interface and monitored there. Loading, unloading and reactivation can take place in the background, even if the associated interface is not active.

The filter definition and some of the interface settings are carried out in the paramtm.ini and patm_xx.ini files for tool management.

Application

The machine operator can use the "Job processing for tools" function to load, unload and reactivate sets of tools according to predefined filter criteria.

- Load,
- unload and
- reactivate.

The function is available within the tool management.

The parameters and other settings for the filter criteria are made in the paramtm.ini file without a separate operator interface.

Functional description**Operator interface:**

The "Job processing for tools" function is selected via the horizontal softkey "Filter lists" from the basic states of the magazine and tool lists in the tool management.

The "Job processing for tools" recognizes three states, which are represented by different screens:

1. **Filter selection**
2. Displays the **hit list**, tool selection and start of job processing in 2 screens: Load job list and standard job list.
3. **Job execution**

Each TOA (data area for tool offsets) has a separate state. You can exit the "Filter lists" in these states and display different tool management screens or switch to other operating areas.

The next time you press the horizontal "Filter lists" softkey, the screen for the noted state is displayed.

The 2nd state "Hit list" stores the hit list and the tool selection as a snapshot.

The 3rd state "Job execution" stores the data for the selected tools and the job type. You can exit the "Filter lists" during job processing. After returning to the "Filter lists", the new status of job processing is displayed. The status of the job as a whole and the states of the individual job elements are visible.

Paramtm.ini

You can set the user rights for the softkeys involved in the paramtm.ini file (section [ACCESSLEVEL], entries "SKB...").

The filter lists are parameterized in the paramtm.ini file in the section [BatchTools].

Country-specific sections are parameterized in the "language\patm_xx.ini" file in the section [BatchTools]; "xx" stands for the 2 letters of the country code.



Precondition

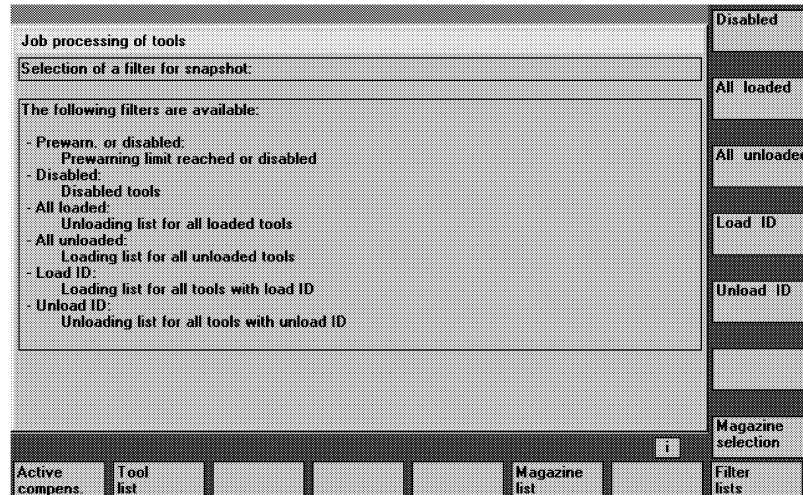
I.

Filter selection

Sequence of operations

Select the tool management

Softkey "Filter lists"



The screen provides up to 6 filters for selection by softkey depending on the configuration in paramtm.ini.

Vertical softkeys

The softkey designation "Filter 1- 6" used by way of example provides for labeling up to 6 filters. Pressing the softkey starts determining the tools that match the criteria in the NC and jumps to a 2nd screen that displays the **hit list**. The filter produces a snapshot of the data. These data are **not** updated later.

Filter 1-6

Magazine
selection

You can use the filter definition to specify whether the filter always applies to the whole TOA or is restricted only to single magazines. You can use the "Magazine selection" softkey to select a particular magazine or "All magazines" for a restrictable filter.

When you switch from a magazine list to the filter lists and no current filter or job processing is active in the TOA (you access the first screen "Filter selection"), the current magazine in the magazine list is taken as the preset value for restrictable filters.

If the situation is the same when coming from the tool list, the preset value is "All magazines".

II.

Hit list
in the "Load point" and
"Standard list" screens

The "Hit list" screen has one of 2 configuration-dependent variants:

- Load point with the functions "Load" and "Reactivate"
- Standard list with the functions "Reactivate", "Unload", "Delete", "In cabinet".

When filtering has started in screen 1 "Filter selection", the tools found in screen 2 "Hit list" are displayed in a list with one line per tool.

The data are a **snapshot** created at the time of filtering; it is **not updated later** when the data in the NC change.

Selection
of tools

No tool is selected for job processing at the outset. By positioning the cursor and pressing the Select key, you can toggle the tool selection for job processing. To change the tool selection for job processing, you can use the softkeys "Select all" and "Deselect".

Selected tools are represented in color and also indicated by a symbol in the 2nd column of the hit list on the display screen.

In the standard setting, a tool selected for job processing is displayed with a checkbox ticked off (and). The color for "Cursor" and for "Selected for job processing" is identical and corresponds to the general selection display.

When the tools have been selected fully, you can start the **job function** per softkey.

Start of job function

"Load"

Job processing of tools										
Loading list for all unloaded tools										
Tools: 45, selected: 6.										
No.	Sel.	Tool id.	Duplo	Mag	Loc.	A	P	M	U	J
31	<input type="checkbox"/>	Test31	1	0	0					
32	<input type="checkbox"/>	Test32	1	0	0					
33	<input checked="" type="checkbox"/>	Test33	1	0	0					
34	<input checked="" type="checkbox"/>	Test34	1	0	0					
35	<input type="checkbox"/>	Test35	1	0	0					
36	<input checked="" type="checkbox"/>	Test36	1	0	0					
37	<input checked="" type="checkbox"/>	Test37	1	0	0					
38	<input type="checkbox"/>	Test38	1	0	0					
39	<input type="checkbox"/>	Test39	1	0	0					
40	<input type="checkbox"/>	Test40	1	0	0					
41	<input checked="" type="checkbox"/>	Test41	1	0	0					
42	<input checked="" type="checkbox"/>	Test42	1	0	0					
43	<input type="checkbox"/>	Test43	1	0	0					
44	<input type="checkbox"/>	Test44	1	0	0					

Vertical softkeys: Cancel selection, Reactivate, Load, Update filter, Filter lists

Horizontal softkeys: Active compens., Tool list, Magazine list

Select all

Vertical softkeys

All tools in the hit list are selected for job processing.

Cancel selection

The selection for job processing is canceled for all tools in the hit list.

Loading

"Loading" is initiated for the selected tools. You are prompted to enter the destination magazine and the load point in a dialog box.

Reactivate

"Reactivate" is initiated for the selected tools. When reactivation of a tool, the actual monitoring values and the wear are reset. You can use the INI file (entry `n_ReactivatePositioningMode`) to specify (for each filter) whether reactivation is carried out "always", "never", or "on request" with magazine positioning. Depending on the setting, the dialog box prompts for positioning and the load point as appropriate to the setting.

"Unload"

Nr.	Sel.	WZ-Bez.	Duplo	Mag	Pla	A	F	G	M	V	E	D
11		Test2	1	2	1							
12		Test6	1	2	2							
13		stale_test1	1	2	3	F	G	M	V			
14		Test18	1	2	4							
15		Test22	1	2	5							
16	X	Test34	1	2	7							
17		Test42	1	2	9							
18		Test46	1	2	10							
19	X	Test48	1	3	10							
20	X	Test39	1	3	9							
21	X	Test9	1	3	2							
22		Test12	1	3	3							
23	X	Test27	1	3	6							
24		Test24	1	3	5							

Delete

"Delete" is initiated for the selected tools. Loaded tools are unloaded prior to deletion. You are prompted to specify the unloading point in a dialog box.

Unload

"Unload" is initiated for the selected tools. The tools are not deleted. You are prompted to specify the unloading point in a dialog box.

In the cabinet

Job processing is initiated "In the cabinet" for the selected tools. This function is similar to the "Delete" function; in addition, the tool data are saved in the tool cabinet database. Loaded tools are unloaded before being saved and deleted. In this case, the unloading point is queried interactively.

Update filter

The current filter and its magazine settings are used again and a new hit list produced. The tool selection for job processing is canceled completely.

Recall "A"

The current number of hits is discarded and the 1st screen "Filter selection" is displayed.





If job processing has been started and the necessary entries have been made, the display switches to the 3rd screen "Job execution".

Job execution

The screen shows the information relating to the job execution as a whole and with respect to the individual tools. The operator may halt, continue or cancel job processing and monitor the results during and after job processing.

Each tool is represented by a separate line in the list. The status of the tool is represented by the configured symbol in the 2nd column.

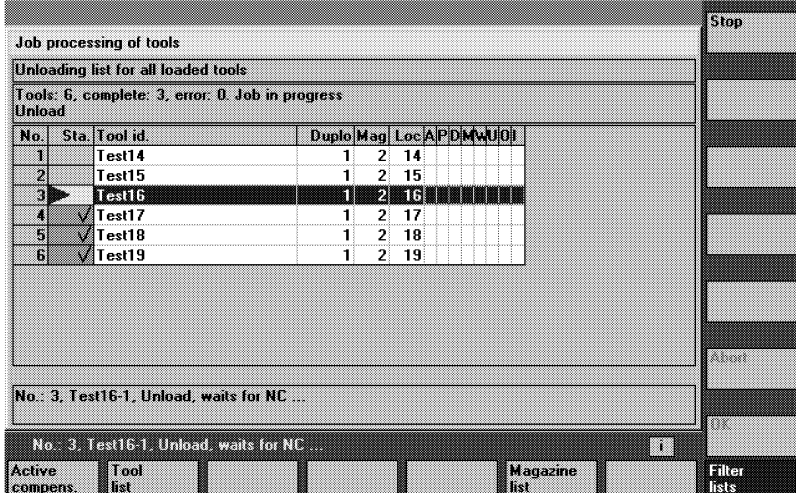
The following states are displayed by default:

- "waiting to be machined": grey surface, 
- "current tool in job processing": yellow/blue arrow, 
- "completed without errors": green field ticked off, 
- "completed with error(s)": red field with "X", 

The status of the tool on which the cursor is placed is displayed in the form of a text, for example any errors, load destination.

The state of the current tool in job processing is displayed in the message line.

If the operator does not move the cursor for a few seconds, the cursor is automatically placed on the current tool for job processing when the next processing step has been completed.



Job processing of tools

Unloading list for all loaded tools

Tools: 6, complete: 3, error: 0. Job in progress

Unload

No.	Sta.	Tool id.	Duplo	Mag	Loc	A	P	D	M	W	U	J
1		Test14	1	2	14							
2		Test15	1	2	15							
3	▶	Test16	1	2	16							
4	✓	Test17	1	2	17							
5	✓	Test18	1	2	18							
6	✓	Test19	1	2	19							

No.: 3, Test16-1, Unload, waits for NC ...

No.: 3, Test16-1, Unload, waits for NC ...

Active compens. | Tool list | Magazine list | Filter lists

Stop | Abort | OK

Hold

Vertical softkeys

Job processing is halted. Processing of the active element up to this point is either completed or canceled depending on the status and type of job.

This softkey can only be used while job processing is halted.

A rectangular softkey with a dotted background and the text "Continue" centered inside.


Any job processing that has been halted is continued.
This softkey can only be used while job processing is halted.

A rectangular softkey with a dotted background and the text "Abort" centered inside.

Any job processing that is halted is canceled. Non-completed jobs are discarded and you are returned to the "Filter selection" screen.
This softkey can only be used while job processing is halted.

A rectangular softkey with a dotted background and the text "OK" centered inside.

All information relating to detail jobs are discarded and you are returned to the "Filter selection" screen:
This softkey can only be used when all detail jobs have been completed, irrespective of whether errors occurred or not.

A rectangular area with a dotted background, representing an empty softkey.

Job processing continues running in the background when you switch from your operator interface to other tool management screens or to other operating areas.

5.4 R variables (arithmetic variables)

5.4.1 Function



Function

Variables are read and written by programs.
In this operating area, variables can be altered manually.

5.4.2 Editing/deleting/finding R variables



Function

The number of channel-specific R variables is defined in a machine data.

Range:

R0–R999 (dependent on machine data).

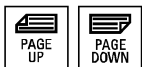
There are no gaps in the numbering within the range.



Sequence of operations

The "R variables" window appears.
The channel-specific variables are displayed.
The vertical softkey bar changes.


R
variables



You can page up and down using the "Page" keys.

Change variables:

Position the cursor bar on the appropriate input field and enter the new values.

Delete area**Delete variables:**

Displays a screen in which you can specify the R parameter range Rx to Ry to be deleted.

Delete all

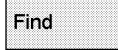
The entire R parameter range is deleted, i.e. all values are set to "0".

Abort

Inputs are discarded.

OK

Deletes the entered R parameter range.

Find**Find variables:**

An input window for a parameter number appears when you press the "Find" softkey.

Enter the R parameter number you wish to find via the numeric keypad.

INPUT

When you press the "Input key", the cursor is automatically positioned on this parameter if it exists.

Additional notes

Input and deletion of variables can be disabled via the keyswitch.

5.5 Setting data

5.5.1 Working area limitation



Function

The "Working area limitation" function can be used to limit the range within which a tool can traverse in all channel axes. This is a way of creating prohibited zones in a working area, in other words zones which the tool in question cannot enter.



Setting
data

Sequence of operations

Select softkey "Setting data".
The vertical softkey bar changes.

Working
area limitat.

Select softkey "Working area limitation".
The "Working area limitation" window opens.

Alter working area limitation:

Position the cursor on the desired field.
Enter the new values on the numeric keypad.
The upper or lower limit of the protection zone changes according to your input.



Activate the relevant working area limitation with the "Select key".



In "MDI" and "Automatic" modes, the working area limitation is not activated according to setting data within the current NC program until a "WALIMON" command is set.



Additional notes

The "Working area limitation" function can be disabled by means of the keyswitch.

5.5.2 Jog data

**G function****Function**

The feedrates must be specified in the unit determined by the G function.

G94 Feedrate in mm (inch)/min

G95 Rotational feedrate in mm (inch)/rev

Jog feedrate

Feedrate value in Jog mode

Jog continuous

- Continuous-trigger mode: The axis moves as long as the key is pressed.
- Momentary-trigger mode: The axis begins to move when the key is pressed once and continues until:
 - The key is pressed again
 - NC stop
 - Reset
 - SW/HW limit switch

Variable increment

Increment value for Jog variable increment

Jog spindle speed

The following data are displayed only if a spindle is configured:
Spindle speed in Jog mode

Spindle

Jog data for the master spindle:

- Spindle no.: Name of master spindle
- Direction of rotation: Direction of rotation of the master spindle
- Spindle speed: Speed of the master spindle in Jog mode



Setting
data

Sequence of operations

Select softkey "Setting data".
The vertical softkey bar changes.

Jog data

Select softkey "Jog data".
The "Jog data" window is opened.

Change jog data:

Position the cursor bar on the appropriate input field and enter a new value or

use the "Select" key to select a new value.

**Additional notes**

The limit values for the maximum and minimum permissible values are defined in the machine data.

5.5.3 Spindle data**Max./min.****Function**

The value entered for the spindle speed in the fields max./min. must be within the limit values defined in the machine data.

Programmed

Programmable upper speed limit (G96) for constant cutting speed.

Setting
data**Sequence of operations**

Select softkey "Setting data".
The vertical softkey bar changes.

Spindle
data

Select softkey "Spindle data".
The "Spindle data – limitation" window opens.

Change spindle data:

Position the cursor bar on the appropriate input field and enter a new value or use the "Select" key to select a new value.





Additional notes

- The limit values for the maximum and minimum permissible values are defined in the machine data.
- The "Spindle data" function is displayed only if a spindle is configured.

5.5.4 Dry run feedrate for DRY mode



Setting
data

Feedrate
DRY

Function

The feedrate entered here is used in the active program instead of the programmed feedrate when the function "Dry run feedrate" (program control) is selected in "Automatic" mode.

Sequence of operations

Select softkey "Setting data".
The vertical softkey bar changes.

Select softkey "Feedrate DRY".
The "Dry run feedrate" window is opened.

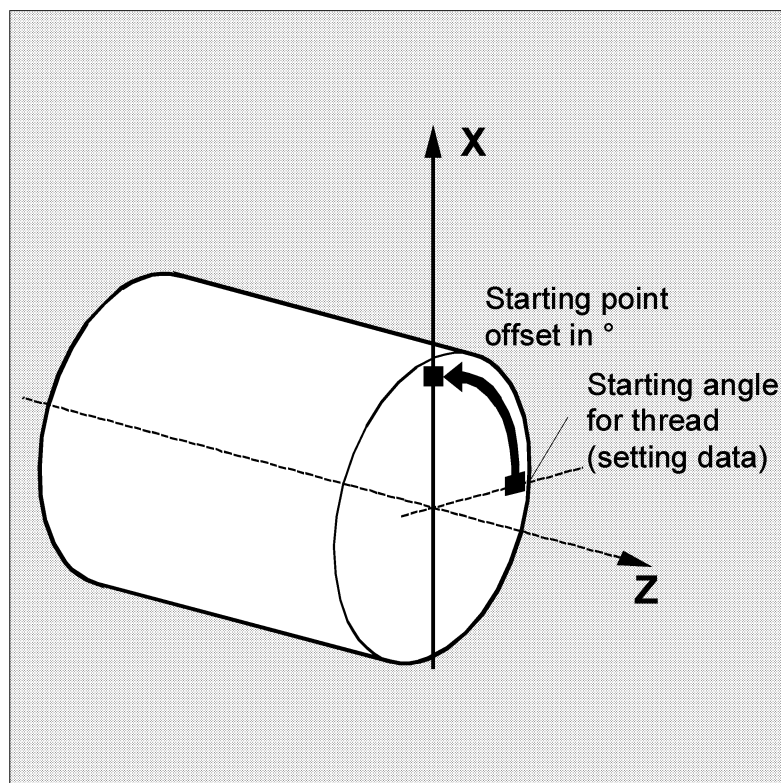
Change the dry run feedrate:
Enter a new value.

5.5.5 Starting angle for thread cutting



Function

For thread cutting, a starting position for the master spindle is displayed as the starting angle. A multiple thread can be cut by changing the angle when the thread cutting operation is repeated.



Setting
data

Starting
angle

Sequence of operations

Select softkey "Setting data".
The vertical softkey bar changes.

Change starting angle:
Select softkey "Starting angle".
The "Starting angle for thread" window opens.

Enter a new value.

5.5.6 Other types of setting data



Function

All the setting data in the control are displayed in tabular form sorted according to general (i.e. NCK-specific), channel-specific and axis-specific setting data. The table contains both the setting data on the vertical softkeys such as Working area limitation, Jog data etc., as well as special setting data such as Software cam, Oscillation, Compensation etc.



Sequence of operations

Select softkey "Setting data".
The vertical softkey bar changes.

Display setting data:

Select softkey "Misc."
The horizontal and vertical softkey bars change.

Select the type:

- The "General SD (\$SN_)" window opens.
- The "Channel-specific SD (\$SC_)" window opens.
- The "Axis-specific SD (\$SA_)" window opens.

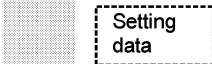
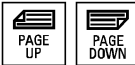
The current setting data of the corresponding type \$SN_, \$SC_ or \$SA_ are displayed.

You can page up and down using the "Page" keys.

Find setting data:

Enter the setting data you wish to find in the "Find" window (initial ID is sufficient).

If several setting data have the same initial identifier, you can display other setting data by selecting softkey "Find next".

Change setting data:

Position the cursor bar on the appropriate input field and enter a new value.

Additional notes

Data can be edited or not depending on the active access protection level.

5.5.7 Protection zones**Function**

The "Protection zones" function allows you to protect various elements on the machine, your equipment or the machined workpiece against incorrect axis motions. You can view up to 10 programmed protection zones in levels G17, G18 and G19.

For further information about protection zones, please refer to */PGA/*, Programming Guide, Advanced.

**Sequence of operations**

Select softkey "Setting data".
The vertical softkey bar changes.

Select softkey "Protection zones".
The "Working area limitations and protection zones" window opens.
The vertical softkey bar changes again.

Select softkey "Protection zone +" or "Protection zone -".
Up to 10 protection zones are displayed in succession.

Select the plane in which the relevant protection zone is located:

- Plane G17 (X,Y; infeed direction Z)
- Plane G18 (Z,X; infeed direction Y)
- Plane G19 (Y,Z; infeed direction X)

Setting data

Protection zones

Protection zone +

Protection zone -

G17

G18

G19

5.5.8 Electronic gear (SW 6.3 and higher)



Setting
data

Further
>>>

Gearbox
link

Following axis

Leading axes

- 1
- 2
- 3
- 4
- 5

Following
axis + +

Following
axis - -

Function

The "electronic gear" function allows you to move a drive axis as a following axis in synchronism with up to five leading axes. The electronic gear is encoded via the following axis and has several lead axes, each of which applies a gear ratio. The ratio to the drive axis has a linear response and is defined as a coupling factor, numerator to denominator. A gear group is defined and activated during part program execution.

For further information, please see /PGA/, Programming Guide, Advanced.

Sequence of operations

Select softkey "Setting data".
The vertical softkey bar changes.

Press the "Further >>>" softkey.
Two further softkeys appear, "Gearbox link" and "<<" for changing to the first softkey menu.

The vertical softkey menu changes to the "electronic gear" area

Coupling factor

Numerator	Denominator
:	:
:	:
:	:
:	:

Synchronization position

Following axis	Leading axis
Displays the position at which the teeth of the electronically simulated gear engage	

You can use the softkeys "Following axis + +" and "Following axis - -" to advance the following axis in both directions through the configured gear stages. The motion component of the following axis is derived from the coupling factors of the individual leading axes.

5.6 Work offset

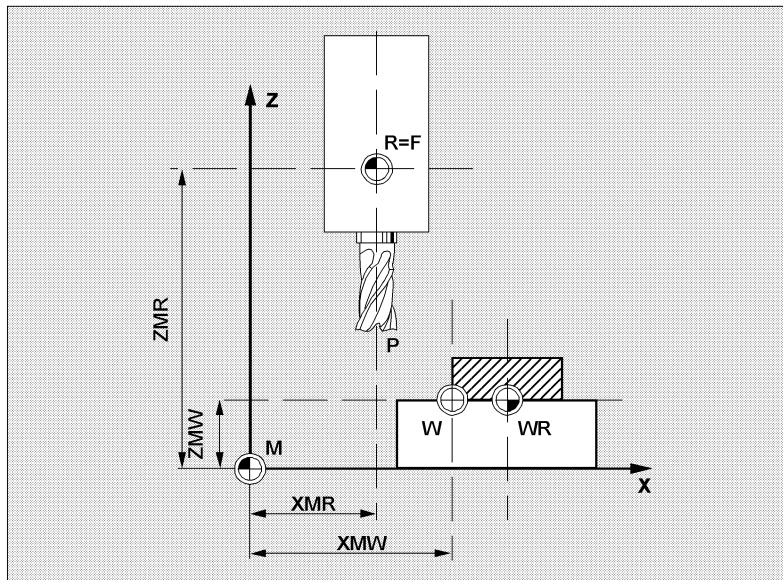
5.6.1 Function

Machine/ tool zero

The actual values are referred to the machine zero after a reference point approach. The machining program of the workpiece refers to the workpiece zero.

Machine zero and workpiece zero need not be identical. Depending on the type of workpiece and the way it is clamped, the distance between the machine zero and workpiece zero can vary. In parts program processing this is compensated for by the work offset.

Work offset on a milling machine



P	Tool setting point
W	Workpiece zero
F	Slide reference point
XMR, ZMR	Reference point coordinates
XMW, ZMW	Work offset
M	Machine zero
R	Machine reference point
WR	Workpiece reference point

Effective WO	The work offset effective in an axis $\$P_ACTFRAME=...$ is calculated from the sum of the following work offsets:
Settable WO	You can activate a settable work offset in the program you have called with G54 to G57 and other G functions or with $\$P_IFRAME=...$ Basic work offset (basic frame): displayed like a settable WO.
Programmable WO	You can use the programmable work offset $\$P_PFRAME=...$ to program an additional work offset for geometry and special axes in the parts program you have called. The values of the programmed work offsets are deleted with end of program or reset.
External WO	In addition to all the offsets which define the position of the workpiece zero, an external work offset can be overlaid by means of the handwheel (DRF offset) or from the PLC.
DRF offset	Differential Resolver Function: An NC function which generates an incremental work offset in AUTOMATIC mode in conjunction with an electronic handwheel.
Frame	Frame is the conventional term for a geometrical expression that describes an arithmetic rule, such as translation or rotation. Frames are used to describe the position of a destination coordinate system by specifying coordinates or angles starting from the current workpiece coordinate system. Possible frames <ul style="list-style-type: none">• Basic frame (basic offset)• Settable frames (G54...G599)• Programmable frames References: /PG/, Programming Guide, Advanced

Frame components**Frame components**

A frame can consist of the following arithmetic rules:

- Work offset, TRANS, ATRANS
- Rotation, ROT, AROT
- Scale, SCALE, ASCALE
- Mirror, MIRROR, AMIRROR

In the parts program, all work offsets can be deselected non-modally with G53.

**5.6.2 Changing the settable work offset (G54 ...)****Function**

`$P_UIFR[]`

This identifier can be used to alter a settable work offset in the program.

Coarse offset

The value of the coarse offset is defined for the relevant axis.

Fine offset

MD 9451 WRITE_ZOA_FINE_LIMIT is programmed to set data limits (absolute) for the fine zero point offset. The fine offset is displayed in the "Settable work offset" display.

Activation of WO via MD MM_FRAME_FINE_TRANS.

5.6.3 Global work offset/frame



Function

In addition to the settable, programmable and external work offsets, up to eight **global** work offsets/frames (**basic WO**) can be defined. This allows offsets, scales and mirrors to be defined simultaneously for all channel and machine axes.

The global work offsets (NCU-global frames) apply uniformly to **all** channels. They can be read and written from all channels. The activation is performed in the relevant channel.

Basic WO (total basic frame)

In addition, eight channel-specific basic work offsets can be defined in each channel. The global and channel-specific frames are combined to produce a total basic frame (**basic WO**).



Machine manufacturer

Recommendation:

Use the 3rd basic offset onwards for your own applications. The 1st and 2nd basic offsets are reserved for setting the actual value and the external work offset.



With global frames, no geometrical relationship exists between the axes. You cannot therefore perform rotations and you cannot program geometry axis identifiers.

The settable work offset and the basic work offset are represented in **one** table. You can edit the values in this table. You can switch between the values of the individual axes.

For **all** work offsets, you can display either the defined offsets (coarse and fine) or the specified rotations, scales and mirrors for each value.



References

/FB/ K2: Axes, Coordinate Systems, Frames

Rotation

The value of the rotation around the respective geometry axis (e.g. X, Y, Z) can be entered.

Rotation can only be programmed around geometry axes.

Scale

The scale factor can be defined for the respective axis.

Mirroring

Mirroring of the relevant axis around the coordinate zero can be activated and deactivated.

Display and edit work offsets

Work
offset

Select softkey "Work offset".

The vertical softkey bar changes.

Axes +

The display switches to the defined work offsets of the next axis.

Axes -

The display switches to the defined work offsets of the previous axis.

Offsets

Rotation
scl, mirr

You can use these softkeys to change the display mode of the currently displayed work offsets.

The display shows:

- either the absolute offsets (coarse and fine) with reference to the coordinate axes
- or a list of individual values, split into components for rotation, scale and mirror.

You can select and, if necessary, change the individual values of the work offsets in both display modes.

Basic WO

All defined basic work offsets (global and channel-specific) are displayed in a table.

The display mode can be changed by softkey (see above).

You can edit the values directly in the table.

Rotations are not possible with global frames, since no geometrical relationship exists between the axes in this case.

Settable
WO

All defined settable work offsets are displayed in a table and can be edited if necessary (select and edit).



Active WO
+ offset

Axes +

Axes -

Offsets

Rotation
scl, mirr

Change
active WO

Additional notes

- Changes to the work offsets are updated immediately on input. The entries no longer have to be confirmed separately.
- If not all work offsets are displayed in the tables, you can scroll through the table with the corresponding keys.

Display and edit active work offset

Select horizontal softkey "Active WO + offset".
The vertical softkey bar changes.

Displays the active work offset of the next axis.

Displays the active work offset of the previous axis.

You can use these softkeys to change the display mode of the currently displayed work offsets.

A table of currently active work offsets and the offsets of the selected axis are displayed. You can select and, if necessary, change the individual values in the table.

The following values are displayed:

- Global basic WO; coarse and fine (if defined)
- Channel-specific basic WO; coarse and fine (if defined)
- Settable WO; coarse and fine (G57)
- Programmable WO; G58 (TRANS), G59 (ATRANS)
- T number and D number of active tool
- G17 (geometry, wear, base).

5.6.4 Display active settable work offsets



Function

The active settable work offsets (selected from parts program or MDI) can be displayed.

Work
offset

Go to ...

Sequence of operations

Select softkeys "Work offset" and "Go to ...".
The vertical softkey bar changes.

Active
settable

The "Active settable WO" window opens.
You can alter these values if necessary.



Overview

/PGA, Programming Guide, Advanced

An overview shows all values of the active work offset and offsets (no changes possible) including a selection of tool data (T number, D number, etc.). The basic work offset and the settable work offset are displayed as the sum.

Parameter	CHAN1	AUTO	MPF DIR	TEST.MPF	
Channel active			Program running		Axis +
					Axis -
Übersicht der aktiven Nullpunktverschiebungen und Korrekturen					
	Achse	X1 [mm]	Y1 [mm]	Z1 [mm]	
	Istwert[MKS]	000.000	000.000	000.000	
	Istwert[ENS]	000.000	000.000	000.000	
Überlagerte Bewegung		0.000	0.000	0.000	
DRF-Verschiebungen		0.000	0.000	0.000	
Externe NV		100.000	100.000	0.000	
Summe NV		399.000	100.000	000.000	Change active WO
	grob	1.000	0.000	0.000	
	fein	0.000	0.000	0.000	
	Drehung [Grad]	0.000	0.000	0.000	
	Maßstab	1.000	1.000	1.000	Overview
	Spiegeln				
Akt. Werkzeug	T-Nr.	34	D-Nr. 2	Ebene G17	
	Längen	100.000	100.000	000.000	
	Radius	0.000	0.000		
	Istwert [WKS]	500.000	400.000	200.000	
		X	Y	Z	
Work offset	R variable	Setting data	Zero offset	User data	



The following values are displayed:

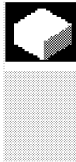
- Actual value of MCS and settable zero system
- Superimposed movements
- DRF offsets
- External work offsets
- Sum of the work offsets, calculated from the basic, settable and programmable work offsets (corresponds to the table "Change active ...")
- Data of the active tool (T number, D number with reference to the plane G17, lengths, radius)
- WCS actual value.



Additional notes

The active work offset must be changed only when the NC program is stopped. Changes are updated immediately. The work offset values in the display are updated cyclically.

5.6.5 Display active programmable work offsets



Function

The active, selected programmable work offsets (from parts program or MDI) can be displayed.

Values cannot be edited in this display.



Sequence of operations

Select softkeys "Work offset" and "Go to ...".

The vertical softkey bar changes.

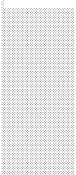
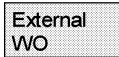
The "Active progr. WO" window opens.

Work
offset

Go to ...

Active
progr. WO

5.6.6 Display active external work offsets



Function

The active, external work offsets can be displayed.
Values cannot be edited in this display.

Sequence of operations

Select softkeys "Work offset" and "Go to ...".
The vertical softkey bar changes.

The "External WO" window is opened.

Additional notes

The 2nd basic offset is recommended as an external work offset (PLC offset) if the functionality of the standard external work offset is not sufficient.

The frame components are available if the 2nd basic offset is used as the external work offset.

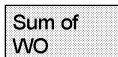
5.6.7 Display the sum of the active work offsets



Function

The sum of the active work offsets from parts program can be displayed.

Values cannot be edited in this display.



Sequence of operations

Select softkeys "Work offset" and "Go to ...".

The vertical softkey bar changes.

The "Sum of WO" window is opened.

The sum of work offsets is calculated in the following way:

Sum WO = active settable WO + active programmable WO



The values can be changed only in the "Settable work offsets" menu.
(see Section "Settable work offsets" menu)

5.6.8 Activate work offset and basic frame immediately



Function

Machine data MD \$MM_ACTIVATE_SEL_USER_DATA can be set to ensure that the work offset and basic frame are made immediately effective when the parts program is in the "Reset" state. This also occurs if the parts program was first switched to JOG status.

If the channel is in the "Reset" state, active work offset and basic frame are not activated until the parts program is continued.

Additional notes

If the function is used in the Reset state, MD \$MC_RESET_MODE_MASK must be set in such a way that settable work offset and basic frame cannot be reset on Reset.



Machine manufacturer

See machine manufacturer's specifications



/FB/ K2: Axes, Coordinate Systems, Frames



Danger

The compensation is performed the next time the parts program is started.

5.6.9 Actual value display: settable work offset system, SZS



Function

You can make a setting in the MD to define whether

- the position of the workpiece coordinate system, WCS
(= programmed position, corresponds to default setting) or
- the tool holder position of the active tool relative to workpiece zero
(settable work offset system) is to be displayed in the actual value display.



For configuring instructions see

/IAM/ IM4: Installation/Start-Up Functions HMI, Section Work Offset

5.7 User data/user variables (GUD, PUD, LUD)

5.7.1 General information



Function

User data (UD) can be defined by means of a variety of variables:

- GUD – global variables which are valid in all programs.
- LUD – local variables which are valid only in the program or subroutine in which they have been defined.
- PUD – program-global variables.

Local variables (LUD) defined in the main program are turned into program-global variables (PUD) by a setting in a machine data.

PUD variables are valid on all subroutine levels, where they can also be written and read.

The definition of user data (GUD) can be created for HMI in operating area Services (in the Definitions directory) without reinitialization.

The following rules apply:

- Definition files stored on the hard disk are not active.
- Files can be transferred to NC and activated using the "Activate" softkey.

The user memory must be configured to a large enough size before the GUD definition file is loaded to the control.

All relevant machine data have the GUD string in their names.

The display of global user data (GUD) can be locked by means of the keyswitch or a password.



5.7.2 Changing/finding user data/user variables



User data

Sequence of operations

Press softkey "User data".
The "Global user data" window is displayed.
The vertical softkey bars change.

Global user data

You can toggle between windows

- "Global user data" (GUD)
- "Channel-specific user data" and

Channel-sp user data

Program user data

• "Program user data"
Program-global variables (PUD) and local variables (LUD) are displayed.



You can scroll up and down in the list using the "Page keys".



Edit user data

Position the cursor on the user data that you wish to edit and enter a new value.

New values are automatically accepted.

GUD +

GUD -

Find user data

Press softkeys "GUD +" and "GUD -" to scroll through user data from GUD 1 to GUD 9.

GUD:

The "Select global user data" window is opened. The following values may be selected:

- 1 = SGUD (Siemens)
- 2 = MGUD (machine manufacturer)
- 3 = UGUD (machine user)
- 4 ... 9 = GD4 ... GD9 (others, e.g. grinding cycles, etc.)

The selected data are displayed in the "Global user data" window.

Find

Select softkey "Find".

Dialog window "Find user data" appears on the screen.

The data name or a character string within the name can be entered as the search target.

The cursor must be positioned on the user data to be found.

Find next

The next user data with the initial identifier searched for is displayed.



User data of types `AXIS` and `FRAME` are not displayed.

Only those local user data that still exist in the execution chain of the control are displayed.

The list of local user data for the display window is updated every time "NC Stop" is actuated. The data values are updated continuously. Before global user data definitions can be made operative in the control, it may be necessary to set machine data.



Additional notes

A description of how to define and activate user data can be found

- in Chapter "Services" Operating Area.

5.8 Displaying system variables



Function

System variables can be used for a wide range of different functions (e.g. as variables or in synchronized actions).

- View variables in a defined display (e.g. as a value or graphic characteristic) or
 - Manage views of variables
 - Display variables of a view
 - Define views of variables
- Generate a log of the response of variables during a program run
 - Define logs
 - Start a log
 - Manage a log
 - Display a log



Sequence of operations

The "System variable display" appears.

System
variable

Views of system variables		View 1 *
Actual values		View 2 *
\$AA_IW[1]	10000.144	View 3 *
\$AA_IW[2]	20089.000	View 4 *
Calculation parameters		More views
\$R[0]	100.0	Edit view
		Select view
		Variable log

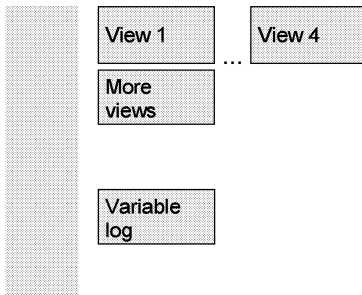
Edit
view

Select
view

* These softkeys can be named user-specifically.

This softkey opens a window in which views can be created or edited.

With "Select view" a dialog is started in which the user can select individual views or a file with several views.



The names of the views of a file are assigned to the vertical softkeys "View 1" to "More views". If a file contains more than five views, you can press softkey "More views" to select the views stored in the file in blocks of four.

The display "Logging system variables" is selected with the softkey "Variable log".

5.8.1 Processing/creating variable views



Function

You can compile your own views of variables. The display of variables can be edited with respect to

- layout (e.g. 2 columns) and
- properties (e.g. name, input limits).



Sequence of operations

The softkey "Edit view" changes to edit mode.

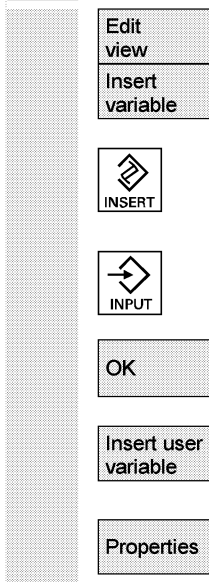
With "Insert variable", a dialog box is opened in which the required system variable can be selected from a list with the cursor and the "Edit" key. The full name of the variable appears in the info line. It is included in a new line or column in the view with the "Input" key.

The variable is inserted after the cursor position.

Confirm your input with OK.

By selecting softkey "Insert user variable", it is possible to access any existing user data by the same method.

The softkey "Properties" opens a dialog box in which the text displayed with a variable can be altered.



The following properties can be altered:

- Name
- Set another variable
- Display method (display or input)
- Display type
- No. of places after decimal point
- Input limits
- Text size (character size)
- Text alignment (left, right) and positioning (from left/from top)
- Width of input/output field

When 0 is entered, the field is automatically set to the relevant default.

You can display additional information about the variables (variable description) by pressing the "info key".

If you want to remove a system or user variable from a view, select it and press the softkey "Delete variable".

You can delete the entire contents of the display with "Delete all", which does not affect a stored view on which the screen contents are based.



Delete
variable

Delete
all

5.8.2 Managing variable views



Function

The views you create are stored and managed in file form.



Sequence of operations

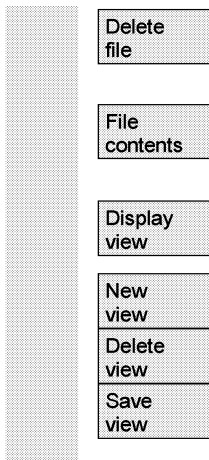
With the softkeys "Edit view" and "Manage views" you here can open a window in which files and views can be

- created
- displayed and
- deleted.

Edit view	Manage views
--------------	-----------------

New
file

File
contents



The softkey "File contents" takes you to another window in which the views of the previously selected file are displayed. With the softkey "Display view" you exit the dialog box and the previously selected view in display "Views of system variables" is displayed.

These softkeys also enable you to create views,

delete views and

save the currently displayed view under a selected name.

5.8.3 Logging system variables



Function

When variables are used in synchronized actions, it may be necessary to evaluate and log the status of actions in the interpolation cycle. This is done by writing the values selected for a log definition to a log file of defined size in the specified cycle.

Recording of synchronized action variables can be limited to the event with identification number 1.

This event records variables in the IPO cycle or multiples thereof.

- A maximum of 6 variable contents can be recorded simultaneously in the log file.
- Size for memory depth: Values between 3 and 50 KB.

The HMI interprets the contents of the log file and displays them in graphic form.

Sequence of operations

The "System variable display" appears.



The softkey "Variable log" opens the screen with the heading "Logging system variables".

Insert
variable

The softkey "Insert variable" opens a dialog box in which the system variable to be recorded can be selected.

OK

The variable name in the higher level window in the current recording list appears behind "OK".

If the list already contains 6 entries, the entry on which the cursor is positioned is overwritten.

Initial.
log

Whenever you create a log, you must first initialize the logging function in the NC by pressing the button "Initial. log".

You are informed when initialization is complete in the bottom left-hand corner of the display with the message "Logging initialized - You can now start".

Start log

or

`$A_PROTO=1`

You can start logging by pressing the button "Start log" or with the system variable `$A_PROTO=1` in the parts program.

Stop
log

or

`$A_PROTO=0`

You stop logging by pressing the button "Stop log" or with the system variable `$A_PROTO=0`.

When you have stopped logging, the log memory is automatically transferred to a file.

Manage
log

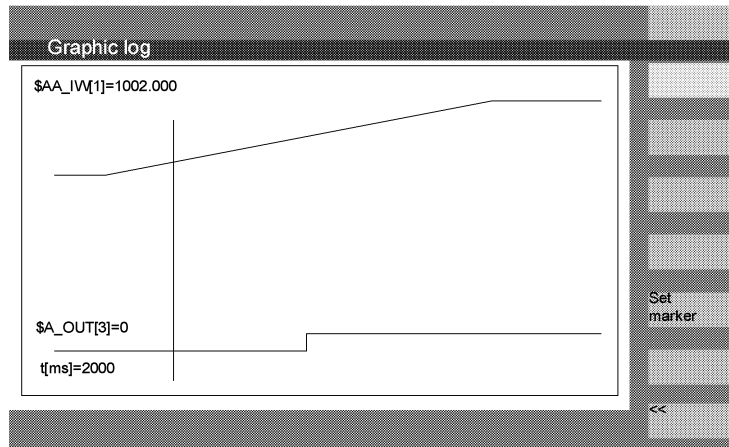
The button "Manage log" calls up a dialog box in which you can

- save a new log to a file or
- reselect a log that is already saved.

Graphic
log

With the button "Graphic log" you call up a window in which the changeover time of the variables is displayed as a curve.

Measuring times are output along the horizontal axis and the variable contents along the vertical axis. A graphic log window may contain several curves, the appropriate variable name is displayed at the top left of each curve.



Set
marker

Display "Graphic log" provides you with a zoom function in which a section that you have already selected can be expanded to cover the entire display surface.

With the softkey "Set marker" a vertical cursor line appears which is moved with the cursor left and cursor right keys.

The softkey text then changes from "Set marker" to "Set marker2" and from "Set marker2" to "Expand".

Variable values marked by the cursor line are displayed on the left of the display.

You do not need to normalize the display in any way, normalization values are calculated automatically. The characteristic is automatically normalized to values between minimum and maximum. Binary signals are represented in expanded form.

5.8 Displaying system variables



"Program" Operating Area

6.1	Program types	6-279
6.1.1	Part program	6-279
6.1.2	Subroutine	6-279
6.1.3	Workpiece	6-279
6.1.4	Cycles	6-279
6.2	Storing programs	6-280
6.2.1	HMI	6-280
6.3	Program basic display	6-280
6.4	Editing programs	6-282
6.4.1	Text editor	6-282
6.4.2	Undo and redo in the editor (SW 6.3 and higher)	6-283
6.4.3	Selective program protection: *RO*	6-284
6.5	Free contour programming	6-285
6.5.1	General	6-285
6.5.2	Programming a contour	6-286
6.5.3	Contour elements	6-290
6.5.4	Graphic representation of the contour	6-291
6.5.5	Input screen forms for parameterizing the contour elements	6-292
6.5.6	Specifying contour elements in polar coordinates, Closing the contour	6-295
6.5.7	Undercuts in turning technology	6-300
6.5.8	Help	6-303
6.5.9	Parameter description of straight line/circle contour elements	6-304
6.5.10	Examples of user-defined contour programming	6-305
6.5.11	Cycle support	6-308
6.6	Program simulation	6-309
6.6.1	Simulation user interface	6-310
6.6.2	Simulation settings	6-321
6.6.3	Downtime settings	6-325
6.6.4	Display and colors	6-326
6.6.5	Section by section simulation (SW 5.2)	6-327
6.6.6	Simulation for orientable toolholder	6-328
6.7	Simulation with external network drive	6-329
6.8	Program management	6-330
6.8.1	Overview	6-330
6.8.2	NC file types and directories	6-331
6.8.3	New workpiece/parts program	6-332
6.8.4	Storing setup data	6-335
6.8.5	Selecting a program for execution	6-336

6.8.6	Loading/Unloading a program	6-339
6.8.7	Program management	6-340
6.8.8	Copying/inserting	6-341
6.8.9	Deleting	6-344
6.8.10	Renaming	6-345
6.8.11	Enabling.....	6-347
6.8.12	Log.....	6-348
6.9	Accessing an external network drive/computer.....	6-349

6.1 Program types

6.1.1 Part program



A parts program consists of a sequence of instructions to the NC control. In its entirety, this sequence affects the production of a specific workpiece or a particular machining process on a given blank.

6.1.2 Subroutine



A subprogram is a sequence of instructions in a parts program which can be called repeatedly with different defining parameters. Cycles are a type of subprogram.

6.1.3 Workpiece



1. A workpiece is a part to be produced/machined by the machine tool.
2. In the HMI, a workpiece is a directory in which programs and other data for machining a particular workpiece are stored.

6.1.4 Cycles



Cycles are subprograms for the execution of a recurring machining process on the workpiece.

6.2 Storing programs

6.2.1 HMI

Programs can be saved in the NC memory or on hard disk. The currently available memory space is displayed in the dialog line. Cycles can also be stored in the NC Flash File System.

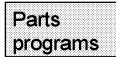
6.3 Program basic display

The Program basic display contains a complete overview of all workpiece and program directories.

The screenshot displays the 'Program basic display' interface. At the top, it shows 'Program CHAN1' and 'Mda'. Below this, there are buttons for 'Channel reset' and 'Program aborted'. The main area is titled 'Program overview' and contains a table with the following data:

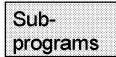
Name	Type	Loaded	Length	Date	Enable
HUG0	WPD			01.06.2000	x
WKS	WPD			01.06.2000	
WKS	WPD			01.06.2000	

Below the table, there are status indicators: 'Free memory:', 'Hard disk:', and 'NCU:'. A prompt says 'Press INPUT key for program overview'. At the bottom, there are several buttons: 'Work-pieces', 'Parts programs', 'Sub-programs', 'Standard cycles', 'User cycles', and 'Manufact. cycles'. On the right side of the interface, there is a vertical menu with buttons: 'New...', 'Load HD->NC', 'Unload NC->HD', 'Simulation', 'Manage programs...', 'Select', and 'Save setup data'.



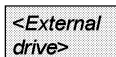
Horizontal softkeys

An overview of all parts programs (main programs) stored in the selected directory is displayed.



An overview of all subprograms stored in the selected directory is displayed.

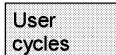
Subprograms are processed in the same way as described for "Process main programs".



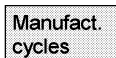
If external drives are configured, up to three softkeys can be found in the softkey menu with the configured drive/computer designation. See softkey assignment below.



Select softkey "Standard cycles" to display a list of the cycles that Siemens has added. See softkey assignment below.



Select softkey "User cycles" to display a list of the user cycles that you have added.



Select softkey "Manufacturer cycles" to display a list of the cycles that the manufacturer has added.




After pressing the "ETC" key, you can display a list of the programs/data stored in the clipboard by pressing softkey "Clipboard".




After pressing the "ETC" key and the softkey "Log", you can display the log of error messages for errors which occurred during "Copy", "Rename", "Load", etc.

Softkey assignment

If external network drives are configured for the control, the softkeys for these drives replace the softkeys "Standard cycles", "User cycles" and "Manufacturer cycles". If four softkeys have been configured, press the "ETC" key to access the fourth key on the continuation of the bar. If network drives are configured, the softkeys "Standard cycles", "User cycles" and "Manufacturer cycles" also appear in the continuation of the bar after pressing the "ETC" key.



Vertical softkeys

Creates a new file for a workpiece/parts program.



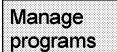
The highlighted program (or programs) is (are) loaded from the hard disk to the NC memory.



The highlighted program is unloaded from the NC memory to the hard disk.



With the function "Simulation" you can display axis movements graphically and simulate the results of machining on the screen. See also 6.6.



With the softkey "Manage programs" you can manage the functions "New", "Copy", "Insert", "Delete", "Rename" and "Enable".



The name of the selected workpiece is displayed on the screen in field "Program name" at the top. The program is also loaded and selected for processing.



When you branch to a workpiece, softkey "<<" automatically appears to allow you to return to the main menu.

6.4 Editing programs

6.4.1 Text editor



Chapter 2, "General operating sequences"

- The editor only displays those characters that can be entered on the operator panel front keyboard.
- A parts program opened by the editor cannot be simultaneously started in the NC (enable is canceled); an alarm (14011) is indicated in this case. If the control is switched off while the editor is open, the enable might have to be set manually.
- When you exit the editor after editing cycles in the NC and if these cycles are also stored in the Flash File System, then an activity symbol (fan symbol) is displayed while the data is being copied into the Flash File System. See also 6.2.1.

6.4.2 Undo and redo in the editor (SW 6.3 and higher)



Function

Editing

When editing a part program or a text file, you can undo complete blocks in the editor. Once you undo a series of blocks, you can redo the same number of blocks.

The two functions are called up by pressing the following key combinations:

- "Ctrl" + "Z" for "undo"
- "Ctrl" + "Y" for "redo"



Sequence of operations

"Ctrl" + "Z"

Press the key combination "Ctrl" + "Z"

All changes in the entire block are undone. Repeat the key combination to undo the previous change. The "CR" symbol at the end of the block is also considered. This procedure can be repeated until the last change or the set number of changes has been reached.

"Ctrl" + "Y"

Press the key combination "Ctrl" + "Y"

The last changes undone in the editor are redone. This procedure can be repeated until the last change is redone or the set number of maximum changes has been reached.



The default number is 10 steps; this setting can be changed.

Setting the number to zero deactivates the function.

The "undo" and "redo" functions affect files in the operating memory, not NC files or sub-areas.



References: /IAM/, IM4 Installation and Start-Up, Chapter 5
Functions/Parameters

6.4.3 Selective program protection: *RO*



Function

In programs written with program templates or when using the programming support function, certain machine-specific code lines can be protected against editing.

A "Read Only" identifier is inserted as a comment (";*RO*") at the end of these code blocks. The ASCII Editor identifies these blocks and displays them in "Read Only" text colors (gray text) and rejects any attempt to modify them.

```

Editor          \MPF.DIR\HARR.MPF          1
:Test-Programm
;Nr 1
;141197
N100 g01 x50 f200
lab1:
N120 g01 x100 f500          ;*RO*
N130 x10                   ;*RO*
N140x20                    ;*RO*
N150 x40                   ;*RO*
gotob lab1
;Test 14.11.97
N200 g01 x200
N210 x150
N220 x40
N230 x200
  
```

Irrespective of the above setting, the user can identify protected program sections by the "Read Only" identifier (";*RO*").

Any attempt to edit a program section protected by a "Read Only" identifier is rejected with the message "Block cannot be written".

Additional notes

When creating a program template, remember to position the "Read Only" identifier directly at the end of the block.

6.5 Free contour programming

6.5.1 General



Function

User-defined contour programming is a support tool for the editor. The contour programming function enables you to create simple or complex contours.

An integrated contour calculator calculates any missing parameters for you, provided that they can be computed from other parameters. You can concatenate contour elements. Contour transition elements "radius" and "chamfer" are also provided to help you chain contour elements.

The programmed contours are transferred to the edited parts program.

The contour calculator (geometry processor) includes **channel-specific** display machine data in software versions SW 6.2 and higher. As it does not know which channel the program is being created for, it always analyzes the display machine data of the **1st channel**.

In software versions SW 6.3 and higher, allowances parallel to the contour, as needed for grinding, can be created by free contour programming and modified again if necessary.

The following are contour elements:

- Start point
- Straight line (planar, longitudinal, inclined)
- Circular arc
- The pseudo-contour element pole is available in HMI Advanced SW 6.2 or later. Straight lines and circular arcs can also be defined by polar coordinates in reference to a pole. See Subsection 6.5.6.



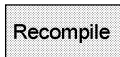
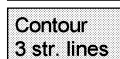
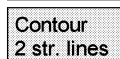
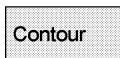
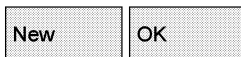
Additional notes

1. The valid geometry axes in the first channel are determined and used in the parts program.
2. The contour editor uses the last programmed axis position for the starting point, without allowing for previously valid G functions.
3. You must specify the side for the contour allowance, e.g. "right" or "left".

6.5.2 Programming a contour



New contour



Sequence of operations

Use softkeys "Workpiece" and "Parts program" in the Program operating area to select an existing program and press the "Input" key or select softkey "New" to open a new parts program, then enter a name and confirm with "OK". You are now back in the ASCII editor.

The softkey "Contour" displays a vertical softkey menu bar with the following options:

The geometry processor for generating a contour chain from contour elements is activated.

Preparatory cycles are displayed with help displays for creating straight lines and transition elements, if applicable.

You can edit an existing contour by selecting softkey "Recompile", making sure that the editor cursor is positioned inside the relevant contour.

Notice

When you recompile, only those contour elements are generated that were created with free contour definition programming, and only those texts are recompiled that were appended via the input field "Free text input". Any changes you made directly in the program text are lost.

However, you can insert and edit user-defined text afterwards, which will not be lost.

Define the starting point

The input screen form for the contour **start point** is displayed.

The input field with the input focus has is indicated by the yellow background color. Once the input is acknowledged with "Accept element" or "Abort", you can navigate around the contour chain using the \uparrow , \downarrow arrow keys. The current position in the chain is indicated in red.

When entering a contour, begin at a position which you know and enter it as the starting point.

The default tool axis or program level (defined in the machine data) can be changed for machines with more than two geometry axes.

The associated starting point axes are automatically adjusted.

In SW 6.2 or later, both the contour starting point and a pole for contour programming can be defined in polar coordinates. 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. See Subsection 6.5.6.

Accept
element

Select softkey "Accept element" to store the starting point. You can add the next element by selecting the appropriate softkey:

Alternative

The approach motion to the starting point can be changed from G0 (rapid traverse) to G1 (linear interpolation) via the new field "Approach starting point". You can define a specific feedrate for G1 via the field "Free text input", e.g. `G1 F0.3`.

1

0

...

Position the cursor on the "Facing axis dimension" field and click on the field using softkey "Alternative" (or with the "Selection" key) repeatedly until the dimension you require is displayed.

Enter the values for the starting point.

Accept
element

The values you have entered are accepted when you select "Accept element", you can add the next element by selecting the appropriate softkey.

Define contour element

Beginning at the starting point, enter the first contour element, e.g. a straight line. Input all the data specified on the workshop drawing: Length of straight lines, end position, transition to following element, angle of pitch, etc.

All
parameters

Select softkey "All parameters" to display a selection list of all the parameters for the 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 them from the settings of other parameters.

The contour is always machined in the programmed direction.

INPUT

You can select an existing contour element with "INPUT". A new contour element is inserted after the cursor when you select one of the contour elements on the horizontal softkey menu, the input focus is then switched to the parameter input on the right of the graphic display. You can navigate around the contour chain again after selecting "Accept element" or "Abort". The following contour elements (example for turning: G18) are available for the definition of contours:

Horizontal
line



Straight line in horizontal direction. Enter the end point of the straight line (incremental/absolute can be selected with softkey "Alternative"), program the transition to the following element and then press softkey "Accept element".

Vertical
line



Straight line in X direction.

Any
line



Oblique line in X/Z direction. Enter the end point of the line as a coordinate or angle.

Arc



Arc with any direction of rotation

Contour transition elements

A transition element can be used whenever there is a point of intersection between two neighboring elements which can be calculated from the input values.

You can choose between a radius **RD** and a chamfer **FS** to be inserted as the transition between any two contour elements. The transition element is always added at the end of a contour element. You select transition elements in the parameter input screen form for the relevant contour element.

Abort

When you select "Abort", the contour element values are discarded and you return to the basic display. The input focus switches back to the contour chain.

Delete
value

The values for the element are deleted.

Parameters on gray background

These parameters have been calculated by the control system. You cannot alter them.

When the programmed parameter input fields (white background) are altered, the control calculates new data, which are then immediately displayed again in the input screen form.

Input value is already calculated

With some contours, the control system may already have calculated an input value from other settings.

Problems may then arise if the control-calculated value does not tally with the workshop drawing. In this case, you must delete the settings from which the control has automatically calculated the input value. You can then enter the setting exactly from the workshop drawing.

Free text input

Under "Free text input" you can enter a comment that is inserted in the program at the end of the contour (e.g. specifying the technology, feedrate, M function).

Contour allowance (SW 6.3 and higher)

By selecting "Cont. allow.", you can specify the allowance parallel to the contour and the side of the contour on which the allowance applies. It is displayed as an allowance in the graph window.

It is possible to change the allowance and the parameters from the original contour. These changes are applied to subsequent machining operations in the part program, e.g. for grinding.

6.5.3 Contour elements**Contour chain****Function**

The elements of the contour and pole, if applicable, are displayed symbolically in the sequence in which they were programmed in a contour chain next to the graphic window.

6.5.4 Graphic representation of the contour

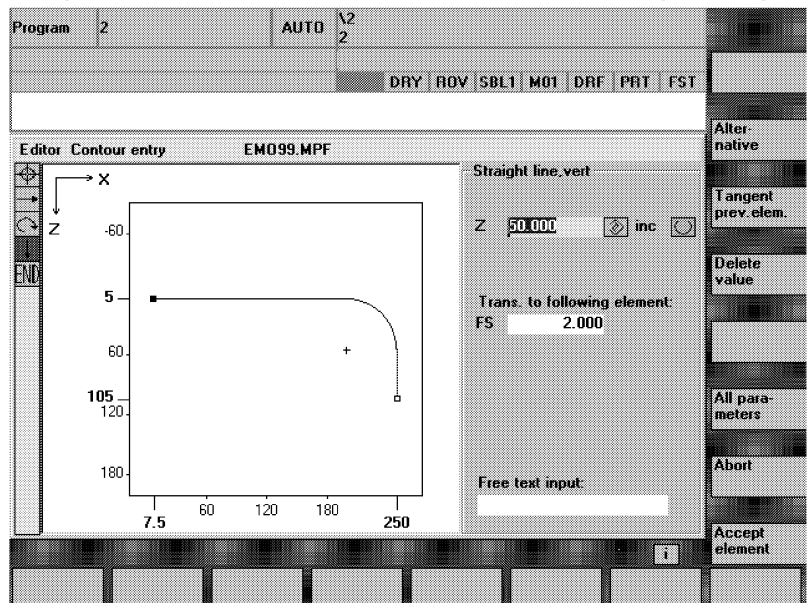


Function

The graphics window displays the progress of the contour chain as you are parameterizing the contour elements.

The element you have selected is displayed in red in the graphics window.

Navigation within the contour is described in "Contour programming".



The created contour element can be displayed in various line types and colors depending on its status:

	Color/line	Meaning
	Black	Programmed contour
	Red	Current contour element
	Green	Alternative element
	Black solid line	Element is fully defined
	Light blue line	Element is partially defined
	Dashed green line	Alternative element

The current status of the contour is displayed insofar as 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 still need to be entered. Check the contour elements you have already programmed. You may have forgotten to enter all the data you know.



Contour allowance (SW 6.3 and higher)

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.

Here, you enter the complete description of the allowance parallel to the contour and the side of the contour on which the allowance applies.

6.5.5 Input screen forms for parameterizing the contour elements



Function

Input screen forms are handled in principle according to the same procedure already described for handling contour elements STRAIGHT LINE, CONE and CIRCLE.

To help you to program a contour, the following softkeys are also available:

The "Tangent prev. elem." softkey presets the angle α_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 (α_2) is set to zero degrees.

Tangent to preceding element

Tangent
prev. elem.

Displaying additional parameters

All
parameters

Alternative

If your drawing contains further data (dimensions) for a contour element, select softkey "All parameters" to extend the range of input options for the element.

Softkey "Alternative" is displayed only in cases where the cursor is positioned on an input field with several selectable settings.

Selecting a dialog

Select
dialog

Select
dialog

Accept
dialog

Some constellations of parameters can produce several different contour characteristics. In such cases, you will be asked to select a dialog. By clicking the softkey "Select dialog", you can display the available selection options in the graphic display area.

Select softkey "Select dialog" to make the correct selection (solid line) and confirm your choice with softkey "Accept dialog".

Changing a dialog selection

Change selection

Select dialog

Accept dialog

If you have already chosen a dialog and want to change it, you must first select the contour element for which the dialog was required. Both alternatives are displayed again when you select softkey "Change selection".

You can select another dialog.

If the selection has become unnecessary as a result of other input values, you will no longer be requested to select a dialog!

Clearing a parameter input field

Delete value

You can delete the value in the selected parameter input field with the DEL key or softkey "Delete value".

Saving a contour element

Accept element

If you have entered the available data for a contour element or selected the desired contour by means of softkey "Select dialog", select softkey "Accept element" to store the contour element and return to the basic display.

You can then program the next contour element.

Adding a contour element

Accept 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 form for that element. Confirm your inputs with softkey "Accept element".

Selecting a contour element

INPUT

Position the cursor on the desired contour element in the contour chain, and select it with the "Input" 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 display area, i.e. the color of the contour element changes from black to red.

Changing a contour element

INPUT

Using the cursor keys, you can select a programmed contour element in the contour chain. The "Input" key displays the parameter input fields. These can now be altered.

Inserting a contour element

A rectangular softkey with a light gray background and a thin black border, containing the text "Accept element" in a sans-serif font.

Use the cursor keys to select the contour element **behind** which you wish to insert another element.
Then select the contour element to be inserted from the softkey menu.

After you have parameterized the new element, confirm the insert operation by selecting softkey "Accept element".

Depending on the new state of the contour, the contour elements below are updated automatically or when the cursor arrow is played on them. Contour elements that appear after the insertion point and are not updated are displayed in green in the contour chain.

Deleting a contour element

A rectangular softkey with a light gray background and a thin black border, containing the text "Delete element" in a sans-serif font.

Use the arrow 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 the softkey "Delete element" and confirm the query.

Undoing an input

A rectangular softkey with a light gray background and a thin black border, containing the text "Abort" in a sans-serif font.

By selecting softkey "Abort" you return to the basic display, **without** transferring the last edited values to the system.

6.5.6 Specifying contour elements in polar coordinates, Closing the contour



Function

When defining the coordinates of contour elements in previous sections, positions were entered using the Cartesian coordinate system. In HMI Advanced SW 6.2 or later, positions can be defined using polar coordinates as an alternative.


When programming contour definitions, a pole can be defined at any time before the polar coordinates are used.

Programmed polar coordinates later relate to this pole. The pole is modal and can be respecified at any time. It is always entered in absolute Cartesian coordinates. The geometry processor converts values entered as polar coordinates into Cartesian coordinates.

Programming in polar coordinates is only possible **after** a pole has been entered. 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.

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.

Entering polar coordinates

More

The softkey "More" in the basic plane of the contour programming accesses the "Pole" screen form and the softkey "Close contour".

Pole

The pole can only be entered in absolute Cartesian coordinates. The softkey "Pole" is also present in the starting point screen form. 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.

Close contour

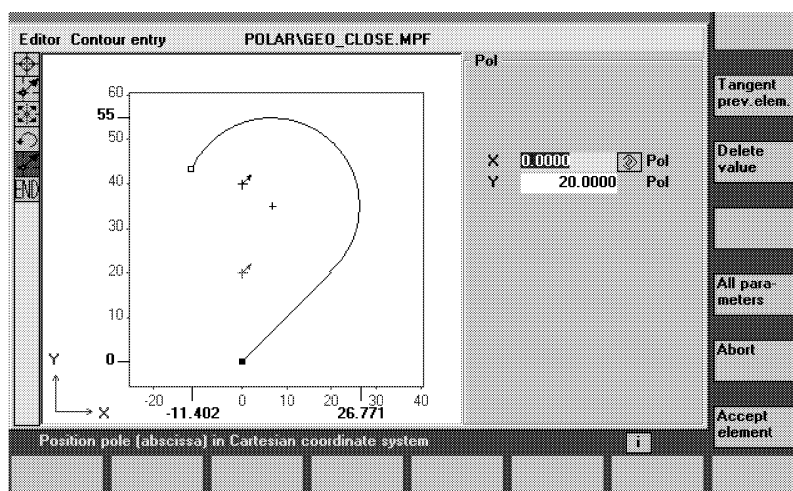
The contour is closed by a straight line between the last entered contour point and the starting point.

Additional 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, input key, enter radius/chamfer, accept element. The result then corresponds exactly to what would occur if the linked element were to be entered with the radius/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 at the closing point.



Input switchover:
Cartesian/polar

Once a pole has been set, whether inserted in the starting point or later, the contour elements can be entered as:

- Circular arcs
- Straight lines/(inclined)

or as polar. For the Cartesian/polar switchover, additional toggle fields are displayed for "Straight line, any" and "Circular arc" in both the basic contour input view and the view with "All parameters".

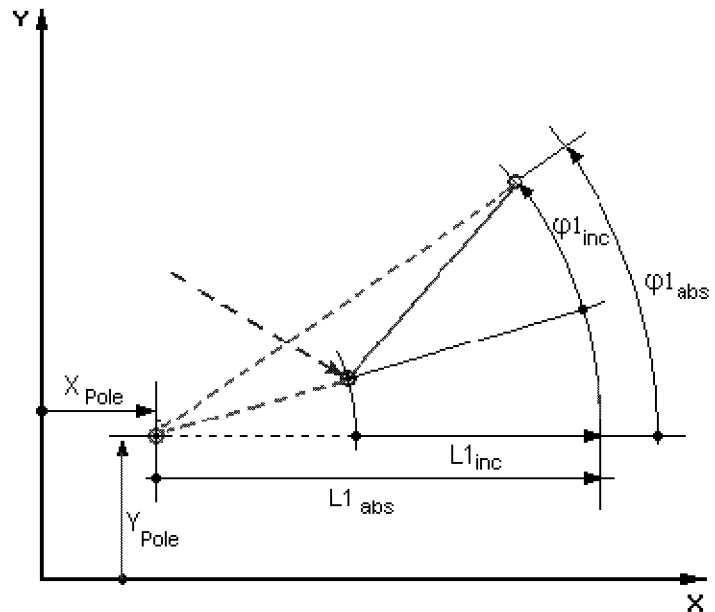
A toggle field is not displayed if a pole does not exist. Input fields and display fields are then only available for Cartesian values.

Absolute/incremental
input

In the case of "polar", absolute and incremental polar coordinates can be entered. 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 $0^\circ \dots \pm 360^\circ$.

For an absolute input, the angular reference is based on a horizontal axis on the working plane, e.g. X-axis for G17. The positive direction of rotation runs in the counterclockwise direction of rotation. If there are several input poles, the definitive pole is 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 of the end point for the preceding element to the pole plus the length increment that was entered.

The increment can be positive or negative.

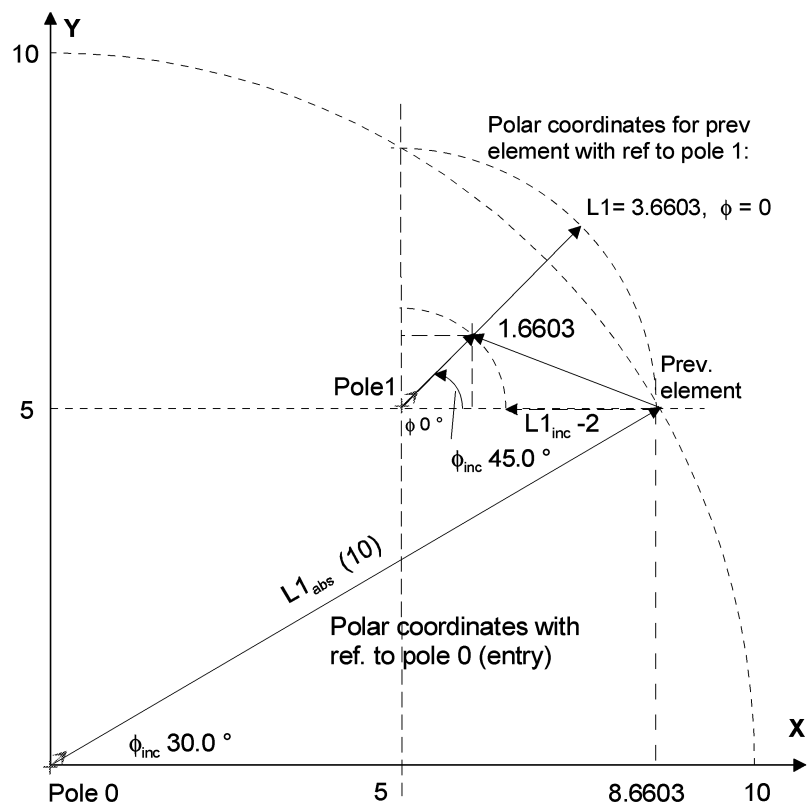
The absolute angle is calculated accordingly using the absolute polar angle of the preceding element plus the angular increment. Here it is not necessary for the preceding element to have been entered as polar.

The geometry processor of the contour programming always converts the Cartesian coordinates of the previous end point into polar coordinates using the definitive pole. This also applies if the preceding element was entered as polar, and could relate to a different pole, if a pole was set in the meantime.

6.5 Free contour programming

Pole change example

Pole:	$X_{\text{Pole}} = 0.0,$	$Y_{\text{Pole}} = 0.0$	(Pole 0)
End point:			
$L1_{\text{abs}} = 10.0$	$\varphi_{\text{abs}} = 30.0^\circ$	Calculated Cart. coordinates	
		$X_{\text{abs}} = 8.6603$	$Y_{\text{abs}} = 5.0$
New pole:			
$X_{\text{Pole1}} = 5.0$	$Y_{\text{Pole1}} = 5.0$	(Pole 1)	
		Calculated polar coord. for prev. element	
		$L1_{\text{abs}} = 3.6603$	$\varphi_{\text{abs}} = 0.0^\circ$
Next point:			
$L1_{\text{inc}} = -2.0$	$\varphi_{\text{inc}} = 45.0^\circ$	Absol. pol. coord. for current element	
		$L1_{\text{abs}} = 1.6603$	$\varphi_{\text{abs}} = 45.0^\circ$
		Calc. Cart. coordinates	
		$X_{\text{abs}} = 1.1740$	$Y_{\text{abs}} = 1.1740$



Circular arc polar input example

You want a counterclockwise arc to travel from the starting point X67.5 Y80.211 around the center point I=50, J=50 (according to the pole) and the radius 34.913 to the end point with an absolute polar angle of 200.052 degrees.

The inputs for the circular arc are as follows:

Parameter	Value	Unit/Type
R	34.9132	abs
L1	34.9130	abs
φ	200.0520	abs
I	50.0000	abs
J	50.0004	abs

Trans. to following element:
FS 0.0000

Free text input:
: Kreisbogen links

End point distance to pole absolute: i

The code created in the parts program is as follows:

```

; 200,052 , r 34,913
; I=50, J=50
G17 G90 DIAMOD ;*GP*
G0 X67.5 Y80.211 S1000 M04 F200 ; Startpunkt kartesisch ;*GP*
G3 X17.2034 Y38.0293 I=AC(50) J=AC(50.0004) ; Kreisbogen links ;*G
P*
;Mein Kommentar ;*GP*
=eof=

```

Edit Contour Drilling Milling Turning Simulation Re-translate

Compare: **References:** /PG/, Programming Guide Fundamentals
"Circular programming with polar coordinates"

6.5.7 Undercuts in turning technology

Supplementary conditions The form E and F undercut and form DIN 76 and general thread undercut functions are only activated when turning technology is enabled. The display machine data MD 9020: TECHNOLOGY is analyzed to determine the technology.

Form E and F undercuts as well as thread undercuts are only available if level G18 is set. Undercuts are only permitted 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: 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: DIAMETER_AX_DEF does not contain a name that conforms 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° angles. A tolerance of +/- 3° is required here, so that conical threads are also possible (however, the undercuts do not meet the standard in this case).

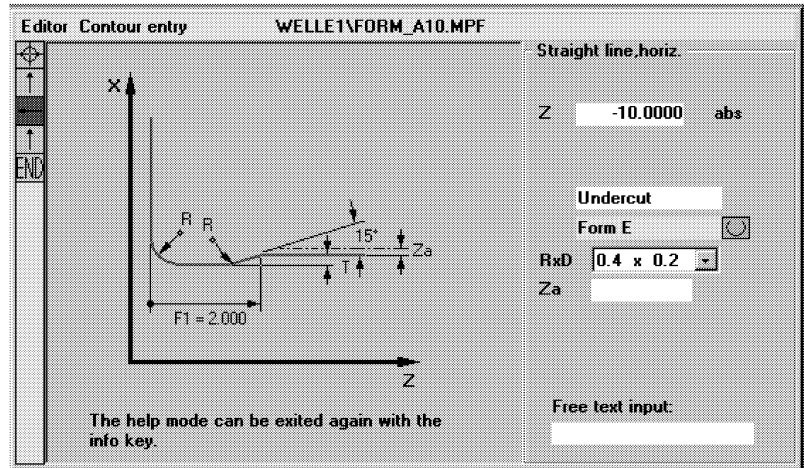
Operation

When the operator focus is on "Trans. to following element", use the Select key or softkey "Alternative" to select Undercut.

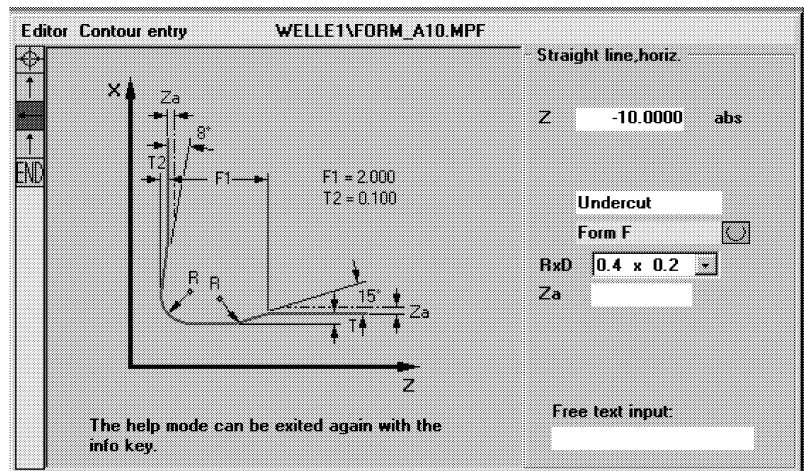
When the focus is on the following field, the undercut form can be defined. The Select key and softkey "Alternative" can be used to select the following options:

- Form E
- Form F
- DIN 76 thread
- General thread

Form E



Form F



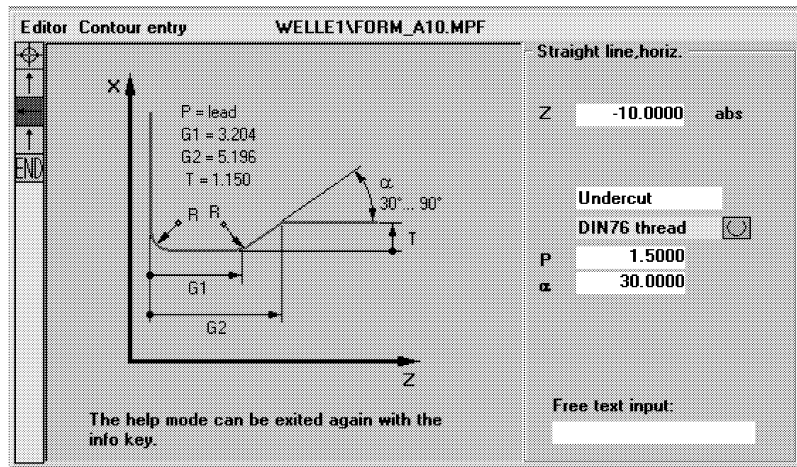
Operation (continued)

If the undercut form is specified, the RxD field can be opened with the input key and the desired coordinates can be selected with the cursor keys. The desired coordinates can also be selected by repeatedly pressing the Select key.

If the diameter is already known when selecting the undercut, the list box displays a suggested value. **Za** is the machining allowance (grinding allowance) permitted according to DIN 509.

6.5 Free contour programming

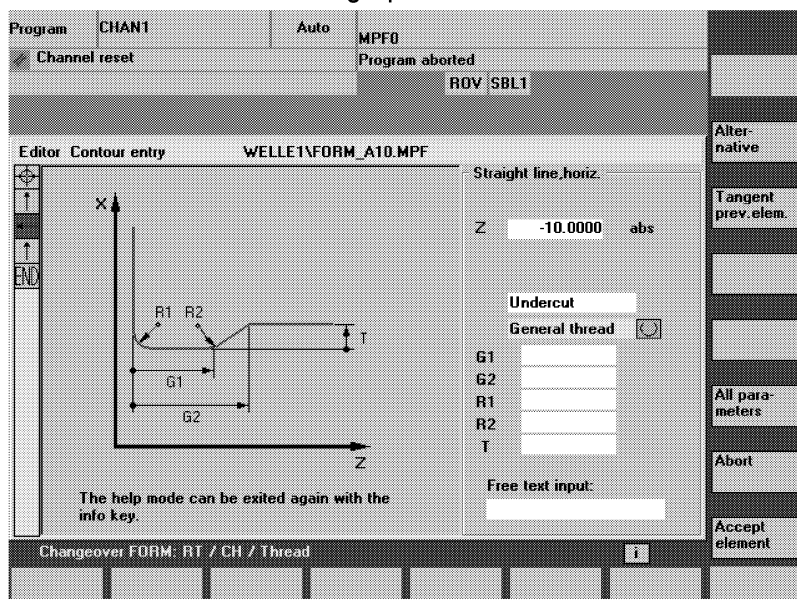
DIN thread



The characteristic size of the thread pitch for standard thread undercuts 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 arc-in 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.

General thread

Following the thread undercut according to DIN (Fig. above), each special undercut, e.g. inch thread, can be generated with the "General thread" undercut. The following inputs can be made:



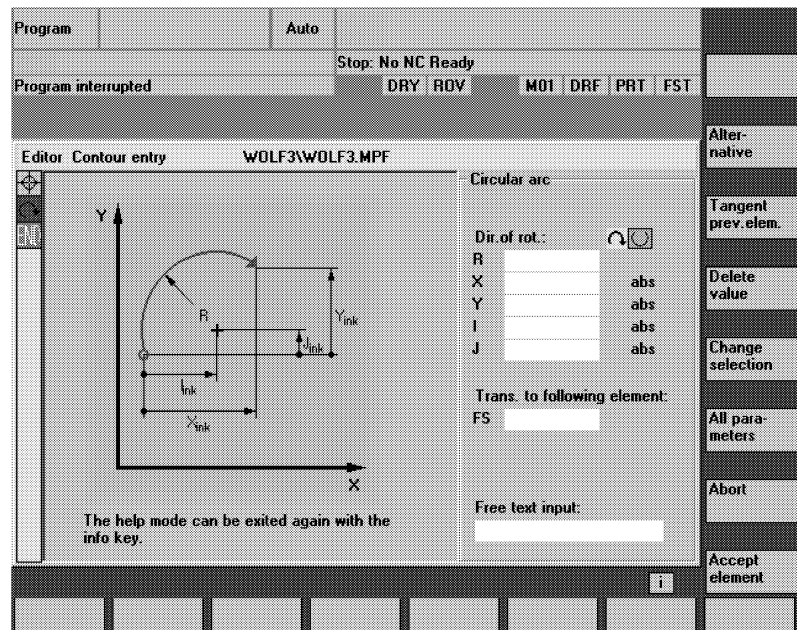
6.5.8 Help



Function

When you enter parameters you can call up a help screen with the Info key which graphically represents the parameters you are entering. The help screen that appears depends on the cursor position in the parameter display.

The help screen covers up the display graphic.



If you press the Info key again the help screen is closed and the display graphic is activated again. The help screens displayed correspond to the selected coordinate system. The axis names are derived from the current geometry axis names.

Help screens are displayed for the following entries:

- Start point
- Straight vertical line
- Straight vertical line, angle entry field
- Straight horizontal line
- Straight horizontal line, angle entry field
- Straight line, any
- Straight line any, angle entry field
- Circle
- Circle, angle entry field
- Radius/chamfer

6.5 Free contour programming

6.5.9 Parameter description of straight line/circle contour elements



Parameters	Contour element "Straight line"
X absolute	Absolute end position in X direction
X incremental	Incremental end position in X direction
Y absolute	Absolute end position in Y direction
Y incremental	Incremental end position in Y direction
L	Length of line
$\alpha 1$	Pitch angle referred to X axis
$\alpha 2$	Angle to preceding element; tangential transition: $\alpha 2=0$
Transition to following element	Transition element to next contour is a chamfer (FS) Transition element to next contour is a radius (R) FS=0 or R=0 means no transition element



Parameters	Contour element "Circle"
X absolute	Absolute end position in X direction
X incremental	Incremental end position in X direction
Y absolute	Absolute end position in Y direction
Y incremental	Incremental end position in Y direction
$\alpha 1$	Starting angle referred to X axis
$\alpha 2$	Angle to preceding element; tangential transition: $\alpha 2=0$
$\beta 1$	End angle referred to X axis
$\beta 2$	Arc angle of circle
Direction of rotation	In clockwise or counter-clockwise direction
R	Radius of circle
I	Position of arc center point in X direction (abs. or incr.)
J	Position of arc center point in Y direction (abs. or incr.)
Transition to following element	Transition element to next contour is a chamfer (FS) Transition element to next contour is a radius (R) FS=0 or R=0 means no transition element



Machine manufacturer

The names of the identifiers (X or Y ...) are defined in the machine data where they can also be changed.

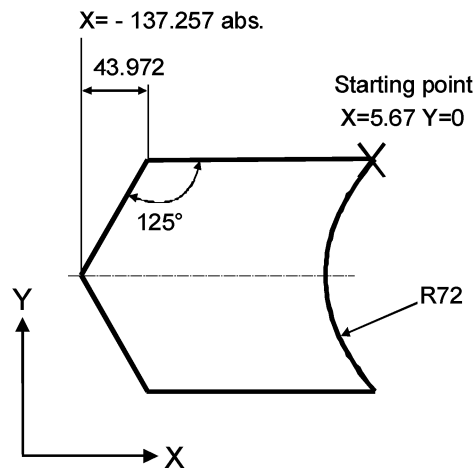
6.5.10 Examples of user-defined contour programming



Example 1

Starting point: X=5.67 abs., Y=0 abs., machining plane G17
The contour is programmed in a counter-clockwise direction.

Workpiece drawing of contour



Element	Softkey	Parameter	Remarks
1		All parameters, $\alpha_1=180$ degrees	Observe angles in help screen!
2		X=-43.972 inc, all parameters X=-137.257 abs $\alpha_1=125$ degrees	Definition of coordinates in X in "abs" and in "inc" Observe angles in help screen!
3		X=43.972 inc $\alpha_1=55$ degrees	Definition of coordinates in X in "inc" Observe angles in help screen!
4		X=5.67 abs	
5		CW direction of rotation, R=72, X=5.67 abs., Y=0 abs., Make a dialog selection	

6.5 Free contour programming



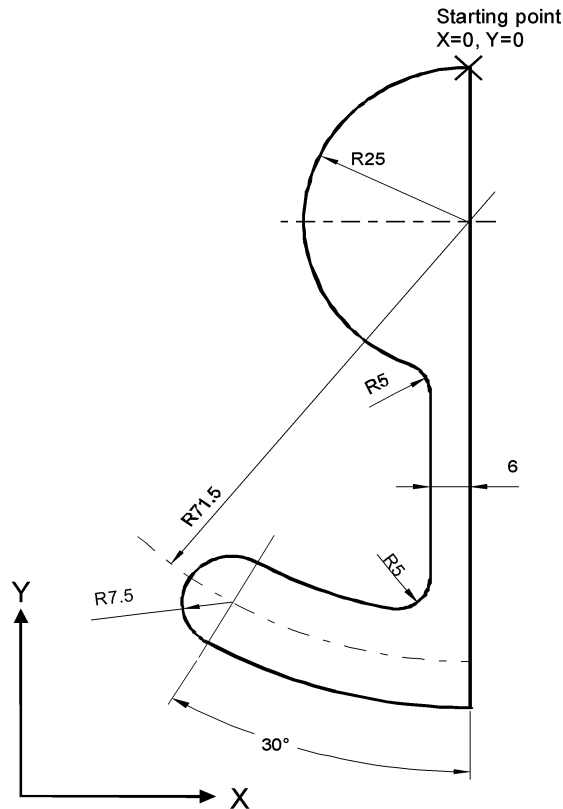
Example 2

Starting point: X=0 abs., Y=0 abs., machining plane G17

The contour is programmed in the clockwise direction with dialog selection.

For this contour it is advisable to display all parameters via the softkey "All parameters".

Workpiece drawing of contour

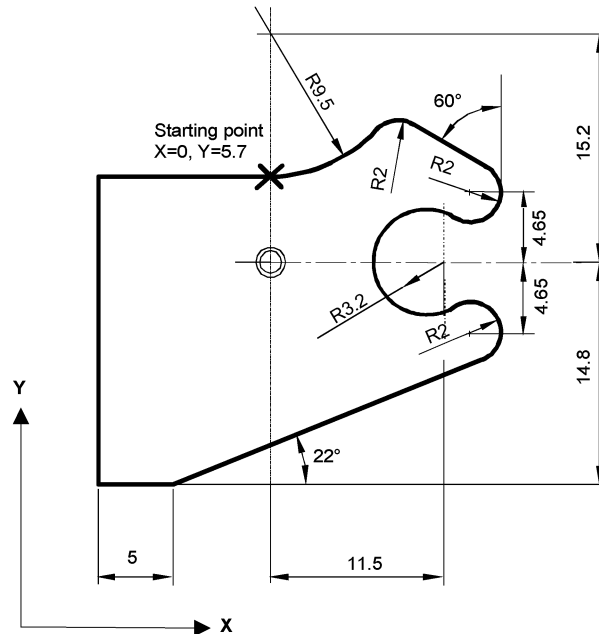


Element	Softkey	Parameter	Remarks
1	▲▼	Y=-104 abs.	
2	↻	Direction of rotation right, R=79, I=0 abs., Make dialog selection, all parameters, $\beta_2=30$ degrees	
3	↻	CW direction of rotation, tangent to preced. R=7.5, all parameters, $\beta_2=180$ degrees	
4	↻	Direction of rotation left, R=64, X=-6 abs., I=0 abs., Make dialog selection, make dialog selection Transition to following element: R=5	
5	▲▼	all parameters, $\alpha_1=90$ degrees, Transition to following element: R=5	Observe angles in help screen!
6	↻	Direction of rotation right, R=25, X=0 abs., Y=0 abs. I=0 abs Make dialog selection, make dialog selection.	

**Example 3**

Starting point: X=0 abs., Y=5.7 abs., machining plane G17
The contour is programmed in a clockwise direction.

Workpiece drawing of contour



Element	Softkey	Parameter	Remarks
1		Direction of rotation left, R=9.5, I=0 abs., make dialog selection, transition to following element: R=2	
2		$\alpha 1 = -30$ degrees	Observe angles in help screen!
3		CW direction of rotation, tangent to preced. R=2, J=4.65 abs.	
4		CCW direction of rotation, tangent to preced. R=3.2, I=11.5 abs., J=0 abs., make dialog selection, make dialog selection	
5		CW direction of rotation, tangent to preced. R=2, J=-4.65 abs., make dialog selection	
6		Tangent to preceding element $\alpha 1 = -158$ degrees, Y=-14.8 abs., $\alpha 2 = 0$ degrees	Observe angles in help screen!
7		All parameters, L=5, make dialog selection	
8		Y=5.7 abs.	
9		X=0 abs.	

6.5.11 Cycle support



References

Settings

Editor
settings

Set. con-
tour prog.

Additional tools in the form of prepared cycles requiring parameterization only are also available for the following technologies:

- boring
- milling
- turning.

Programming Guide, Cycles

Settings

Define the following values in the window "Editor Settings":

- Horizontal scrolling ON/OFF
- Display hidden lines ON/OFF
- Time interval for Automatic Save

When you set Automatic Save, you can also set the time intervals at which text must be saved automatically (applicable only to files on hard disk). If a value $\neq 0$ is entered, the softkey "Save file" is not displayed. If the value 0 is entered, automatic save is not performed.

- Automatic numbering ON/OFF
A new block number is automatically inserted on every new line. If you wish to assign new block numbers to an existing program, use the "Renumber" function.
- Number of first block
- Incrementation of block numbers (e.g. 1, 2, 10)

The following settings are possible for contour programming:

- Last line
Each time you complete a program step in the contour programming you can insert a text in the last line (e.g. "End of contour").

Additional notes

- The coordinate system and the technology used are set in the machine data; see /IAM/ HMI Advanced Installation and Start-Up Guide.
- Edited programs are automatically enabled after saving.

6.6 Program simulation



Function

Simulation of **drilling/milling and complete turning operations**

Simulation selection

The graphical simulation is implemented as a self-contained process. In addition to dialog programming selection under "Display mode", simulation in operating area "Program" can be selected directly from the program overview, after selection of a parts program, or from the ASCII editor, after a parts program has been opened.

Start-up

Special measures (see Simulation supplement in /IAD/ Installation and Start-Up Guide 840D, Section MMC, Section about simulation) are available for optimizing the start-up phase and basic behavior of the graphic processing simulation at various points. Various operational modes also allow the user to modify the selection and response time and the memory behavior of the simulation with the aid of a settings screen form.

Technology versions

- Drilling/milling
- Complete turning operations
- Workpiece-specific assignment through local "dpwp.ini" file

The model used is based on a 3D Cartesian model. It only controls the paraxial orientation of the tools.

Superimposing principle

The simulation results of several parts programs in succession (e.g. for multilateral machining during milling, internal/external machining during turning, multi-slide machining, etc.) can be superimposed on an overall display of the same blank (see machining list in Channel/ Spindle softkey). The finished part results from the sequential interaction of all the simulated parts programs. **The direct simulation of several parts programs (simultaneously) is not possible.** The scope of the parts program currently selected (displayed in the header of the simulation window) is shown in the status line at the bottom of the simulation window (channel, spindle, active tool, sequence pointer).

ASCII editor interface

After deliberate interruptions (simulation STOP or Single-block in the main simulation menu) or on alarms, you can use the "Correct program" softkey to open the program in the editor at the point of interruption. For interruptions within protected subprograms (e.g. cycles), the program pointer is positioned at the line with the corresponding subprogram call.

If changes are made in the editor, simulation is returned to the last active section (intermediate model if available).

Additional notes

1. The integrity of the simulation data (programs, subroutines, tool data, etc.) is assured by **systematic time-stamp evaluation**.
2. **NC language labels** (e.g. **WAITE2: WAITM...** ...), programmed to identify particular points in the program, **can be inserted in the simulation graphic as path labels**. You can simultaneously initiate generation of intermediate models at these points in the program (see "Managing Settings\Display and colors...\Path label management").
3. Basic information about simulation can also be found in **References: /FB/, D2 Interactive Programming**.

6.6.1 Simulation user interface

Basic horizontal menu

Match data
or

Time evaluation

Correct program

Channel/spindle

Menu tree

... after a simulation start-up or alarm status

With the correct password (can be set in ..\MMC2 or USER\dpsim.ini USER=n), you can match the active simulation data (tool data, machine data, cycles) with the corresponding "NC active data".

⇒ See menu "Data comparison"

Tabular evaluation at freely definable sections of the machining times calculated and estimated for the current simulation session (see Chapter "Setting down times")

Activate the ASCII editor from the current simulation interruption status (cursor position synchronized with graphic, user interface: standard ASCII editor subset)

⇒ Return to simulation with "Close editor"

Program-specific channel and spindle assignments (activate processing list in combination with the overlay principle)

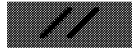
⇒ See menu "Channel/spindle"

Vertical main menu

or



Simulation START or simulation STOP
(program-by-program in conjunction with the machining list)



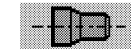
RESET simulation
(program-by-program in conjunction with the machining list)



SINGLE BLOCK simulation mode on/off
(status display in header SBL1 or SBL2)

Milling

or

**Technology-specific workpiece views**

Milling: Plan view

Turning: External view, frontal



or



Milling: Default plan view
and front view
(free selection under "Details...")

Turning: Full section, frontal



or



Milling: 3D view

Turning: Default half-section, frontal
or wire-frame model
front and wire-frame model
(free selection under
"Details...")



Select status-dependent (active view, alarm status) detail menus



or



- Select user and vendor-specific setting menus
(.. in RESET or STOP mode)

..or

(.. in RUN state)

- Show current simulation override bar
(% setting options:
 - Increment of 10 with "+" or "-" key
 - Increment of 50 with "Cursor right" or "Cursor left"
 - Max/Min value with "Cursor up" or "Cursor down".
 - Standard value 100 with "toggle" key)

**Notes**

1. On simulation START and program change, any 3D view currently displayed is automatically deselected and replaced by a technology-specific default view.

2. If you repeat machining simulation START after end of program M2/M30 of the last program to be simulated, the simulation channels are usually RESET and the simulation graphic reset if no machining list has been activated with softkey "Channel/Spindle".
3. If the machining list under "Channel/spindle" is activated, the overlay principle takes effect for the listed programs, where global reset in combination with a simulation START is only activated after a query on the last M2/M30.
4. New selection or reselection of one of the listed workpiece views is implicitly linked to automatic screen size adjustment.
5. During turning, drills and milling tools in the G19 plane can only be displayed as polymarkers (cross symbol) in the side views. Only the tool center point path without the tool body is displayed. The tool track is visible on the front face view (G17) or on peripheral surface processing (G19).

Menu "Details...." standard horizontal

Milling		Turning
Plan view	or	External view

Front view	or	Half-section	ditto
------------	----	--------------	-------

Side view	or	Full section	ditto
-----------	----	--------------	-------

Wire model	or	Wire model	ditto
------------	----	------------	-------


End face	ditto
----------	-------

Peripheral surface	ditto
--------------------	-------

... assuming 2 window views with no alarm status:

- ⇒ "Top view and front view" as default for milling
- ⇒ "Half section and wire-frame model" as default for turning.

With 2 window views only:

Free selection of basic view type in the window  activated with (milling and turning)

Menu "Details...." standard vertical

... assuming one of the active views with no alarm status:

- ⇒ "Top view" or "top view and front view" for milling
- ⇒ "Wire-frame model" (3D without tool data) for milling
- ⇒ All views for turning

Tool paths on/off

or

Tool paths on/off

Activate/deactivate representation of the tool center point path. Deactivation also deletes any tool paths from the current model that have already been stored (default setting: tool paths on).



With 2 window views only: Select active window (also possible with "TAB" or "END" key)

View from front..


Conditionally with 2 window views: "from front.." depending on type of view active (horizontal selection) same meaning as "from above.." or "from left.."

View from rear...

Conditionally with 2 window views: "from rear.." depending on type of view active (horizontal selection) same meaning as "from below.." or "from right.."

Automatic screen size

Automatic screen size adjustment,

applies to the window activated with  in two-window views (also possible with "ENTER" or "INPUT" key)

Zoom...

Show window pane border in active window (size can be altered with "+" - and "-" keys, position can be altered with cursor keys.)

<<

Return to main simulation menu

Menu "Details..." 3D**vertical**

. . . assuming from the following active view, with no alarm status:

- ⇒ "3D view" (3D with tool data) for milling
- ⇒ Not effective for turning

Standard 3D view, orientation top/front



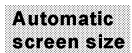
3D view, orientation top/left
(Standard rotated clockwise through 90°)



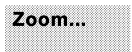
3D view, orientation top/right
(Standard rotated counterclockwise through 90°)



Standard 3D view, orientation bottom/front
(Standard flipped upwards)



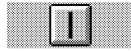
Automatic resizing
(also possible with "ENTER" or "INPUT" key)



Show window pane border in active window
(size can be altered with "+"- and "-" keys, position with "cursor" keys)



Return to main simulation menu

Menu "Details..." Alarms vertical

... assuming an alarm status during simulation, independent of the view currently active

Reset POWER ON alarms of simulation. The simulation is terminated and loaded again.



Reset RESET alarms of simulation. The simulation interpreter is reset. The simulation can be started again.



Reset CANCEL alarms of simulation. The simulation can be continued.

Close simulation

The simulation process is terminated. A load operation is necessary before the simulation is selected again.

Details view...

Display the menu bar "Details..." (standard or 3D) for the active workpiece view from alarm status. The simulation alarms remain active.



Return to main simulation menu

Additional notes

1. Simulation alarms are only messages of the simulation interpreter and have no direct association with the current NCK machining status on the machine tool.
2. If more than one simulation alarm is pending, you can show or hide the complete alarm list with the "toggle" key. You can select an alarm with the cursor keys.
3. If you press the information key "I" the online help with an explanation of the selected alarm is displayed.
4. Program sequences which cannot be interpreted in simulation contexts only and only then trigger an alarm (e.g. in user cycles because the relevant PLC data and signals are not available in the simulation interpreter), **must be jumped conditionally** in the corresponding NC program **with evaluation of system variable \$P_SIM during the simulation** (...IF \$P_SIM GOTOF label). The components relevant to simulation (e.g. tool change position and M switching functions for tool change in tool change cycles, etc. cannot be skipped, they must be executed.
5. The simulation does not evaluate events set by MD 20108, which lead to program activation.

Menu "Settings..."**vertical**

... only accessible from the RESET or STOP state (e.g. single block mode) of simulation

⇒ Settable parameters: See Section "Simulation settings"

Load standard

Load vendor-specific default settings

(Source: DHDP.DIR\SIM.DIR\simini_m.com for milling version
DHDP.DIR\SIM.DIR\simini_t.com for turning version)

Options on/off

or

Options on/off

Display/hide optional setting parameters in the current window
(Initial setting: options off)

Change standard

With the correct password (can be set in ..\MMC2 or USER\dpsim.ini SETUP=n) it is possible to modify the manufacturer-specific default setting values

Downtime settings

With the correct password (can be set in ..\MMC2 or USER\dpsim.ini USER=n) it is possible to set the required time recording for downtimes and define estimated downtimes for the NC functions T, S, M and H selectively.

⇒ Settable parameters: See Section "Setting downtimes"

Display and colors

With the correct password (can be set in ..\MMC2 or USER\dpsim.ini USER=n) it is possible to modify the predefined display and color properties active in simulation.

⇒ Settable parameters: See Section "Display and colors"

Abort

Return to main simulation menu. The old settings before the setting screen was selected are retained.

OK

Return to main simulation menu. The modified settings are saved and activated immediately (Destination: Workpiece or program-specific "dpwp.ini" or "for "Change standard" ..\DHDP.DIR\SIM.DIR\simini_m.com or simini_t.com").



Menu "Data comparison" vertical

Loaded
data

Match setup
data

Match
tools

Additional notes

The changes to manufacturer-specific standard setting values (..in files "simini_m.com" or "simini_t.com") and in files "dpmwp.ini" or "dptwp.ini" are incorporated in..\USER\ directory as difference parameters (delta values for "dppwp.ini" templates in ..MMC2\ directory) and are therefore included in all **new workpieces**.

... if the correct password has been entered (operator), accessible from the RESET or STOP state of simulation via the horizontal softkey bar.

The loading list of the simulation is displayed. Using vertical softkeys you can now choose:

- User programs
- User cycles
- Standard cycles
- Manufacturer cycles
- Basic data

The data you wish to select are loaded into the simulation environment from the NC. These data are now available to simulated programs as they would be to the NC during the program sequence.

Program	CHAN1	MDA	\SYF.DIR
Channel reset			OSTORE1.SYF
Program aborted			

Load setup data into simulation		Channel
Data select		1
<input type="checkbox"/> R parameter (RPA)	<input checked="" type="checkbox"/> Setting data (SEA)	
<input type="checkbox"/> User data (GUD)	<input checked="" type="checkbox"/> with global	
<input type="checkbox"/> with global	<input checked="" type="checkbox"/> with axis-specific	
<input type="checkbox"/> Zero offsets (UFR)	<input type="checkbox"/> Protection zones (PRO)	
<input type="checkbox"/> with global	<input type="checkbox"/> with global	
	<input type="checkbox"/> Sag/angularity(CEC)	
Data loading		
<input type="radio"/> for current channel		Abort
<input checked="" type="radio"/> for all channels		OK

CH_RPA.INI

After modifying NC data, new data must be adapted if necessary.

On the other hand, the response of the NC to modified data can be simulated in advance by modifying the data within the simulation environment.

Assuming the presence of an NCK component, an automatic alignment of the NCK and simulation tool data is performed.

(Source: ..\NC-active data\TO_INI.INI Destin.: H\DP.DIR\SIM.DIR\..)

Match
mach. data

Assuming the presence of an NCK component, an automatic alignment of the NCK and simulation initialization data is performed.
(Source: ..\NC-active data\INITIAL.INI Destination: DHDP.DIR\SIM.DIR\...)

Match
cycles

The cycles which have already been loaded into the simulation are replaced by cycles with more recent time stamps.
(Source: NCK or MMC ..\CST.DIR and ..\CUS.DIR)

<<

Return to main simulation menu.

Additional notes

1. If a data alignment has not yet taken place (and the NCK component is installed), a request is automatically issued in the form of a message when the simulation is initialized. The user is automatically informed of modifications to tool data.
2. The working cycles are loaded from the parts program **once on the first call** and remain active for all subsequent simulation sessions.
3. With the softkey "Compare cycles" you can load updated cycles with a new timestamp in the simulation. Cycles with access protection are always reloaded, irrespective of the timestamp. A realignment is only necessary after changes have been made to the cycles.

**Menu "Channel/spindle"
vertical**

... from simulation RESET or STOP status only can be called up via the horizontal softkey bar

The form shows the **current machining list** of selected parts programs in the current simulation session. The following parameters control the simulation sequence:

- ⇒ **Sequence:** Simulation sequence
- ⇒ **Program name:** Program identifier
- ⇒ **Channel:** Processing channel in the SIMNCK interpreter
- ⇒ **Spindle:** Location where named program takes effect:
 - ⇒ Milling: Currently not used
 - ⇒ Turning: On the main spindle, on the counterspindle, can be altered using NC keywords
- ⇒ **Skip:** The named program is ignored in the current session

**Setting
channels**

Calls manufacturer-specific channel settings (if password in ..MMC2 and USER\dpsim.ini USER=n has been set):

- Milling: Constant machine arrangement
- Turning: Machine arrangement in front or behind turning center.

**Setting
spindles**

Calls manufacturer-specific spindle settings (if password in ..MMC2 and USER\dpsim.ini USER=n has been set):

- Milling: Currently not used
- Turning: Defines longitudinal offset for main spindle and counterspindle, longitudinal mirroring on/off, NC keyword definitions for spindle switchover.

Copy

Copy selected line of machining list into buffer.

Paste

Paste the line which has been copied/cut out of the machining list from the clipboard at the position of the selected line (the selected line is pushed back one line).

Delete

Delete the selected line from the machining list

Abort

Return to main simulation menu: Modifications to the current machining list are not saved

OK

Return to main simulation menu: Modifications to the current machining list are saved and activated with the information displayed

Additional notes

1. **To activate the machining list after a workpiece selection, it must be selected explicitly at least once with the softkey "Channel/spindle".** The list is displayed implicitly every time a program is subsequently selected from the same workpiece directory and can be added to as required. It only makes sense to define a simulation sequence using the machining list "Channel/spindle" provided if the overlay principle for part machining is used (.i.e. when the simulation result of several parts programs in succession is to be viewed at a single blank). You must then pay special attention to the messages along the bottom of the screen form. For example, lines for which an intermediate model has already been generated contain special status displays. By selecting such a line in the machining list you can place the simulation directly on the associated intermediate model again.
2. If individual programs are simulated, the machining list default produces the correct simulation sequence even if menu item "Channel/spindle" is not selected (and therefore without applying the overlay principle).
3. With the correct password you can define manufacturer-specific default setting values with "Change default" under menu items "Channel settings" and "Spindle settings", which then apply **globally for all new workpieces.** (applicable password for "Change default" can be set in ..MMC2 or USER\dpsim.ini SETUP=n).

6.6.2 Simulation settings

Basic setting parameters are displayed immediately when the settings screen is called, without further intervention.

Optional setting parameters can also be displayed with softkey "Options on/off", if required.

Basic "Settings..."

- **Blank**
 - Milling: without model, cuboid, cylinder
 - Turning: without model, cylinder
 - Display range for "Without model"
 - Blank dimensions cube for "Cube"
 - Blank dimensions cylinder for "Cylinder"
 - ⇒ Turning: Additional active winding diameter for peripheral representation (default: external diameter)
- **Active view**
 - Milling: X-Y, Z-X, Y-Z (for "Cube" and "Without model" only)
 - Turning: predefined, always Z-X
- **Program control**
 - Take skip blocks and/or programmed STOP into account
 - Single block mode: STOP after each machine function (SBL1) or after each block (SBL2)
 - Display all blocks or only traversing blocks
- **Tool data (source)**
 - NC active data (provided NCK component is available)
 - MMC data (local TOA data, global SPF file, data from graphic DP tool catalog, tool data match from DH\DP.DIR\SIM.DIR\TO_INI.INI)
 - Default tool (for milling/drilling only, with tool diameter from optional "Settings..."/default values")
 - Without tool data (broken-line graphics with offset D0)

Additional notes

1. In order to reduce the input effort when the blank type is repeatedly changed, the blank dimensions and the display area are compared internally when the settings are saved.
2. **A suitable image of the NC-active data INITIAL.INI and TO_INI.INI under DH\DP.DIR\SIM.DIR is required for simulation with the option "Tool monitoring"**. Default settings for the tool management are taken from the supplementary tool data `..\mmc2\dp\sim\to_addon.ini`. Tools which are not loaded in the active magazine image (...from TO_INI.INI) can therefore be called during simulation if necessary.
3. In simulation "without tool data", using standard cycles results in the representation of the final contour derived from the available cycle parameters.
4. Simulation "without model" and/or "without tool data" both limits the required graphic memory and increases the simulation speed.

Optional "settings..."

- **Depths for color classification**
 - Depth range across which the available VGA system colors for displaying depth information are distributed (default range = blank thickness. The rounding inaccuracy in defining depth of color is 10^{-3} units)
- **Default values**
 - Tool diameter: Tool diameter used for milling simulation with default tool (end mill/drill) (only if option tool management is not active!).
 - IPO mm or inch: Approximation precision of simulation interpolator in mm or inches depends on actual measuring system.
 - F override %: Default setting of simulation feedrate

- **Display options**

- Actual position: Display hide actual value of simulated channel axes (Note: TRANS, ROT, SCALE and MIRROR are not taken into account in the actual value display).
- NC block: Activate/deactivate display of current NC block
- Machining time: Activate/deactivate display of the calculated machining time in the header of the basic simulation window
(T = calculated machining time (from the programmed feedrates)
 Σ = machining time + sum of all estimated down times).

- **Simulation mode**

- Always reload tools
 - * In position "on" (default setting), all the necessary tool data are reloaded on each program change.
 - * In position "off", tool data are only reloaded if required, if the timestamp is altered (with automatic query). Otherwise, the existing tool environment is not changed.
- Save tool path
 - * In position "on" (default setting), all the tool paths in the simulation sequence are visualized and stored temporarily in the model for future manipulations (zoom, move etc.).
 - * In position "off", the resulting tool paths are visualized once and not stored temporarily in the model. When the display is subsequently manipulated in such a manner as to cause a screen refresh, the tool paths are lost.
- Block-by-block processing
 - * In position "on", discontinuous path processing, but largely accelerated (reduced number of IPO interpolation points, for example, only at block end points on straight lines).
 - * In position "off" (default setting), largely continuous path processing (constant distance between interpolation points depending on the IPO setting).

- Waiting for dwell times
 - * In position "on", program instructions with dwell times cause real dwell times in the simulation.
 - * In position "off" (default setting), the dwell is suppressed in the simulation and the dwell times are only taken into account in time calculations.

Additional notes

1. Changes to the blank dimensions in the infeed axis (min or max) are automatically traced in the depths for color separation (min or max).
2. High values for the interpolator approximation accuracy cause increased distortion of the geometry representation (e.g. in details and rounded sections), but also reduce the amount of graphics memory required and increase the speed of the simulation.
3. Recommended setting for group "simulation mode":
 - ... in production
 - Always reload tool "off" and store tool path "off"
 - Block by block preparation "on" and wait with dwell times "off"
 - .. in program mode (default setting)
 - Always reload tool "on" and store tool path "on"
 - Block by block preparation "off" and wait with dwell times "off"
 - ... for training
 - Always reload tool "on" and store tool path "on"
 - Block by block preparation "off" and wait with dwell times "on"
4. The following settings are recommended for demonstrations with endless program loops (accelerated execution of simulation with reduced demands on the graphic memory):
 - Always reload tool "off" and store toolpath "off"
 - Block by block preparation "off" and wait for dwell times "off"

and additionally:

- Blank "without model" (graphic memory not used!)
- Possibly "without tool data", if representation of the tool center path (broken-line graphics) is sufficient.

If the blank model is active ("cube", "cylinder"), reduced graphic memory capacity might result, which, depending on the complexity of the part and model resolution set, could exhaust the graphic memory.

6.6.3 Downtime settings

Time recording mode

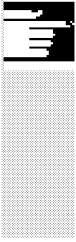
Setting time recording mode for downtimes:

- **OFF** (default setting).
Central time recording does not include fixed downtimes or the processing of information for tabular "Time evaluation".
- **For whole programs**
Central time recording includes the elements listed in "Inclusion on" in downtime consideration. The tabular "Time evaluation" (see horizontal softkeys) is processed once per program in each case with M30, for example.
- **For program sections with labels**
Effect as for "whole programs", only that processing of the tabular "time evaluation" is also performed when freely definable program labels occur or in combination with the path markers displayed in the simulation graphic ("..\Display and colors...\Path label management").

Inclusion

Enabling and setting of fixed downtimes for:

- | | |
|---------------|---|
| ON/OFF | <ul style="list-style-type: none"> • Tool change
– Definition of a fixed downtime for tool change commands |
| ON/OFF | <ul style="list-style-type: none"> • Spindles
Definition of one fixed downtime for main spindle instructions and one for secondary spindle instructions |
| ON/OFF | <ul style="list-style-type: none"> • M functions
– Multiple definitions of M identification & fixed downtime |
| ON/OFF | <ul style="list-style-type: none"> • H functions
– Multiple definitions of H identification & fixed downtime |



Additional notes

With the correct password it is possible to define other manufacturer-specific default setting values with "Change default" under menu item "Downtime settings". They apply **globally for all new workpieces**. (applicable password for "Change default" can be set in ..MMC2 or USER\dpsim.ini SETUP=n).

6.6.4 Display and colors

General attributes

Setting the general properties of the simulation graphics:

- **Rapid traverse broken line**,
alternatively, rapid traverse as an unbroken line, as used for feedrate
- **Scale in the window margin**,
alternatively, as a scale along the coordinate axes.
- **Path label management**,
Offers various possibilities for displaying **program labels**, which have been inserted freely to identify specific points in the NC program (observe label syntax), **as path labels** at the corresponding point in the simulation graphics and then storing the associated graphic model in the buffer.
Program labels can also be used to mark program sections which can then be included by the central time recording facility (see.. \Downtime settings\Time recording mode\In sections with labels).

General colors

- **VGA color palette**
In addition to the standard VGA colors, the color elements black with special background properties and transparent to hide graphic elements are available in the color palette.
- Color selection option for **blank, axis intersection, tool holder** and **tool edge**.

Color palettes for tool path

- Two freely definable color palettes are available for tool paths for differentiating between **feedrate and rapid traverse movements**.
- In each color palette it is possible to differentiate between different **basic tool types** (without tools, drilling tools, milling tools, turning tools, threading tools, special tools), to allow differentiation for path visualization.

Channel assignment of color palettes

- **One of the two** tool type specific color palettes can be freely assigned to each of the required simulation channels.

Depths for color classification

- In the case of milling/drilling, the available colors are assigned to the defined **cutting depth range**.
- The required cutting depth range (default value: blank thickness) is defined under
`..\Settings\Optional settings\Depths for color classification`

Additional notes

With the correct password you can define manufacturer-specific default setting values with "Change default" under menu item "Display and colors...", which then apply **globally for all new workpieces**. (applicable password for "Change default" can be set in `..\MMC2 or USER\dpsim.ini SETUP=n`).

With the function block search you can start simulation in specific program blocks.

6.6.5 Section by section simulation (SW 5.2)

A procedure in which first the individual sections of a program are optimized one after the other without collision consideration is the preferred method for testing parts programs graphically.

Section by section simulation allows the user to go to the individual section of the program via reference points (using block search).

The reference points are defined by path markers (program labels).

Precondition:

- In order to manage path markers (program labels, e.g. MARKER1), they must be programmed at the required position in the program.
- With the setting "Manage path markers" (settings -> Display/Colors -> Manage path markers) you can decide whether
 - the path markers are to be displayed in the graphic and/or
 - the associated intermediate model is to be saved.

Intermediate stages of the simulation model can be stored next to the path markers, allowing synchronized resumption of simulation without resetting the graphic that already exists.

You can skip any sections that are already optimized.

Block search:

In the menu "Block search" you can select the path marker you wish to jump to.

Additional notes

- The data loaded during the simulation session (user programs, cycles, standard cycles, basic data such as initial.ini, DEF files) can be displayed as a submenu in the menu "Data comparison".
- You can now also load the machining list in menu "Channel/spindle" directly from the current workpiece with the softkey "Program selection" or from a JOB list.

6.6.6 Simulation for orientable toolholder



Function

You can process parts programs for orientable toolholders using the simulation function. The following boundary conditions must be observed:

- The simulation distinguishes whether a toolholder Y has been activated for a tool x.
- Changes in the active toolholder are not detected. This is why the simulation uses the toolholder kinematic set initially for a tool y.
- Changes made after first activation (TCARR=x) are not considered.

- To use several toolholder kinematics for one tool y, create several identical tools with different toolholder settings.

6.7 Simulation with external network drive



Function

With the SINDNC software, you can link your control to external network drives or other computers and use this facility for program simulation. It is possible to access files on network drives from a parts program using the command EXTCALL.

- With EXTCALL, the drives in the network are also searched for subroutines (SPF only) if the program is called without a path. Subdirectories are not searched. The program will also be found if the search range is defined with the variable \$SC_EXT_PROG_PATH or if a path to a file in the network – even in a subdirectory – has been defined.
- Programs in the network drives (with the extension MPF and SPF) can be simulated.
 - If write access is set for a network drive, file DPWP.INI is created and the current directory is treated like a workpiece.
 - If write access has not been set, a DPWP.INI is created for each network drive in the TEMP directory of the HMI irrespective of the current directory. In this case, the simulation settings are lost when the directory on the drive is changed.

6.8 Program management

6.8.1 Overview

To allow you to handle data and programs flexibly, you can organize, store and display them according to different criteria.

The memory is divided into:

- NC memory (main memory and program memory) containing the active system and user programs as well as all parts programs for immediate execution and
- Hard disk

Programs can be exchanged with the program memory and hard disk. Parts programs may be stored either in the program memory of the NC or on the hard disk, but they are always executed from the program memory.

The "Load" and "Unload" functions are used to load or unload programs to or from the program memory.

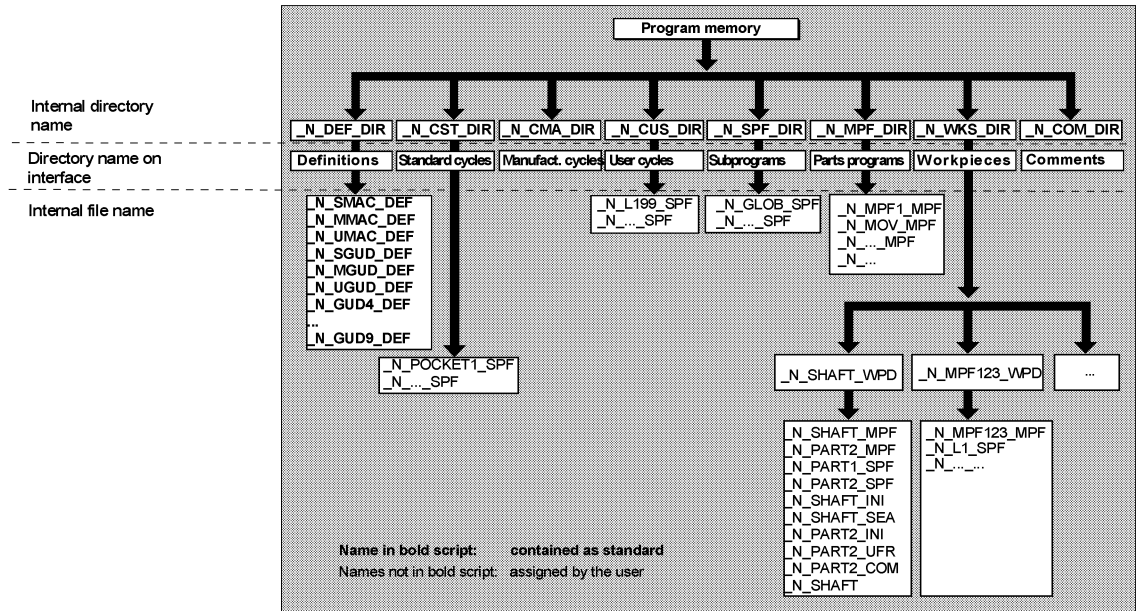
Program management

The programs and files are stored in different directories and can be administered in the Program and Services operating areas.

The details are shown in the following overview:

Name of directories:	Operating area:
• Subprograms	Program and Services
• Part programs	Program and Services
• Workpieces	Program and Services
• Definitions	Services
• Comments	Services
• Standard cycles	Program and Services
• Manufacturer cycles	Program and Services
• User cycles	Program and Services

The following diagram shows an example of directory contents:



6.8.2 NC file types and directories

File types

File types can be identified by the file name extension (e.g. .MPF).

name .MPF	Main program
name .SPF	Subprogram
name .TEA	Machine data
name .SEA	Setting data
name .TOA	Tool offsets
name .UFR	Zero offsets/frame
name .INI	Initialization file
name .COM	Comment
name .DEF	Definition of global user data and macros

Please note that the directories for comments \COM.DIR and definitions \DEF.DIR can only be accessed in the Services operating area using the "Manage data" function.



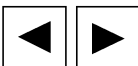
6.8.3 New workpiece/parts program



Workpieces

Parts
programsSub-
routinesUser
cycles

Clipboard



Selecting a workpiece/parts program

The following subsection describes how you can select workpieces and parts programs in a directory. A selected file can then be called and edited in the text editor.

Sequence of operations

Select workpiece/parts program:

- Workpieces
- Parts programs
- Subprograms
- User cycles
- Clipboard

Position the cursor in the directory on the desired file.

For each file, the file name, file type, length, date of creation/last change are displayed.

You can change the properties of the file display (see Section "Start-up", "Settings" menu)

Call a parts program:

Use the cursor to select a program in the program overview and press the "Input" key.

The text editor is displayed with the file you have selected.

You can now edit the parts program.

Open a workpiece:

The workpiece directory is opened and the programs it contains displayed on the screen.



Workpieces

New



...

Create workpiece directory

You can set up various types of files such as main programs, initialization files, tool offsets, etc. in the new workpiece directory.

Sequence of operations

The current overview of all workpiece directories appears on the screen.

Input window "New" is opened.

The cursor is located in the input field for the name of the new workpiece directory.

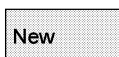
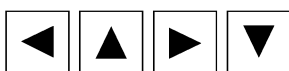
Enter the name of the new directory on the alphanumeric keyboard.

A new directory is set up in the workpiece overview.

You are asked straight away for the name of the first parts program and the Editor is opened.

You can create your own templates for job lists or standard parts programs/subprograms with the name of the workpiece in the workpiece directory.

For more detailed information, see Chapter "Job list" under "Creating the file "Workpiece.JOB" (e.g. SHAFT.JOB)".



Set up programs/data in a workpiece directory

This section explains how you can set up a new file for a parts program or workpiece.

Sequence of operations

The current overview of the workpiece directories stored on the NC appears.

Position the cursor on the required workpiece and open it.

You obtain an overview of the data and programs that have already been set up under the workpiece directory. If no data exist, an empty program overview is displayed.

A dialog box appears when you select softkey "New".

Enter the new file name.

You can also enter the corresponding file type using the "Insert key". Some of the possible file types are listed below:

File type	Meaning
.MPF	Main program (Main Program File)
.SPF	Subprogram (Sub Program File)
.TOA	Tool offset (Tool Offset Active)
.UFR	Work offset (User Frame)
.TEA	NC machine data (Testing Data Active)
.SEA	Addresses with assigned values (Setting Data Active)
.COM	Comment file
.INI	Initialization data
.GUD	Channel user data
.QEC	Quadrant error compensation
.CEC	Sag/angularity compensation
.TOP	Tool plan (for SINTDI)
.TCM	Tool plan, unformatted (for SINTDI)
.JOB	Job list
.RPA	R variable
.TMA	Magazine data
.PRO	Protection zones

Parts programs or Sub-programs

New



Number of workpieces

Creating parts programs in parts program/subprogram directory:

You can set up main programs and subprograms by opening directories "Parts programs" and "Subprograms".

Select softkey "New" to display a dialog window in which you can enter the names of the new main programs and subprograms. The matching file type is automatically assigned in this case.

You can manage up to 310 workpieces/programs/files in each directory.

6.8.4 Storing setup data



Workpieces



Save setup data

Function

With the softkey "Storing setup data", you can store all the active data belonging to a particular workpiece located in the RAM of the NC. The data are stored for each channel under the same name under a workpiece.

Additional notes

"Storing setup data" can be disabled by the manufacturer using the protection levels.

Sequence of operations

The current overview of all workpiece directories appears on the screen.

Place the cursor on the workpiece directory or workpiece in which you want to store the workpiece-specific data.

The input window "Store workpiece data" is opened when you select "Save setup data".

In this input window you can select the workpiece data that you wish to back up.

E.g. you can select the following data types:

- R variables (RPA)
- Work offset (UFR)
- Setting data (SEA)
- ...



Save

If a job list exists, it is used as the basis for the backup procedure.

With the softkey "Save", you store the workpiece data of the selected data types into the relevant workpiece directory.



Load
standard

If a job list exists in this workpiece directory, the data for all the channels involved in the workpiece are automatically backed up. For this, the main program selected at the beginning is searched for each channel in the job list. The data are then backed up according to the data types selected under its name.

With the softkey "Load standard" you can load the default settings for input window "Store workpiece data".

Note:

Default settings are always supplied by SIEMENS.



Set
standard

If you want to define your own defaults in the input window, press this softkey.

6.8.5 Selecting a program for execution



Function

Workpieces and parts programs must be selected for machining/execution before you press the NC Start key.



Sequence of operations

Select a program:

Use the cursor keys to select a program in the

program overview, e.g. parts programs, and

then press softkey "Selection".

The program name is displayed in the "Program name" window at the top right.

You can start the parts program by pressing "NC Start".



Parts
programs




Selection



Cycle Start

Workpieces



Selection



Selection

Find path for program call

Select a workpiece:

A workpiece directory can be selected for machining in the currently active channel.

Use the cursor keys to select the workpiece

in the workpiece overview and then

press softkey "Selection".

- If only one main program (MPF) exists in this directory, it is automatically selected for execution. If the directory contains several main programs, the main program with the same name as the directory is automatically selected for execution (e.g. if you select workpiece `SHAFT.WPD`, main program `SHAFT.MPF` is automatically selected).

The program name is output along with workpiece information in the "Program name" window at the top right.

You can start the parts program you have selected by pressing the "NC Start" key.

- If an INI file of the same name exists, it will be executed immediately you select the parts program (e.g. `SHAFT.INI`).
- Machine data 11280 `$MN_WPD_INI_MODE` controls which programs are executed in a workpiece directory.

When you select workpiece directory `SHAFT.WPD` you implicitly select program `SHAFT.MPF`.

If a `.JOB` file with the same name is stored in the workpiece directory, then it is executed immediately.

See also Chapter "Job list" and "Sequence of operations" "Executing job list".

If the call path for a subprogram (or an initialization file) is not explicitly specified in the parts program, a fixed search strategy is applied to find the called program.

Case 1: When a subprogram is called by

name **with specification of the file type** ("identifier" or "extension"), e.g. `SHAFT.MPF`,

the system searches through directories in the following order:

1. Current directory / name.type Workpiece/standard directory `MPF.DIR`

- | | |
|------------------------|---------------------|
| 2. /SPF.DIR / name.typ | Global subroutines |
| 3. /CUS.DIR / name.typ | User cycles |
| 4. /CMA.DIR / name.typ | Manufacturer cycles |
| 5. /CST.DIR / name.typ | Standard cycles |

Case 2: When a subprogram is called by name **without specifying the file type** ("identifier" or "extension"), e.g. SHAFT1,

the system searches through directories in the following order:

- | | |
|---------------------------------|---|
| 1. Current directory / name | Workpiece/standard
directory MPF.DIR |
| 2. Current directory / name.SPF | |
| 3. Current directory / name.MPF | |
| 4. /SPF.DIR / name.SPF | Subroutines |
| 5. /CUS.DIR / name.SPF | User cycles |
| 6. /CMA.DIR / name.SPF | Manufacturer cycles |
| 7. /CST.DIR / name.SPF | Standard cycles |

/PGA, Programming Guide, Advanced



Machine manufacturer

See machine manufacturer's specifications

Preconditions:

- a main program (MPF) in the workpiece directory is selected
- "NC Start" has been pressed

\$MN_WPD_INI_MODE=0:

The INI file with the same name as the selected workpiece is executed.

E.g. if SHAFT1.MPF is selected, SHAFT1.INI is executed with "NC Start".

(the response is the same as in previous versions)

\$MN_WPD_INI_MODE=1:

All files with the same name as the selected main program and extensions INI, SEA, GUD, RPA, UFR, PRO, TOA, TMA and CEC are executed in the specified order.

- The main programs stored in a workpiece directory can be selected and processed by several channels.

For further information, please see

/IAM/ Installation and Start-Up Guide HMI/MMC, IM4



6.8.6 Loading/Unloading a program



Load
HD->NC



Change
enable

Unload
NC->HD



Function

Programs can be stored in the NC memory ("Load") and then erased from it again ("Unload") after execution. This prevents the NC memory from being overloaded unnecessarily.

Sequence of operations

Position the cursor on the program to be loaded.

The highlighted program is loaded from the hard disk to the NC memory.

The selected program is deleted on the hard disk.

See also Chapter "Job list".

If enable is set "(X)", the program can be executed.

The highlighted program is unloaded from the NC memory to the hard disk.

The selected program is deleted in the NC memory.

Additional notes

Programs which have been loaded to the NC memory are automatically marked with an "(X)" (in the "Loaded" column) in the program overview.

If the file is located both in the HMI and in the NC, the identifier "X" is only removed when the files are no longer identical.

If the files have different time stamps or are of different lengths, the identifier is "!X!".

If you want to "load/unload" a workpiece directory and a job list with the name of the directory exists in the directory, then this job list is processed.

If a job list does not exist, all the files in that directory are loaded/unloaded (RAM of NC might overflow!).

6.8.7 Program management



Function

You can reorganize programs and files in the "program overview" using the "Manage programs" function:

New ...	Select a new workpiece/part program
Copy/paste	Copy directories and files
Delete	Delete workpieces and files
Rename	Rename file and file type
Alter enable	Define whether or not each workpiece/part program is enabled in the program overview.

Sequence of operations

Press the horizontal softkey "Manage programs" in the "program overview" window. Active functions are indicated by softkeys in which the text appears on a black background. Inactive softkeys are grayed out.

Manage programs...

Press the "New ..." softkey to select a new workpiece or part program. The "interactive programming" window is opened and the cursor is positioned in the input field for the name of the new directory. Enter the new name of the program. The file type is assigned automatically. The display also shows whether or not a template exists.

New ...

Default

You can choose between interactive programming and the text editor. The more frequent choice, file editing with "interactive programming", is the default setting. Alternatively, you can use the text editor to edit files.



Abort

OK

Confirm your input with the "Input" key.

This key cancels the entire operation.

Press the "OK" softkey to load the workpiece or part program you have just selected. You can then use the program management functions available in the horizontal softkey menu.

6.8.8 Copying/inserting



Function

This section tells you how to copy files from a source directory to an existing destination directory.

Except when overwriting workpieces, the files are copied from the NC program memory or from the hard disk to:

Hard disk	(HD)	or
NC memory	(NCK)	or
Clipboard		or
Disk (if entered as a network drive)		or
Network drives (Network1 to Network4, SW 6.3 and higher)		

All directories with the exception of compile cycles can be copied to the destinations specified above.

Copy/paste, SW 6.3 and higher

Several selected files or workpieces can be copied together in a single operation.

When copying workpieces:

All associated files are copied.

All files with the same name as the workpiece are automatically renamed with the new workpiece name.

Files from the workpiece directory can only be copied to a different directory.

All workpiece files with the extension .WPD can be copied from one drive to any other.



Specifying the file name and file type

Copying network drives (SW 6.3 and higher):

All data types recognized by the HMI are set according to their extension and data management scheme.

Files with unknown extensions are displayed in a dialog box in which the name and data type, consisting of up to 3 letters, can be changed.

If a file of the same name is already stored on the hard disk or in the NC memory, a dialog box prompts you to change the name and file type. If the source file and destination are identical, the existing file is not overwritten. The copy operation can either be canceled or an integer is added to the existing file name in ascending order.



Manage programs



Copy



Insert

OK



Sequence of operations

The softkey "Manage programs" must be pressed.

Position the cursor on the file that you want to copy and press the softkey "Copy".

The file is marked as the source for copying. An existing file is either overwritten or, if the name or file type was changed, saved with the new name.

Please note that several "Save as" dialogs give you the chance to change the file name and file type before the files are overwritten. The "Save as" dialog box appears if the vertical softkey "All without query" was not pressed and:

1. you have pressed "Insert" for the first time.
2. the file cannot be created in the current directory.
3. the file already exists.

Press the softkey "Insert", enter a new name and confirm with "OK".

When you insert a workpiece directory, you can also change the file type with the "Toggle key". The file types are matched automatically in the global parts program and in the global subprogram directory.



"Save as" query dialogs

All without
query

Skip file

Abort

OK

Additional notes

- Only files can be stored in a workpiece directory but not other workpiece directories.
- If the target specified is incorrect an error message is output.
- If a workpiece directory is copied, all the files that it contains are copied at the same time.
- If the files of a workpiece directory are copied to a new directory, all the files with the same directory name are renamed to the new workpiece directory name.
- If a job list with the name of the directory exists, all the instructions in that job list are renamed.

This function applies to operating area "Program" only.

When you copy under "Services", the names remain unchanged.

See also section "Renaming".

In the "Start-Up" operating area, press the MMC/System settings/ Query softkeys to specify that a dialog box is to appear before files are overwritten. Otherwise, the files are overwritten without prompting or a copy is created.

Vertical softkeys for "Save as"

Press the "All without query" softkey if all existing files in the current directory are to be created with new names without a "Save as" dialog. All files for which the original file type cannot be created are automatically converted to the specified data type.

Press the "Skip file" softkey if the copy operation is to be continued with the next file.

This key cancels the entire copy operation.

An existing file is either overwritten or, if the name or file type was changed, saved with the new name. The "OK" softkey is disabled if you have to enter a new name. The "Data type" input field accepts 0 to 3 letters.



Query dialogs

Copy file from hard disk:

- The file exists on hard disk and is overwritten when you select "OK" if the name/data type are to remain unchanged!
- The file exists on hard disk. When you select "OK" a copy is created if the name/file type are to remain unchanged!

Copy file from NC memory:

- The file exists on the NCK and is overwritten when you select "OK" if the name/data type are to remain unchanged!
- The file exists on the NCK. When you select "OK" a copy is created if the name/file type are to remain unchanged!

Copy a workpiece:

- The workpiece already exists. When you select "OK" a copy of the workpiece is created if a new name is not specified!

Copy a directory:

- The directory already exists. When you select "OK" the contents are overwritten if a new name is not specified!
- The directory already exists. When you select "OK" the contents are overwritten if the directory with a fixed data type cannot be changed.

Copy a file of data type main program (MPF):

- The file cannot be created at this location under its original "main program" data type!

6.8.9 Deleting



Manage
programs



Function

This section explains how you can delete workpieces or files.

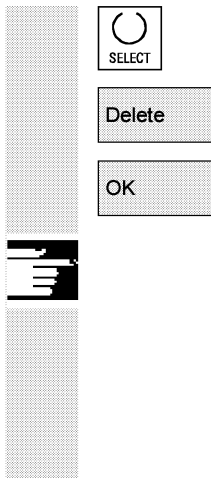
Sequence of operations

The softkey "Manage programs" must be pressed.

Position the cursor on the workpiece or the file you want to delete.

Delete several files:

If you wish to select several files, position the cursor on the first file, press the "Select" key and then position the cursor on the last file.



The files you have selected are highlighted.

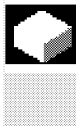
The prompt "Do you really want to delete the file?" appears.

Confirm your input.

Additional notes

- You can only delete programs that are not currently running.
- If you want to delete a workpiece directory, make sure that none of the programs it contains is currently selected.
- If a workpiece directory is deleted, all the files that it contains are deleted at the same time.

6.8.10 Renaming



Function

As regards files, you can alter their name as well as the associated file type.



Manage programs



Rename

Sequence of operations

The softkey "Manage programs" must be pressed.

Position the cursor on the file you want to rename.

The "Rename" dialog window opens.

Enter the new name.



When you rename a workpiece, you can also change the file type with the "Toggle key".

File types are matched automatically in the parts program and subprogram directories.

There are two ways of renaming files:

- Renaming the workpiece directory
- Renaming a directory in the workpiece directory

Renaming a workpiece directory:

When you rename a workpiece directory, all the workpiece files under that directory that have the same name as the directory are renamed. If a job list with the name of the directory exists, the instructions in that job list are also renamed. Comment lines remain unchanged.

Examples:

Workpiece directory A.WPD renamed to B.WPD:

All files with the name A.XXX are renamed to B.XXX, i.e. the extension is not altered.

If a job list called A.JOB exists, it is renamed to B.JOB.

If this job list contains instructions of file A.XXX located in this workpiece directory, then that file is also renamed to B.XXX.

If job list A.JOB contains an instruction

```
LOAD/WCS.DIR/A.WPD/A.MPF
```

it is renamed to

```
LOAD/WCS.DIR/B.WPD/B.MPF
```

However, if a job list contains the instruction

```
LOAD/MPF.DIR/A.MPF or  
LOAD/WCS.DIR/X.WPD/A.MPF
```

the files are not renamed.

Renaming a directory in the workpiece directory:

If you rename the files in the workpiece directory, all files with the same name but a different extension are renamed.

Exception: If a job list of the same name exists in the directory, then this one is not renamed.

6.8.11 Enabling



Change
enable



Function

The program overview indicates whether a workpiece or parts program is enabled.

If a program is enabled, it may be executed (e.g. because it has already been tested) by the control after you select softkey "Program selection" and the "NC Start" key.

If you set up a new program, it is automatically enabled.

Sequence of operations

To set the enable for a program or abort it, position the cursor on the desired workpiece or parts program in the program overview.

Select softkey "Change enable".

A cross indicating "Enable issued" appears behind the workpiece or parts program.

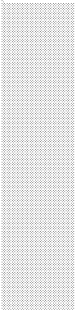
(X) Enable issued (program may be executed)

() Not enabled (program must not be executed)

Additional notes

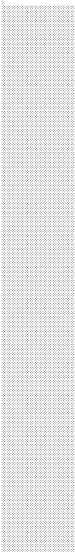
- The system checks whether a program may be executed when the program is called (after selection via operator input or from parts program). If an enable is required it must have been set previously. (See also Chapter 6 "Changing properties of file/directory/archive")

6.8.12 Log

**Function**

If you are working on the hard disk, the following data are included in the log:

- Name of program currently being executed (for "Processing from external source")
- Names of previously executed programs
- Prompts, e.g. "Do you really want to delete job?"
- Error list: Names of previously executed programs in which an error occurred.



Log

Yes

All

No

Stop

Sequence of operations

Select softkey "Log".

The "Job log for programs" window opens.

Depending on the status of the current program run, the following functions can be executed by means of the vertical softkeys (e.g. when prompt "Really delete?" is displayed in the "Query" window):

- The program currently being executed is deleted.
- All programs in the current job list are deleted.
- The program currently being executed is not deleted.
- The program currently being executed is aborted.

6.9 Accessing an external network drive/computer



Function

With the SINDNC software, you can link your control to external network drives or other computers. The following conditions apply:

- The SINDNC software is installed.
- The computer or drive which you want to link up to is accessible/enabled.
- A connection to the computer/drive has been established.
- The softkeys for selecting the drive/computer link have been configured by making entries in the file "MM.INI", see /IAM/ Installation and Start-Up Guide HMI/MMC, IM4



Sequence of operations

You can access network drives 1 to 3 from level 0 of the Program operating area using horizontal softkeys 4 to 6. With the "ETC" key, you can access network drive 4 from level 2 using horizontal softkey 7. The cycle directories are also stored at level 2 and are assigned to softkeys 4 to 6.

Program



Drive F:

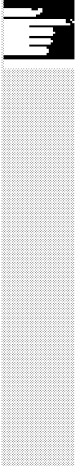
or

R4711

When you press a softkey, e.g. "Drive F:" or "R4711" the Explorer appears on the screen with the data of the external drive, e.g. "Drive F" or of computer "R4711".

You can perform the following operations (in addition to paging) via the vertical softkeys in the Program operating area:

- Copy/paste files (not directories):
 1. From network drives to the data management. A type conversion cannot be performed according to the destination directory.
 2. From the data management or a network drive to network drives. The files are set up on the network drive using DOS naming conventions. The file name used in the data management (source) is maintained.
- Deleting files (no directories) on the network drives
- Simulation
- Editing files (find/go to, mark block, change) if write access to the drive is enabled.
- Files on the network drives can be simulated. This applies to files with the identifier MPF or SPF.



Additional notes

- If the drive/computer is not connected or enabled, the message "No data available" is displayed.
- A root directory can only be selected as the destination for copying to a network drive if "." is displayed.
- In the Machine operating area, only files with an HMI-compatible name can be selected for "Exec. f. harddisk" (i.e. up to 27 characters, no special characters, no blanks).
- In operating area Program, the functions Copy, Insert, and Delete can only be applied to files **without** blanks in their name.
- The file name is displayed as it would be in the Windows Explorer with a long file name, but only up to 25 characters are displayed.



"Services" Operating Area

7.1	Function.....	7-352
7.2	Directory structure.....	7-352
7.2.1	NC active data.....	7-352
7.2.2	Hard disk.....	7-353
7.2.3	Directories.....	7-356
7.2.4	Data selection.....	7-357
7.3	Formats for saving and importing data.....	7-360
7.3.1	Punched tape format.....	7-361
7.3.2	PC format.....	7-365
7.4	RS-232 interface parameters.....	7-366
7.4.1	Interface parameters.....	7-369
7.5	Operator interface.....	7-371
7.5.1	Services basic display.....	7-371
7.5.2	Setting the RS-232 interface.....	7-374
7.5.3	Reading in data.....	7-379
7.5.4	Reading out data.....	7-381
7.5.5	Log.....	7-382
7.5.6	Importing/exporting ISO programs.....	7-384
7.6	Manage data.....	7-389
7.6.1	Standardized program and data management (SW 6.3 and higher).....	7-389
7.6.2	Creating a new file/directory.....	7-390
7.6.3	Loading/unloading.....	7-390
7.6.4	Copying/inserting.....	7-391
7.6.5	Deleting.....	7-393
7.6.6	Changing the properties of a file/drive/archive.....	7-394
7.6.7	Defining and activating user data (GUD).....	7-396
7.7	Start-up functions.....	7-398
7.7.1	Series start-up.....	7-398
7.7.2	Restoring the original state via NC card.....	7-401
7.7.3	Software update.....	7-402

7.1 Function



The "Services" operating area provides the following functions:

- Read data in/out
- Manage data
- Series start-up

7.2 Directory structure



All files are organized in a directory structure.

Files in the NC memory and on the hard disk, sorted in "file trees" in directories.

7.2.1 NC active data



The NC memory contains data (e.g. R parameters, tool offsets, machine data) that are not stored there in file format.

The directory "NC Active data" is provided in the file manager to allow the user to access these data and store them on the hard disk in file format.

This directory contains an overview of data that can be copied from the NC memory.

If the operator wishes to save data in files on the hard disk, they can set up the same directory structure under a directory of any name for "NC data" (.MDN).

Via "Copy/paste" any active data can be fetched from the NC and stored in files on the hard disk.



NC-active data cannot be unloaded.

7.2.2 Hard disk

General information

In addition to the NC working memory, SINUMERIK controls with HMI are provided with a hard disk. It is thus possible to store all data or programs which are not required in the NC on the hard disk.

All data are displayed in a single file tree on the user interface.

In the "Services" operating area, all files on the hard disk and in the NC memory can be:

- transferred to and from diskette via two RS-232 interfaces,
- managed (new, load, unload, copy, delete, change properties),
- saved for a series start-up (NC, PLC and HMI data) and
- loaded to the NC memory (programs and files).

Interfaces

The parameters of the two RS-232 interfaces are also set in the "Services" operating area. The device-specific interface parameters and the communication protocol can be defined and stored separately for each RS-232 interface.

Copying to disk

When you copy to diskette the full file names are stored on the diskette.

Additional notes

The end of block character is displayed not as "LF" but as "¶".

System queries

The system behavior when copying/creating data (e.g. overwrite existing files or query first) can be configured for all operating areas.

"Save as" query dialogs

In the "Start-Up" operating area, press the MMC/System settings/Query softkeys to specify that a dialog box is to appear before files are overwritten. Otherwise, the files are overwritten without prompting or a copy is created.



Query dialog

Copy file from hard disk:

- The file exists on hard disk and is overwritten when you select "OK" if the name/data type are to remain unchanged!
- The file exists on hard disk. When you select "OK" a copy is created if the name/file type are to remain unchanged!

Copy file from NC memory:

- The file exists on the NCK and is overwritten when you select "OK" if the name/data type are to remain unchanged!
- The file exists on the NCK. When you select "OK" a copy is created if the name/file type are to remain unchanged!

Copy a workpiece:

- The workpiece already exists. When you select "OK" a copy of the workpiece is created if a new name is not specified!

Copy a directory:

- The directory already exists. When you select "OK" the contents are overwritten if a new name is not specified!
- The directory already exists. When you select "OK" the contents are overwritten if the directory with a fixed data type cannot be changed.

Copy a file of data type main program (MPF):

- The file cannot be created at this location under its original "main program" data type!

Vertical softkeys for "Save as"

All without query

Press the "All without query" softkey if all existing files in the current directory are to be created with new names without a "Save as" dialog. All files for which the original file type can be created are automatically converted to the specified data type.

Skip file

Press the "Skip file" softkey if the copy operation is to be continued with the next file.

Abort

This key cancels the entire copy operation.

OK

An existing file is either overwritten or, if the name or file type was changed, saved with the new name. The "OK" softkey is disabled if you have to enter a new name.



Display

The file tree display can be modified by the user:

- Display of file properties
- Number of displayed directories



Data management

Data management limits (DM):

A maximum of 100,000 files can be stored in the following data management directories:

- Workpieces
- Parts programs
- Subprograms
- User cycles
- Standard cycles
- Manufacturer cycles

The number of files in each directory (each *.WPD workpiece directory for workpieces) is limited to 1000 files.

The total of 100,000 does not apply to the other DM directories, however each directory is also limited to a maximum of 1000 files, e.g. a maximum of 1000 archives in the archive directory. Network drives are also limited to a maximum of 1000 files in each directory.

The options that are available depend on the file size and the available memory space. A large number of files slows down the display building for directory displays.

7.2.3 Directories

The following directories contain special files:

1. **Clipboard:**

Files and directories of any type may be created/stored in the clipboard. They can be used, among other things, to copy and rename files and directories.

The clipboard is a directory on the hard disk, where files, which cannot be included in the copy target directory, are stored. This may be the case, for example, if their file type is unknown, or is not permitted in the copy target directory.

2. **Archive:**

If you wish to save several files, you can store them in an archive file (.ARC). Archive files are generated in a special format:

- a. Punched tape format
- b. PC format

(see also Sections "Punched tape" and "PC format")

The source path of files stored in an archive file is also saved in the archive, allowing the packed file to be transferred back to the same directory from which it was copied when the archive file is unpacked again.

Series start-up archives are also stored in this directory.

7.2.4 Data selection

Under softkey "File selection", you can choose the directories that you wish to have displayed in the "Services" area. The directories can be selected for two different access levels:

- User
- Maintenance

Data selection for display: User
User cycles
Display machine data
Archive
Machining sequence
Data management
Definitions
Diagnosis
Interactive programming
Manufacturer cycles
MSD data
Start-up
Comments
MBDDE alarm texts
NC-active data
NC data backup
OEM data
Standard cycles
System
Parts programs
Templates
Subprograms
RS-232 interface
FDD data
Workpieces
Tool management
Clipboard

**Keywords/directories
in the hierarchical file
structure**

The different files are made available for transfer under the following directories:

- Data (general)
 - Option data
 - Machine data (all, NC MD, channel MD, axis MD)
 - Setting data
 - Tool offsets
 - Work offsets
 - Global user data
 - R variables
- Start-up data
 - NCK data
 - PLC data
- Compensation data
 - Leadscrew/encoder error
 - Quadrant error
 - Beam sag/angularity
- Display machine data
- Workpieces
- Part programs
- Subprograms
- User cycles
- Standard cycles
- Comment data
- Definitions
- Feed drives
- Main spindle drives
- OEM data
- System data (NC)
- Logbook
- Communications error log

If your control includes additional directories, these can be found in the file tree.



Services

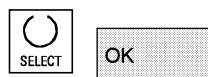
Data selection



User

Maintenance

Default settings



Abort



Data selection

Sequence of operations

The "Services" operating area is selected.

Press softkey "Data selection".

The "Data selection for display" window is opened.

The vertical softkey bar changes.

You can page through the window using the "Page" keys.

You can toggle between the settings

- User
- Maintenance
- Default settings

The softkey "Standard values" assigns default values to the settings "User" or "Maintenance". These are suitable defaults that are supplied in the software.

Select, for example, the "User" setting. Position the cursor on an additional directory.

Mark the directory of your choice and press softkey "OK".

A reset command by means of softkey "Abort" for the selected access level resets the data selection.

Additional notes

The file tree displays files that the operator may view on the basis of his or her access rights.

7.3 Formats for saving and importing data

Path name

The path name is automatically entered when files are saved (archived).

The path is named in the first line of a file:

```
;$PATH=/_N_WCS_DIR/_N_SHAFT_WPD
```

When the file is re-imported into the control, it is stored in this path.

If no path name is specified, then files with identifier `.SPF` are stored in `SPF.DIR` (subprograms), files with extension `.INI` in the working memory and all other files in `MPF.DIR` (parts programs).

Example of file with path name:

```
%_N_SHAFT_MPF
;$PATH=/_N_WCS_DIR/_N_SHAFT_WPD
N10 GO X... Z...
...
M2
```

NC-active data

A backup of an entire directory of NC active data is saved in a single file whose name starts with `COMPLETE...`

All NC-active data, with the exception of compensation data, are saved in file `INITIAL.INI`.

Using commands

- `COMPLETE` or
- `INITIAL`

you can set up an `INI` file:

`INITIAL.INI` that includes all areas (see also Section "Directory structure").

/!AD/, Installation & Start-up Guide

Formats

Files can be stored in two different formats in archive files:

- a) Punched tape/ASCII format
- b) PC/binary format

- With the "Data out" function, files are **always** stored in an archive file with one of the formats named below.
- Files can be stored **without** format conversion only when the "Manage/copy data" function is used.

7.3.1 Punched tape format

1. Only files with displayable characters, i.e. files set up in the text editor, can be saved in this format. Binary data are excluded.
2. Files in punched tape format can be edited with the text editor.
3. Files can be set up externally in punched tape format provided that they are formatted in compliance with the format specified below.
4. If a file is set up manually, it must begin with %<name>, "%" must be typed in the first column of the first line. An archive in punched tape format may contain several files, each of which must begin with %<name>.

The structure of archive files in punched tape format is as follows:

```

<Leader>                                ;can be included
%<File name1>
; $PATH=<Path name1>                    ;can be included
1st block   LF                          ;contents of file 1
2nd block   LF
...         LF
last block  LF

%<File name2>
; $PATH=<Path name2>                    ;can be included
1st block   LF                          ;contents of file 2
...         LF
last block  LF
...
last block  LF                          ;contents of file n
<Trailer>                                ;can be included

```

<Leader>

Information of any type (characters with ANSI values < ANSI value 32 (blank)) which is not part of the useful data on the punched tape.

They might be positioned at the beginning of the tape so that it can be inserted into the punched tape reader.

When the archive file is read, a check is made to determine whether it was saved with a leader. If it was, then it is read in again with a leader.



7.3 Formats for saving and importing data

LF	Character for block end/new line; ANSI value 10 (0x0A)
CR LF	ANSI value 13 (0x0D)
%	<p>Identifier positioned in front of a file name.</p> <p>The identifier must be positioned in the first column of the relevant line (at beginning of block).</p>
File names	<ol style="list-style-type: none"> 1. File names can include the characters 0...9, A...Z, a...z or _ and must not be more than 24 characters long. 2. File names must have an identifier of 3 characters (_xxx). 3. Files in punched tape format can be set up externally or edited with an editor. The file name of a file stored internally in the NC memory starts with "_N_". <p>A file in punched tape format begins with %<name>, "%" must appear in the first column of the first line.</p>
Examples:	<pre>%_N_SHAFT123_MPF = parts program SHAFT123 or %Flange3_MPF = parts program Flange3</pre>
;\$PATH=	<p>Path statement; identifier in front of a path name.</p> <p>The path statement must always be programmed as the next block after the file name.</p> <p>The ";" character in the path statement must be positioned in the first column of the relevant line (at beginning of block).</p>
Path names	<ol style="list-style-type: none"> 1. Path names end in _DIR (directories) or _WPD (workpieces). 2. Path names may contain the characters 0...9, A...Z, a...z or _. 3. Paths must be specified absolutely (starting with "/"). The separator for the directory hierarchy is "/". 4. A path in punched type format begins with ; \$PATH=<pathname> in the first column of the program. <p>Path names in punched tape format start with _N_ and end in _DIR (any directory) or _WPD (workpieces directory).</p>



Example:

```
;$PATH=/_N_WCS_DIR/_N_PIVOT_WPD
Workpiece directory PIVOT in directory Workpieces
```

The data listed after the file name/path name belong to the file with the name specified after "%" in the directory specified after ";\$PATH=".

<Trailer>

Any information (characters with ANSI values < ANSI value 32 (blank) and not equal to ANSI value 10 (0x0A)) which is not part of the useful tape data.

Search strategy when no path is named

If no path is named in the punched tape format, the specified file name must be interpreted when the file is read into the control so that the file can be stored at a suitable position in the file tree.

Files are stored in the file tree according to the following strategy:

File name in tape format	Converted internal file name	Interpreted internal path	Stored in directory
%* _INI	N * _INI	/ N NC_ACT DIR	NC-active data
% N * XXX	N * XXX	/ N XXX DIR	XXX / N NC DIR
%MPFn	N MPFn MPF	/ N MPF DIR	Parts programs
%SPFn	N SPFn SPF	/ N SPF DIR	Subprograms
%Ln	N SPFn MPF	/ N SPF DIR	Subprograms
%*	N * MPF	/ N CLIP DIR	Clipboard

* = any file name

n = any program number (e.g. MPF123)



- The search strategy is applied only if no path has been named. Paths detected using the search strategy are otherwise overwritten by the " ;\$PATH=" statement.
- Spaces in the name are ignored.

Examples

1. *.MPF files

- PC format:

Parts program
%MPF123

Directory: Parts program
(/_N_MPF_DIR)

- Punched tape format:

Parts program
%_N_MPF_MPF

Directory: Parts program
; \$PATH=/_N_MPF_DIR

2. *.INI files

- PC format:

Parts program
%COMPLETE_TEA_INI

Directory: NC-active data
(/_N_NC_ACT_DIR)

- Punched tape format:

Parts program
%_N_COMPLETE_TEA_INI

Directory: NC-active data
; \$PATH=/_N_NC_ACT_DIR

3. Parts programs with name which cannot be assigned

- PC format:

Parts program
%HUGO

Directory: Clipboard
(/_N_CLIP_DIR)

- Punched tape format:

Parts program
%_N_HUGO_MPF

Directory: Clipboard
; \$PATH=/_N_CLIP_DIR

7.3.2 PC format



Files which contain non-displayable characters/binary format can only be saved in PC format.

- Some file types such as ARC, BOT, AWB, TRC, BIN, BMP, ... can be saved **only** in PC format.

The PC format contains a header in which a checksum specifying the useful contents is entered. This sum is checked when the file is re-imported to ensure that all file contents have been transferred correctly.

- A PC format may also include commands such as NC RESET, PLC_STOP or PLC_MEMORYRESET. For this reason, start-up and update archives are always stored in PC format.
- If you save files in PC format and then edit them with a text editor, you will not be able to re-import them again. The file cannot be edited or else the checksum will no longer be correct.
- Start-up and update data must always be saved in PC format.

7.4 RS-232 interface parameters

Notes on operation can be found in 7.5.2 Setting the RS-232 interface. Information about the configuration of the interfaces can be found in /IAM/, IM4 Installation and Start-Up Guide HMI Advanced.

Protocol

As soon as RS-232 transmission starts, detailed messages indicating the current transmission status are output in the dialog line of SERVICES. These messages are:

"Wait for CTS signal"

"Wait for DSR signal"

"Wait for Xon character"

"Data transfer active"

The following protocols are supported for transmission via RS-232:

- XON/XOFF and RTS/CTS,
- Software Flow Control and Hardware Flow Control

XON/XOFF (Handshake)

It is possible to set the two modes for RS-232 transmissions under "Interface", i.e. Wait for Xon for data receive and Send Xon for data transmission. The default setting is H11 or H13.

One possible way of controlling transfer is to use control characters XON (DC1, DEVICE CONTROL 1) and XOFF (DC3). If the buffer of the peripheral device is full, it sends XOFF and as soon as it can receive data again (= default) it sends XON.

RTS/CTS (Handshake)

The RTS signal (Request to Send) controls the send mode of the data transmission equipment:

Active: Data can be transmitted.

Passive: The CTS signal (Clear to Send) is the acknowledgment signal for RTS and confirms that the data transmission equipment is ready to send.

Transmission

In HMI software version 6.2 or later, there is also the option of transmission via secured protocol (ZMODEM protocol).

Baud rate**Normal/secured**

Secured transmission is set in conjunction with the RTS/CTS Handshake for the selected interface. "Normal transmission" is the default setting.

The protocol setting is addressed in:

- Data input/output
- Series installation and startup/upgrade

in conjunction with RS-232 or PG.

SinuCom PCIN software is required on the external PC/PG as a partner.

Input: By selection in display "Interface" under "baud rate" with the selection key

300 baud
600 baud
1200 baud
2400 baud
4800 baud
9600 baud
19200 baud (default)

:

115200 baud

A baud rate of up to 115 kbaud can be set. The usable baud rate depends on the connected device, cable length and electrical environmental conditions.

Data bits

Number of data bits for asynchronous transmission.

Input: By selection in display "Interface" under "Data bits"

- 7 data bits
- 8 data bits (default)

Parity

Parity bits are used to detect errors:

The parity bits are added to the coded characters to make the number of places set to "1" an odd number (odd parity) or an even number (even parity).

Input: Selected in the "Interface" display under "Parity"

- No parity (= default)
- Even parity
- Odd parity

Stop bits

Number of stop bits for asynchronous transmission.

Input: Selected in the "Interface" display under "Stop bits"

- 1 Stop bit (= default)
- 2 Stop bits

Special functions

The following special functions are also provided. These can be activated in the "Interface" display.

A checkbox with a cross in it means: special function active.

Stop on end of transmission character

- Active: Text mode: The end of transmission character is active.
- Inactive: Binary mode: The end of transmission character is not evaluated.
Default value for end of transmission character is hexadecimal 03 (ETX).

With leader and trailer

- Active: Skip leader on input, output 120x0(hex) on output (feed before and after data).
- Inactive: Both leader and trailer are read in.
No 0(hex) leader on output.
Read-in is automatically recognized.



Saving RS-232 interface settings

Archive format

- Binary format (PC format)
 - Punched tape format with LF
 - Punched tape format with CR LF
- are available

Time monitoring (always active)

- Active: Transmission is aborted after the specified number of seconds in the case of transmission errors or end of transmission (without end of transmission character). This function is controlled by a timer that is activated with the first character and reset every time a character is transmitted.
- Inactive: Transmission is not aborted.

The time monitoring function can be set (seconds).

In HMI Advanced software version 6.2 or later, RS-232 interface settings can be stored in specific files in the RS-232.DIR directory. The parameters in a file can be assigned to one of the interfaces. Additional functions are planned for managing/updating these files. See Setting the RS-232 interface.

7.4.1 Interface parameters

Parameters for archiving with PG/PC

Default setting: RS-232 PG/PC

- Interface: COM2
- Protocol: RTS/CTS
- Parity: none
- Stop bits: 1 with leader and trailer
- Data bits: 8 Stop with end of transmission character
- Baud rate: >= 9600
- Archive format: Binary format (PC format) Time monitoring (sec.): 04

This setting allows files in SINUMERIK 840D PC format to be archived and imported.

"Stop with end of transmission character" must not be selected for transmission of MSD and FDD files.

With ASCII data, other settings are possible. These must agree with those of the PG programming unit. Cable 6FX 2002-1AA01 is intended for this.

Parameters for DIN programs

Default setting: RS-232 user

Interface: COM1
 Protocol: RTS/CTS
 Parity: none
 Stop bits: 1 with leader and trailer
 Data bits: 8 Stop with end of transmission character
 Baud rate: 9600

Archive format: punched tape with ... Time monitoring (sec.): 04

With this setting, programs conforming to DIN (beginning with %) are read in or out.

7.5 Operator interface

7.5.1 Services basic display

All programs/data stored on the hard disk or in the NC memory are listed in the "Services" basic display.

Name	Type	Loaded	Length	Date	Enable
Archive	DIR			22/10/2001	X
Definitions	DIR	X		15/01/2002	
Display-machine-data	DIR			09/11/2000	X
Manufacturer-cycles	DIR			10/01/2002	X
NC-active-data	DIR			09/11/2000	X
Part-programs	DIR	X		15/01/2002	
Subprograms	DIR	X		15/01/2002	
User-cycles	DIR			10/01/2002	X
Workpieces	DIR			10/01/2002	X

Free memory: Hard disk : 7.724.150.784 NCU : 9.836.664

Explanation of basic display

Display of the current file tree

The following file attributes can be displayed for each file (depending on the default setting):

Name

Directory name/file name

Files with a maximum name length of 25 characters can be managed on the HMI.

Type

Specifies the file type which matches the file identifier.

Loaded

To execute a program in the NC (through NC Start), it must be loaded into the NC main memory. To ensure that the memory does not become overloaded, however, related programs and data can be loaded explicitly (from the hard disk to the NC memory) and unloaded again (from NC memory back to hard disk).

The current status of a file is indicated by an "X" in a column under "Loaded": File loaded, file can be selected and executed with an NC Start.

Notice: Data may only be loaded for programs for which an enable has been set!

Length

File length in bytes (directory length is not displayed)

Date

Date of creation or date of last modification to file

Enable

Enable (=selection/right to execute) activated "X" or not activated " " When you set up a program, you may not necessarily be able to process it immediately with an NC start (e.g. if it is not ready or requires testing).

To indicate that a program is ready for NC Start, it is possible to enable or disable the program.

The current state of a file is indicated by an "X" in the "Enable" column (= enable activated).

Access rights

There are also 5 access rights for each file:

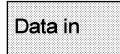
- Read corresponds to level 5
- Write corresponds to level 3
- Execute corresponds to level 7
- Show corresponds to level 2
- Delete corresponds to level 1

The access right for each file is indicated in the file tree.

Not every operator should be able to edit data and programs on the control. Access levels are therefore defined for each file. They range from level 0 (SIEMENS password) to level 7 (keyswitch 0).

A description of how to set the access rights is given in Chapter "Properties".

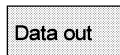
The access rights are not displayed as the numbers 1-7 in the display, instead the display only includes objects which can be executed depending on the rights assigned when the control was started.

A rectangular button with a light gray background and a thin black border, containing the text "Data in".

Horizontal softkeys

Read in archives/files

- RS-232
- PG
- Diskette (if diskette drive is installed)
- Archive from (directory "Archive" on the hard disk)

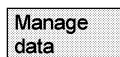
A rectangular button with a light gray background and a thin black border, containing the text "Data out".

Read out archives/files

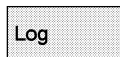
- RS-232
- PG
- Diskette (if diskette drive is installed)
- Archive to (directory "archive" on the hard disk)

A rectangular button with a light gray background and a thin black border, containing a greater-than symbol (>).

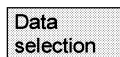
You can archive data for a series start-up. The softkey is password-protected.

A rectangular button with a light gray background and a thin black border, containing the text "Series start-up".A rectangular button with a light gray background and a thin black border, containing the text "Manage data".

Files/directories can be created, loaded, saved, deleted or copied and their attributes can be displayed/changed.

A rectangular button with a light gray background and a thin black border, containing the text "Log".

Current actions, errors and any prompts are displayed in the job list. Prompts must be acknowledged. The "Job log for PG" lists, for example, errors that have occurred in data transmission from/to the PG.

A rectangular button with a light gray background and a thin black border, containing the text "Data selection".

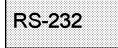
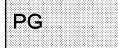
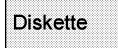
Under softkey "Data selection", you can choose the directories that you wish to have displayed in the "Services" basic display.

A rectangular button with a light gray background and a thin black border, containing the text "Interface".

You can set the interface parameters for the COM1 and COM2 interfaces under this softkey (see also Section "Interface parameters").

Vertical softkeys

The vertical softkeys allow you to select the source area (for data import) or the target area (for data export). The yellow title in the window indicates the area.

A rectangular button with a light gray background and a thin black border, containing the text "RS-232".A rectangular button with a light gray background and a thin black border, containing the text "PG".A rectangular button with a light gray background and a thin black border, containing the text "Diskette".A rectangular button with a light gray background and a thin black border, containing the text "Archives".A rectangular button with a light gray background and a thin black border, containing the text "NC card".

- RS-232
- PG
- Diskette
- "Archives" directory on the hard disk
- "Archives" directory on NC card

7.5.2 Setting the RS-232 interface

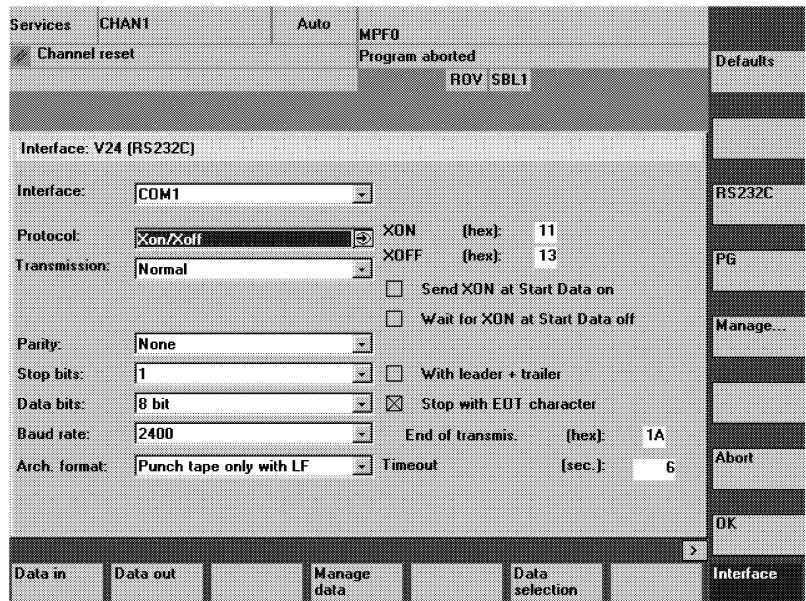


Function

You can output files to an external device or read them in from there via the RS-232 interface of the SINUMERIK 840D/810D. The RS-232 interface and your device must be compatible. The control provides you with an input screen form for this purpose in which you can define the specific data for your device.

You can set separate parameter settings for each RS-232 interface:

1. RS-232
2. PG/PC



Sequence of operations



Interface

Select softkey "Interface".
The vertical softkey bar changes.
Select the interface you want to be parameterized:

Default settings

The default settings for "RS-232 interface" or "PG interface" are accepted depending on which of the interfaces is currently selected for parameterization. If a file is assigned to the interface this assignment is substituted. The file name is removed from the softkeys.

RS-232

- RS-232 (default); in SW 6.2 or later, a parameter file is assigned to the current interface and the name of this file is displayed instead of RS-232. "Printer" in the display above.

PG

- PG/PC; in SW 6.2 or later, a parameter file is assigned to the current interface and the name of this file is displayed instead of PG.

Management

An additional screen form is displayed, which can be used to create, delete, copy, assign and display RS-232 parameter files. See **Management**. (HMI Advanced SW 6.2 or later).

Abort

Return to previous screen form

OK

The current settings are actively set or saved in the parameter file (SW 6.2 or later).

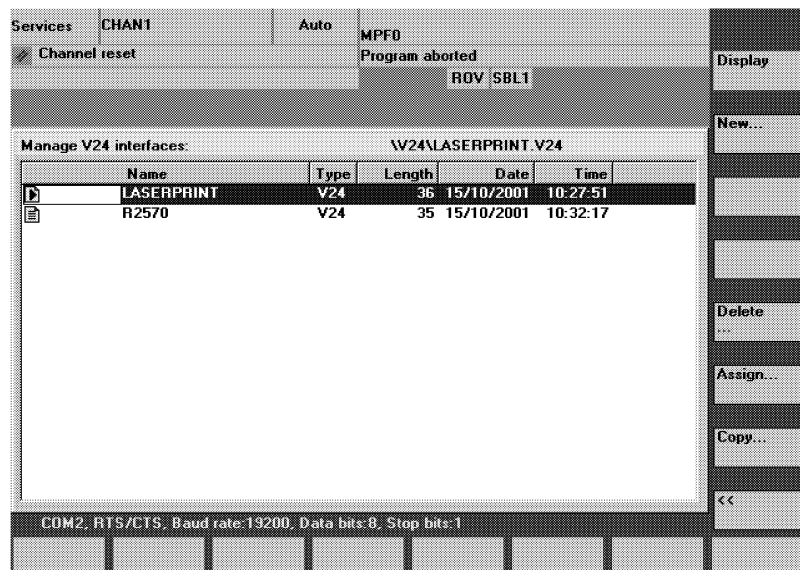
If the name of a parameter file is displayed in the title bar, the interface parameters that are displayed are stored in this file. Saving in a file with the setting "Interface: none" is rejected with an error message. The settings are actively set for the current interface if no file name is displayed in the title bar or if this file is assigned to the current interface. See **Management**.

For interface parameter settings, see also Section "RS-232 Interface parameters" or Section "Interface parameters".

The RS-232 interface screen form is displayed if:

- An RS-232 parameter file is selected in the Services basic display and confirmed with the "Input" key (available in HMI Advanced SW 6.2 or later).
- The softkey "Interface" is pressed in the Services basic display. In this case, the parameters for the RS-232 and PG interfaces are available for editing as before. The parameters for the interfaces are saved independently from the RS-232.DIR directory.

Management



The vertical softkeys offer the following options:

The parameters of the selected RS-232 parameter file are displayed in the "RS-232 interface" display. The file name is displayed in the title bar of this screen form.

After entering the file name (max. of 10 characters) the current RS-232 parameters are written to a new file in the RS-232.DIR directory.

Display

New ...

Delete ...

The selected RS-232 parameter file is deleted after being confirmed by the user. If a file that is assigned to an interface is deleted, the file name is also removed from the softkeys for the interface.

Assign...

In SW 6.2 or later: An RS-232 parameter file is assigned to the current interface via a dialog box. The corresponding parameters are actively set and the file name is displayed on the relevant softkeys (instead of RS-232 or PG). The status bar indicates the parameters saved in the file.

The current assigned interface can be changed temporarily using the vertical softkeys.

Copy ...

1. a) The current RS-232 parameter file is copied to the RS-232.DIR directory after entering a new name (maximum of 10 characters).
 - b) Or is copied to a diskette by pressing the softkey "Diskette".
 - c) Or is copied to the clipboard by pressing the softkey "Clipboard".
2. If several files are selected simultaneously, only copying to a diskette or to the clipboard is active.

When copying to a diskette or to the clipboard, a new file name cannot be specified.

Status bar

The status bar for the current selected parameter file displays the following data:

COM1 or COM2
Xon/Xoff or RTS/CTS
Baud rate
Number of data bits
Number of stop bits

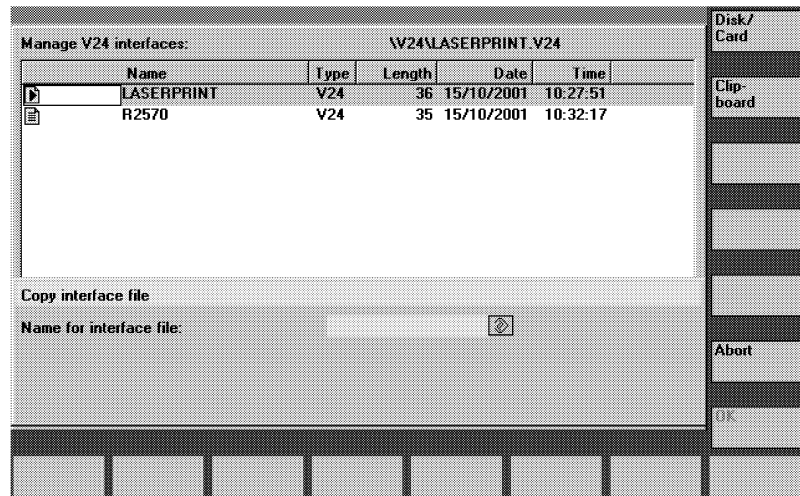
<<

Return to "Interface: RS-232" screen form

Error handling

Only RS-232 parameters with COM1 or COM2 entered in the "Interface:" field are saved.

If an error is detected while displaying/assigning an RS-232 parameter file, the file name is displayed in the title bar of the Interface: RS-232 screen form.



Secondary display for copying the RS-232 interface parameter file

Clearing assignment

If "none" is specified in the interface field in "Interface: RS-232", and an RS-232 parameter file is assigned, the assignment is cleared by pressing OK.

Booting

After booting, the assigned files are displayed on the softkeys RS-232 and PG and their settings are actively set for the interfaces. If the assigned files cannot be found, the default text (RS-232 or PG) is displayed again. The corresponding interface is then assigned the previous RS-232 parameters.

7.5.3 Reading in data



Function

Reading in archives and files: The following are possible source areas

- A device connected to the RS-232 interface (e.g. PC)
- A programming device
- A diskette drive
- Archive (i.e. the "Archives" directory) in the directory tree (even if it is not displayed under "Data selection").
- The NC card (if a flash file system is formatted on the NC card)

- When archives are read in, their format (punched tape/PC format) is recognized automatically.
- Data with longer file names (> 8 + 3 characters) can be read in from diskette.



Data in

RS-232

PG

Diskette

Archives

NC card

Sequence of operations

The "Programs/Data" file tree is displayed.
The vertical softkey bar changes.

Mark the file into which you wish to read the archive.

Select the source area (see title in window):

- RS-232 interface
Interface is made ready to receive. (1st RS-232 interface)
- Programming device (2nd RS-232 interface)
- Diskette drive: Shows the contents of the diskette. Select the archive you wish to import.
- Opens the "Archives" directory on the hard disk. Select the archive you wish to import.
- The contents of the archives directory on the NC card are displayed.
Select the archive you wish to import.

Users with access authorization to level 3 and higher can access this softkey. It is displayed only if archive `_N_ORIGINAL_ARC` is stored on the NC card.

7.5.4 Reading out data



Data out

RS-232

PG

Diskette

Archives

NC card

Function

Creating archives:

With the "Data out" function, the source area (see title in window) for data transmission is the displayed directory tree.

Possible target areas are:

- A device (such as PC) connected to the RS-232 interface,
- A diskette drive,
- "Archives" directory on hard disk.
- Free memory areas on the NC card

Sequence of operations

The "Programs/Data" file tree is displayed.

The vertical softkey bar changes.

You have marked the data that you wish to save/archive.

Select the target area (see title in window):

- RS-232 interface (programming device) (1st RS-232 interface)
The system prompts you to start the data receiving device.
- RS-232 interface (programming device) (2nd RS-232 interface)
The system prompts you to start the data receiving device.
- Diskette drive: The diskette contents are displayed.
Enter the name of the new archive file.
- The contents of the "Archives" directory on the hard disk are displayed.
Enter the name of the new archive file.
- The contents of the archives directory on the NC card are displayed.
Enter the name of the new archive file.



Start

When exporting data from diskette/archive, press the softkey "Start", otherwise the control is ready to receive immediately.

The data transfer is initiated. The vertical softkey bar changes. The softkey labeling of the target area changes to "Stop". To abort data transmission, press the relevant softkey again.

Additional notes

Archives to be stored on diskette need **not** fit completely on a single diskette. Distribution of the archive over several diskettes is supported.

The two RS-232 interfaces (RS-232 and PG) cannot be active simultaneously.

Note:

When archiving workpieces that contain job lists of the same name, in the case of m:n you are asked whether the job lists to be unloaded are to be executed. You can terminate the action with "Cancel", otherwise all joblists are executed and archiving is then started.

7.5.5 Log



Function

You can use the log function to view a job log, e.g. for "Manage data".



Sequence of operations



Services

The "Services" operating area is selected.



Log

The "Log" window is opened. The source or target of the job is displayed in the header.

The vertical softkey bar changes.

Manage data

RS-232

PG

Diskette



Jobs are assigned to the vertical softkeys, i.e. jobs for

- "Manage data"
- RS-232 interface
- Programming device
- Diskette drive

You can switch between windows with these softkeys.

The softkey label displays "...Stop" if a job is not running. You can abort a job in progress by pressing the "Stop" softkey again.

A message line is displayed in the "Error list" field to indicate whether or not errors have occurred during data transmission.

In the case of queries, the prompt "Please acknowledge query in log window" appears in the dialog line.

Confirm the prompt with one of the vertical softkeys:

- Do not confirm
- Confirm all
- Confirm
- Change name/type
- Abort complete job

No

Yes all

Yes

Name/type

Stop

Delete log

The currently displayed log is deleted.

The log window is closed on the last log.

7.5.6 Importing/exporting ISO programs



Function

ISO programs can be imported and exported to and from the HMI in punched tape format.

Additional notes

You can import and export programs from the FANUC 0 control system.

The punched tape format for ISO programs (ISO punched tape format) is different from the Siemens HMI punched tape format.

The first line of a punched tape in ISO format must have the following format: %<Title>LF or %<Title>CRLF.

The title can be omitted and blanks can be skipped. The title may not start with one of the following characters: 0...9, a...z, A...Z or _.

No title is generated when a punched tape is generated in ISO format.

Siemens program headers are introduced by %<Name> and path ;PATH=<Path> in the next block.

ISO program headers are recognized from O<xxxx (Title)> or :<xxxx (Title)> without path in the next block.

x stands for a number between 0 and 9. Between one and four digits can be specified, leading zeros can be omitted.

During export, ISO program headers are only tagged with O<...> and not with :<...>.

Import



The procedure for importing a punched tape in ISO format is the same as the procedure used to import a regular punched tape archive in the "Services" operating area with "Data in". During the import, the system detects automatically whether the archive to be imported is stored in binary/PC, punched tape or ISO punched tape format.

ISO programs which have been imported (e.g. O1234 or :1234) are stored in the NC either as main programs (e.g. _N_1234_MPF) or under a workpiece name defined by the machine manufacturer.

ISO punched tape with two ISO programs:

```
%
O1026 (HYDRAULICBLOCK)
N20 G00 G80 G90 G40 G17
N40 (NC-SPOTDRILL) T01 M06
N50 G55 G43 Z20. H01 S1000 F100 M03
N55 X10. Y-8. M08 T02
(...)
N690 Y-43.
N700 G80 Z35.
N710 T00 M66
N715 G53 Y0. Z0.
N720 M30
:1127 (ANGLE)
N10 (2. SPEEDRANGE)
N20 G00 G80 G90 G40 G17
N120 (TWDRIILL 11) T01 M06
N130 G55 G43 Z20. H01 S2300 F460 M03
(...)
N180 Y-72.
N190 G80 Z35.
N195 T00 M66
N200 G53 Y0. Z0.
N210 M30
%
```

This punched tape generates two programs when imported:

_N_1026_MPF and _N_1127_MPF; the title after the program number is retained:

```
Program _N_1026_MPF:
(HYDRAULICBLOCK)
N20 G00 G80 G90 G40 G17
N40 (NC-SPOTDRILL) T01 M06
(...)
N710 T00 M66
N715 G53 Y0. Z0.
N720 M30
```

```
Program _N_1127_MPF:
(ANGLE)
N10 (2. SPEEDRANGE)
N20 G00 G80 G90 G40 G17
(...)
N200 G53 Y0. Z0.
N210 M30
```

Export



The procedure for generating an archive in ISO format is the same as the procedure used to generate a Siemens punched tape archive in the "Services" operating area with "Data out". The current output format determines whether the archive is created in binary/PC, punched tape or ISO punched tape format.

The output format can be changed in the "Services" operating area with "Interface" -> "RS-232"/"PG" -> "Archive format" and for diskette or archive with "Data out" -> "Diskette"/"Archive" -> "Archive format in the target list box".

You can choose between the following formats:

- Binary (PC)
- Punched tape with LF only
- Punched tape with CR + LF
- Punched tape/ISO with LF only
- Punched tape/ISO with CR + LF

All programs with names in the format `_N_xxxx_MPF` for NC files or `xxxxx.MPF` for HMI data management files (x is a digit between 0 and 9) are treated as ISO programs when creating an archive in ISO punched tape mode. Between one and four digits can be specified.

The file DINO.INI can be used to define the output directories for parts programs in ISO format.

See /IAM/, Installation and Start-Up Guide HMI/MMC, Start-up Functions for the HMI (IM4).

- If both ISO programs and Siemens programs are selected for the creation of an ISO punched tape archive, an ISO punched tape is generated without an alarm or message output; the punched tape contains Siemens program headers in addition to the ISO program headers.

If a Siemens program is followed by an ISO program, a `%<LF>` or `%<CR><LF>` is inserted in front of the ISO program header, depending on the output format, because the character string `O<four digits>` or `:<four digits>` in DIN code cannot be assigned uniquely to a new program.

These "hybrid" ISO punched tape archives can be imported to the HMI again, although the % character would abort any attempt to import the archives into third-party control systems (because the % character indicates the end of the punched tape in ISO format).


```

%
%_N_TEST1_MPF
;$PATH=/_N_WCS_DIR/_N_TEST_WPD
N40 G01 X150 Y150 Z150 F6000
N50 G90 G0 X0 Y0 Z0 G53
; ...
N500 G02 z100 x50 k-50 i0
N510 z50 x100 k0 i50
M30 ;Transition from Siemens prog. to Siemens
prog.
%_N_TEST2_MPF
;$PATH=/_N_WCS_DIR/_N_TEST_WPD
N40 G01 X150 Y150 Z150 F6000
; ...
M30 ;Transition from Siemens prog. to ISO prog.
%
O1127 (ANGLE)
N10 (2. SPEEDRANGE)
N20 G00 G80 G90 G40 G17
(...)
N200 G53 Y0. Z0.
N210 M30
%
```

- If both ISO programs and Siemens programs are selected for the creation of a Siemens punched tape archive, a conventional punched tape is generated which contains only Siemens program headers, i.e. the ISO programs contain Siemens program headers.

```

%_N_TEST1_MPF
; $PATH=/_N_WCS_DIR/_N_TEST_WPD
N40 G01 X150 Y150 Z150 F6000
N50 G90 G0 X0 Y0 Z0 G53
; ...
N500 G02 z100 x50 k-50 i0
N510 z50 x100 k0 i50
M30 ;Transition from Siemens prog. to Siemens prog.
%_N_TEST2_MPF
; $PATH=/_N_WCS_DIR/_N_TEST_WPD
N40 G01 X150 Y150 Z150 F6000
; ...
M30 ;Transition from Siemens prog. to ISO prog.
%_N_1127_MPF
; $PATH=/_N_WCS_DIR/_N_TEST_WPD
(ANGLE)
N10 (2ND SPEEDRANGE)
N20 G00 G80 G90 G40 G17
(...)
N200 G53 Y0. Z0.
N210 M30

```

- The difference is irrelevant for archives in binary format.

Additional notes

Binary files cannot be output in ISO punched tape format.

7.6 Manage data

7.6.1 Standardized program and data management (SW 6.3 and higher)



Function

In the "Manage data" window in software version SW 6.3 and higher, it is possible to manage workpieces, part programs, subprograms, standard cycles, user cycles and manufacturer cycles with the same number of softkey actuations as in the main Program window. This standardized operating sequence applies to the following applications:

- Creating new directories and programs
- Loading programs and files to the program memory
- Loading programs and files from the program memory to the hard disk (unloading the program memory)
- Creating new files
- Copying and pasting files
- Deletion operations

Comparison

Horizontal softkeys

Main screen in operating area

Program

Directories/workpieces/
programs/files

Main screen in operating area

Services

Data/files/comments
definitions/part programs

Main menu:

New ...
Load HD->NC
Unload NC->HD
Simulation
Manage programs...
Selection
Save setup data

Manage programs:

New ...
Copy
Paste
Delete
Rename
Change enable
<<

Manage data:

New ...
Load HD->NC
Unload NC->HD
Delete
Properties
Copy
Paste

7.6.2 Creating a new file/directory



Manage
data

New

END

OK

Sequence of operations

Select softkey "Manage data".

The horizontal and vertical softkey bars change.

The "New" window appears.

Enter a new file name.

If the preassigned file type does not match, use the "End" key to switch between input field "Name" and "File type".

Select the new file type.

The new directory/file is set up in the directory overview.

7.6.3 Loading/unloading



Manage
data

Load
HD->NC

Unload
NC->HD

Sequence of operations

Select softkey "Manage data".

The horizontal and vertical softkey bars change.

Position the cursor on the desired file.

The selected file is deleted from the hard disk and loaded to the NC memory. In the display the entry "X" = loaded is added to the file.

See also Chapter "Job list".

The selected file is deleted from the NC memory and loaded to the hard disk. In the display, the entry " " = not loaded is added to the file.

If you want to "load/unload" a workpiece directory and a job list with the name of the directory exists in the job list, that job list is executed.

Loadable compile cycles, SW 6.3 and higher

If a job list does not exist, all the files in that directory are loaded/unloaded.

You can use the HMI user interface to transfer the loadable compile cycles (CC) easily from an NC card to the NCK control. Further storage locations are any external drive, such as a disk drive, or network drives 1 to 4. This storage location appears in the Service operating area under "Manage data".

You can also use the SinuCom NC or SINUCOPY-FFS commissioning tool to copy the compile cycles onto the control.

Load CC

Each time the NC is booted, all loadable compile cycles with the extension **.ELF**, which are available in the `/_N_CCOEM.DIR` directory, are automatically loaded into the NCK system software. Loadable compile cycles are always loaded on an NCU reset.

For a detailed description of how to install compile cycles, please see: `/FB3/`, TE0 Installation and Activation of Loadable Compile Cycles.

7.6.4 Copying/inserting



Function

You can copy

1. a single file,
2. several files or
3. a complete directory.

Copy CC

Following a reset, the optionally available compile cycles are stored on the NCU in the Flash File System (FFS) in directory `/_N_CCOEM.DIR` and, in SW 6.3 and higher, can be copied selectively from/to an external drive, disk or NC card.

It is thus possible to archive individual compile cycles.

The "Insert" function is available in addition to "Copy".



You can display any existing compile cycles in the Diagnosis operating area under `Service displays/Version/Compile cycles`.



Manage
data



Copy

Programs/
data

Clipboard

NC card

Diskette

Paste



Sequence of operations

Select softkey "Manage data".

The horizontal and vertical softkey bars change.

Select the source files that you wish to copy.

A second window for the target directory (see title in window) is opened.

If available, select the target device via the vertical softkeys.

The contents of "Programs/data" are displayed.

The contents of the "Clipboard" directory are displayed.

Archive files or complete directories can be copied from an existing NC card.

If a diskette drive is installed, you can copy to or from diskette. The diskette contents are displayed.

Select a target directory.

The source files are copied to the target directory you have selected.

With this copying action in operating area "Services" the names remain unchanged.

See Section "Copy/Insert" in the "Program" operating area.

7.6.5 Deleting



File

Directory



Manage
data

Delete

OK



Function

You can delete a single file or a group of files (multiple selection).

You can delete a directory and all its contents.

The system settings relating to deletion determine whether a prompt appears before files/directories/data are finally deleted (see also Section "Start-up" operating area).

Sequence of operations

Select softkey "Manage data".

The horizontal and vertical softkey bars change.

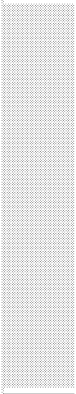
Position the cursor on the desired file.

A query window appears.

You can delete the highlighted file by pressing "OK".

Archives stored on the NC card can be deleted by selecting "NC card" under "Manage data".

7.6.6 Changing the properties of a file/drive/archive

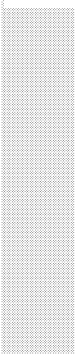


Manage
data

Properties



INSERT



Function

This function allows you to view the contents of a file (or directory) as well as other information, to view file/directory properties and to change some properties.

In this window, you can

- rename a file,
- convert a file to another file type,
- change the access rights to the file/directory and
- view the contents of readable files.

The contents of text files are displayed.

You can change access levels only if you have the appropriate access rights to do so.

Sequence of operations

Select softkey "Manage data".

The horizontal and vertical softkey bars change.

The "Properties" window is opened.

Enter the changes you wish to make, e.g. rename the file or change the file type.

Renaming a file

Place the cursor on the file name and press the "Edit" key (displayed next to the type list), enter the new file name.

There are two ways of renaming files:

- Renaming the workpiece directory
- Renaming a directory in the workpiece directory

Renaming a workpiece directory:

When you rename a workpiece directory, all the workpiece files under that directory that have the same name as the directory are renamed.

If a job list with the name of the directory exists, the instructions in that job list are also renamed.

Comment lines remain unchanged.

Example:

Workpiece directory A.WPD renamed to B.WPD:

All files with the name A.XXX are renamed to B.XXX, i.e. the extension is not altered.

If a job list called A.JOB exists it is renamed to B.JOB.

If this job list contains instructions of file A.XXX located in this workpiece directory, then that file is also renamed to B.XXX.

Example:

If job list A.JOB contains an instruction

```
LOAD/WCS.DIR/A.WPD/A.MPF
```

it is renamed to

```
LOAD/WCS.DIR/B.WPD/B.MPF
```

However, if a job list contains the instruction

```
LOAD/MPF.DIR/A.MPF or
```

```
LOAD/WCS.DIR/X.WPD/A.MPF
```

the files are not renamed.

Renaming a directory in the workpiece directory:

If you rename the files in the workpiece directory, all files with the same name but a different extension are renamed.

Exception: If a job list of the same name exists in the directory, then this one is not renamed.

Changing the file type

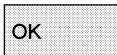
Use the "Edit" key (displayed next to the type bar) to show the list of file types to which the file may be converted.

Only the file types are displayed which are permissible in the directory where the file is located.

Use the "Direction" keys to display the new file type.

Confirm your selection with the "Input" key (displayed next to the file type you have selected).

The file is assigned the new file type.





Additional notes

- No check is made to determine whether or not the file contents may be stored under the new file type!
- The contents of a file are **not** altered when the file type is converted.
- All data types may be stored in the "Clipboard" directory.
- In SW 6.3 and higher, the file type is displayed in the properties window for network drives or disk, and the extension with 0 to 3 letters can be modified in the "Type" input field.

7.6.7 Defining and activating user data (GUD)



Function

Create a definition file for user data:

- in the "Services" operating area using the softkey "Manage data"
- in the "Definitions" directory

Procedure

If you edit a definition file in the NC, a query box asking whether you want the definitions to be activated is displayed when you exit the Editor.

Example:

"Do you want to activate the definitions from the GUD7.DEF file?"

"OK" → A query is displayed asking if the currently active data should be saved. "Do you want to save the previous data for the definitions?"

"OK" → The GUD blocks for the definition file to be edited are saved, the new definitions are activated and the saved data are reloaded.

"Abort" → The new definitions are activated, the old ones are lost.

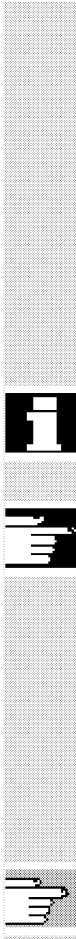
"Abort" → The changes to the definition file are rejected, the associated data block is not altered.

OK

Abort

Unload

If a definition file is unloaded, the associated data block is deleted after confirmation via a query box.



Activate

If a definition file is loaded, a query box appears asking whether you wish to activate the file and/or retain the data. If you do not select activation, then the file will not be loaded.

If the cursor is located in a loaded definition file, the softkey labeling changes from "Load" to "Activate" to activate the definitions. If you select "Activate", you are again asked whether you want to save the data.

Data are saved only in the case of variable definition files, but not with macros.

Additional notes

If there is insufficient memory space available to activate the definition file, then the file must be unloaded. After the memory size has been adjusted, the file must be loaded from the NC to the HMI and back again to the NC. The files are then activated.

Definition and creation of user data

See /PGA/, Programming Guide, Advanced.

7.7 Start-up functions

7.7.1 Series start-up



Function

"Series start-up" means to establish the same initial data status on several control systems. You can select PLC, NC and HMI data and loadable compile cycles, and archive or restore them for a series machine start-up.

Compensation data can be saved at the same time if necessary. The drive data are saved as binary data which cannot be modified.



Sequence of operations

Precondition: The password, e.g. with access level 3 (user) is set.

Press softkey "Series start-up".
The vertical softkey bar changes.



Series
start-up

The "Create series start-up archive" window appears.

Create an archive for the series start-up-file:

You can select which data you wish to save as the archive contents:

- HMI
- NC with compensation data
- Loadable compile cycles (SW 6.3 and higher)
- PLC
- Profibus drives (SW 6.4 and higher)

Archive name:

The suggested archive name depends on the selected area and can be changed if necessary.

Example:

Based on the original archive name MMCNCPLC, the new archive name MMCNCPLC is suggested when you select PLC.

MMC data selection

If you have selected the "MMC" area, you can select the data you wish to archive by pressing softkey "MMC data selection". "MMC data selection" stands for "HMI data selection".

The directory F:\USER is always backed up. Directories \ADD_ON and \OEM can be saved additionally under "Additional products", either completely ("Complete") or just the INI files ("Configuration").

The archiving operation commences when you select the target device.

Archive data to:

- The device connected to the RS-232 interface
- Programming device/PC
- Diskette drive
- "Archives" directory on hard disk
- "Archives" directory on the NC card

Softkey labeling changes to "... Stop". The series start-up archive is created.

RS-232

PG

Diskette ...

Archives

NC card

Read start-up archive

Toggle between functions "Create series start-up archive" and "Read start-up archive".

Reading series start-up archive:

The read-in operation commences when you select the source device connected to the RS-232 interface. The data can be read via:

- The device connected to the RS-232 interface
- A programming device
- The diskette drive
- The "Archives" directory on hard disk
- The archives directory on the NC card

Select an archive file under softkeys "Diskette", "Archives" and "NC card".

RS-232

PG

Diskette ...

Archives ...

NC card

Start

Start reading in the archive. The softkey label changes to "Stop".

Make start-up archive

Switchover from "Read" to "Make".

**SW 6.3 and higher****Additional notes**

For importing a series start-up file from diskette, there must be sufficient free hard disk memory to store the series start-up file.

If a control is configured for M:N (several operator panel fronts and NCUs), a **Power On** must be initiated on the HMI (PCU 50) after a series start-up so that the stations on the bus (NC, PLC, HMI) can be re-synchronized.

Archiving loadable compile cycles (CC):

You can archive loadable compile cycles in the series machine start-up if the following conditions are met:

- At least one CC must be available in the start-up archive in destination directory \NC_CARD.DIR\CCOEM.DIR.
- Selective archiving of individual compile cycles is performed by copying to a data storage device, e.g. disk. Only archives with access rights for series start-up access levels 0 to 4 are read in.

7.7.2 Restoring the original state via NC card



Function

The free memory on the NC card (PCMCIA card) can be used to store a start-up archive.

The archive can be copied onto the NC card by means of SINUCOPY-FFS (on an external programming device/PC).

The series start-up archive can be stored directly on the NC card from the HMI with the name "Original" (see Series start-up – creating a file).



Sequence of operations

Precondition:

The start-up archive named `_N_ORIGINAL_ARC` is already stored on the NC card (in directory `_N_NC_CARD_DIR_N_ARC_DIR`).

Please read machine manufacturer's instructions



Original
state

Select the "Etc" key in the Services basic display and then softkey "Original state".

When you press the softkey, the log window appears with a query "Series start-up active: Execute series start-up?" Confirm to import the data.



Caution

All user-specific NC data (and PLC data depending on contents) will be deleted and replaced by the data from the archive.

7.7.3 Software update



Upgrade

RS-232

PG

Diskette ...

Archives ...

Function

This function supports updating of the NC system software. For this purpose, you can create an update archive. This contains all NC data (like a series start-up archive) including compensation data. Software updates are carried out according to the same principle as series start-ups. The main difference is that drive data are saved and re-imported in ASCII format with software updates (thus allowing these data to be altered if necessary). Another major difference to the series start-up operation is that an update is always performed for the same machine, i.e. it makes sense to import compensation data as well.

For further information, please see Installation and Start-Up Guide for 840D or 810D.

Sequence of operations

Press softkey "Upgrade".

The vertical softkey bar changes.

The "Create update archive" window opens.

The NC data are preset as archive contents.

You can enter any archive name of your choice.

The archiving operation commences when you select the target device.

Data can be archived to

- the device connected to the RS-232 interface
- the programming device
- the diskette drive
- "Archives" directory on hard disk



NC card

- The contents of the archives directory on the NC card are displayed.

Enter the name of the new archive file.

Softkey labeling changes to "... Stop". The update archive is set up.

Read in update archive:

The read-in operation commences when you select the source device connected to the RS-232 interface. The data can be read via:

- The device connected to the RS-232 interface
- A programming device
- The diskette drive
- The "Archives" directory on hard disk
- The archives directory on the NC card

Select an archive file under softkeys "Diskette", "Archives" and "NC card".

Start

Start reading in the archive. The softkey label changes to "Stop".

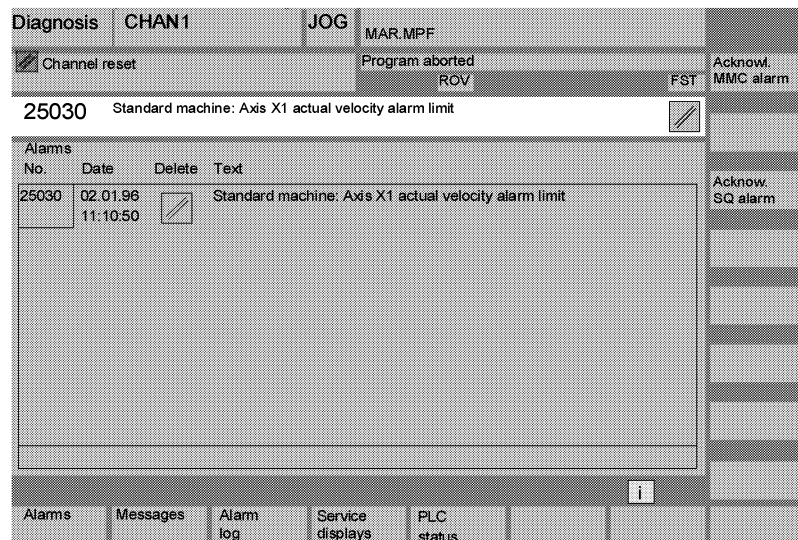
"Diagnosis" Operating Area

8.1	Diagnosis basic display	8-406
8.2	Alarms/messages/alarm log.....	8-408
8.3	Service display	8-410
8.3.1	Service overview (SW 6 and higher).....	8-410
8.3.2	Service axis.....	8-412
8.3.3	Service drive.....	8-413
8.3.4	Service Safety Integrated	8-414
8.3.5	Configuration data	8-418
8.3.6	Communication error log.....	8-419
8.3.7	Action log.....	8-419
8.3.8	Version	8-420
8.3.9	Displaying the version screen for cycles (SW 6.3 and higher).....	8-420
8.3.10	Exporting cycle versions (SW 6.3 and higher)	8-423
8.3.11	Displaying loadable compile cycles (SW 6.3 and higher)	8-423
8.4	PLC status.....	8-425
8.4.1	General	8-425
8.4.2	Changing/deleting values	8-426
8.5	Selecting/creating operand forms for PLC status	8-428
8.5.1	File functions	8-429
8.6	Display NC system resources	8-430

8.1 Diagnosis basic display

A display headed "Alarms" appears when you select the operating area "Diagnosis".

Basic display



Explanation of display

Number	The alarm number is output under "Number". The alarms are output in chronological order.
Date	The date and time at which the alarm occurred is displayed with the date, in hours, min, sec.
Clear criterion	The symbol denoting the alarm abort key is displayed for every alarm.
Text	The alarm text is displayed under "Text".

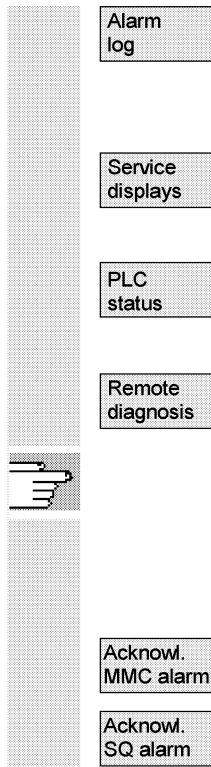
Horizontal softkeys

Alarms

All active alarms are displayed in the "Alarm overview" display.

Messages

An overview of active messages is displayed.



The alarm log of alarms and messages that have occurred is displayed. The log also includes alarms that have already been reset. Default setting for alarm buffer: 150 alarms/messages

You can view updated information about axes and drives installed in your system under softkey "Service displays".

Information on the current status of the PLC memory locations.

The control can be operated externally over a remote connection (e.g. modem) (option).

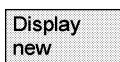
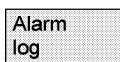
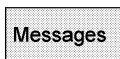
For more information about remote diagnostics please refer to /FB/ F3, Remote Diagnostics.

Vertical softkeys

MMC alarms (alarm number 120...) that have occurred can be acknowledged. MMC alarms correspond to HMI alarms.

SQ alarms (PLC alarms) that have occurred can be acknowledged.

8.2 Alarms/messages/alarm log



Function

You can display a list of alarms and messages and acknowledge them.

Sequence of operations

Alarms:

The alarm overview displays all active alarms with alarm numbers, date, clearance criteria and descriptions.

Clear the alarm by pressing the key that is displayed as a symbol:

Switch device off and on again (main switch)

or NCK POWER ON

Press "Reset" key.

Press "Acknowledge alarm" key.

Alarm is cleared by "NC Start".

Alarm is cleared with the "Recall" key.

Messages:

- PLC operational messages that do not have to be acknowledged (as standard) (configurable).

Alarm log:

A log containing the alarms and messages that have so far occurred is displayed. 150 alarms/messages can be logged as standard.

It also contains the alarms that have already been acknowledged.

This symbol means "Alarm is still active".

The alarm is updated (static display).

Save log

Sorting
old > new

Acknowledgement symbols

By pressing this softkey, the current status of the log is stored in the displayed path and can be archived from here as required.

Changing the chronological sorting sequence of the alarms in the display.

The following acknowledgment symbols apply to the MMC and PLC alarms:

(MMC alarms correspond to HMI alarms).

HMI/MMC alarms:



PLC alarms:



Safety alarms S:



Safety alarms SQ:

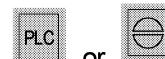


The SQ alarms are cleared with softkey "Acknowledge SQ alarm". The S alarms are display alarms, they do not have to be cleared. The way S and SQ alarms are displayed depends on the settings in the files "MMC.INI" and "MBDDE.INI".

You can determine which acknowledgment symbol is to be displayed for the PLC alarms by an entry in the INI file "DG.INI".

The following symbols are available:

PLC alarms:



Activation of the symbols is described in /IAM/, IM4, Installation and Start-Up Guide HMI/MMC

8.3 Service display

8.3.1 Service overview (SW 6 and higher)



Function

When commissioning digital drives and diagnosing faults, the various operating states of individual machine axes can be visualized with a type of traffic light indicator in the "Service overview" display, which you can open from the "Service display" menu.







Service overview

This overview shows specific enable signals and statuses for each of the available axes 1 to 31. Up to 6 different states can occur on each axis:

SW 6.3 and higher

Columns for machine axes 1 to 31

Traffic light indicator states per axis

Green dot:		The axis is operating normally
Amber dot:		The axis is not ready
Gray dot:		Does not apply to this axis
Red dot:		An alarm is active for this axis
Dash:		The axis is not assigned to a drive
Special character:		Error on reading data e.g. if data does not exist

SW 6.3 and higher

Rows with the drive diagnostic functions

Drive enable (terminal 64/63)
 Pulse enable (terminal 64/48)
 Pulse enable (terminal 663 / SI: drive relay)
 Setup mode (terminal 112)
 PLC pulse enable
 NC speed controller enable

SW 6.3 and higher

DC link status
 Enable pulses
 Drive ready
 Heatsink temperature warning
 Power section in i2t limitation
 Motor temperature warning
 Measuring system 1 active
 Measuring system 2 active



Service displays

Change selection...

Change configurations



Abort

OK

Select axes

Active axes

All axes

Sequence of operations

Select the menu headed "Service displays".

The softkey bars change. The softkeys in the horizontal menu bar allow you to select the various service displays. The vertical softkeys are context-sensitive according to the service display. You can use them to select the axis configuration or define or change a specific axis selection.

Define a specific axis selection

Press the "Change selection..." softkey in the "Service overview" window.

You can compose your own axis list in the "Change configurations" menu. The available axes can be entered in the axis list in any order. Example:

Suppose you want 4 axes to appear in the order 1 3 5 8.

The existing axis list appears in "Change configurations", e.g. 1 7 5. Use the "Insert" key to enter the new axes with a space between each axis number: 1 3 5 8

Axis numbers outside the valid range from 1 to 31 are deleted. A default list ("1 2 3 4") is displayed if you try to enter an empty list.

Returns you to the main "Service overview" menu: the changes to the current axis list are not saved.

Returns you to the main "Service overview" menu: the changes to the current axis list are saved and applied with the displayed information.

Display selected axes

Press the vertical softkey "Select axes".

The window changes to the main "Service overview" menu and the previous axis selection is displayed.

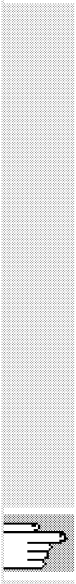
Display active axes

Press the vertical softkey "Active axes" in the "Service overview" window. The active axes appear in the main "Service overview" menu.

Display all axes

Press the vertical softkey "All axes" in the "Service overview" window. The maximum set of possible axes appears in the main "Service overview" menu.

8.3.2 Service axis



Service displays

Service axis



PAGE DOWN PAGE UP

Axis + Axis -

Direct selection...

Function

The information in the "Service Axis" display is used to

- check the setpoint branch (e.g. position setpoint, speed setpoint, spindle speed setpoint prog.)
- check the actual-value branch (e.g. position actual value, measuring system $\frac{1}{2}$, actual speed value), optimize the position control of the axis (e.g. following error, control difference, servo gain factor)
- check the entire control loop of the axis (e.g. through position setpoint/actual-value comparison and speed setpoint/actual-value comparison)
- check hardware faults (e.g. encoder check: If the axis is moved mechanically, then the position actual value must change)
- set and check axis monitoring functions

References: /FB/, Description of Functions D1, Diagnostics Tools

Sequence of operations

Select the menu headed "Service displays".

The softkey bars change. The vertical softkeys are context-sensitive according to the service display. The softkeys in the horizontal menu bar allow you to select the corresponding service displays.

Press the horizontal softkey "Service axis" The "Service axis/spindle" window displays reference values and units for the machine axis together with axis name and axis number.

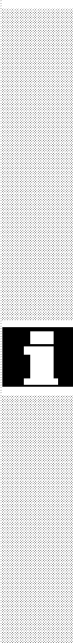
You can page up and down with the "Page keys".

Vertical softkeys for axis selection

The service values of the next (+) and the previous (-) axis are displayed.

The vertical softkey "Direct selection" in the window of the same name allows you to select an axis directly from the list of available axes.

8.3.3 Service drive



Function

The information contained in the "Service drive" display is used to

- check the status of the enable and control signals (e.g. pulse enable, drive enable, motor selection, setpoint parameter set)
- check the status of the FDD/MSD operating modes (e.g. setup mode, parking axis)
- display temperature warnings
- check the current setpoint/actual-value display (e.g. position actual-value measuring system 1/2, speed setpoint, speed actual value)
- check the drive status
- display the current ramp-up phase
- display a group error message (message status class 1)
- display the status messages of the drive (e.g. threshold torque not reached, actual speed = set speed)

References: /FB/, Description of Functions D1, Diagnostics Tools

Sequence of operations

Select the menu headed "Service displays".

The softkey bars change. The vertical softkeys are context-sensitive according to the service display. The softkeys in the horizontal menu bar allow you to select the corresponding service displays.

The "Service drive" window displays information about the axis drive together with axis name and number.

You can page up and down with the "Page keys".

Vertical softkeys for axis selection

The service values of the next (+) and the previous (–) drive are displayed.

The vertical softkey "Direct selection" in the window of the same name allows you to select an axis directly from the list of available axes.

8.3.4 Service Safety Integrated



Function

Three information blocks on Safety Integrated data are offered in HMI Advanced for the selected axis by pressing the softkey "Service SI":

- Status SI (selected by default)
- SGE/SGA
- SPL

References: /FBSI/, Description of Functions Safety Integrated

Status SI

Diagnosis		CHAN1_MILL		Jog		\MPPF.DIR AA_JOL.MPF	
Channel reset				Program aborted			
				RDV			
Status SI							
AX1:X1 (DR1:SRM)							
Signal		NCK		Drive		Unit	
Safe actual position		100.073		100.073		mm	
Position difference NCK/drive		0.000		-		mm	
Monitoring "Safe operational stop" is active		No		No			
Monitoring "Safe velocity" is active		No		No			
Active SG level		None		None			
Active SG correction factor		None		-		%	
Safe actual velocity limit		Inactive		-		mm/min	
Set velocity limitation		Inactive		-		mm/min	
Current velocity difference		0.000		-		mm/min	
Max. velocity difference		0.000		-		mm/min	
Active safe software limit switch		1		1			
Active gear ration [stage]		1		1			
Active stop		None		None			
Currently requested external stop		None		None			
Stop-F code value (alarm 300911)		-		0			

Available signals/values

The vertical softkeys axis +, axis – or direct selection are used to set the desired axis. The current axis is displayed at the top right-hand side of the table.

Safe actual position
 Positional deviation NCK/drive
 "Safe operational stop" monitoring active
 "Safe velocity" monitoring active
 Active SG step
 Active SG correction factor
 Safe actual velocity limit
 Setpoint velocity limit

Current velocity difference
 Maximum velocity difference
 Active safe software limit switch
 Active gear ratio (step)
 Active stop
 Currently requested external stop
 Stop F code value (alarm 300911)
 Pulse enabled
 Traversing disable via stop in other axis

Sequence of operations

Select the menu headed "Service displays".
The horizontal softkey bars change.

The "Service SI" window displays information about Safety Integrated data together with axis name and axis number.

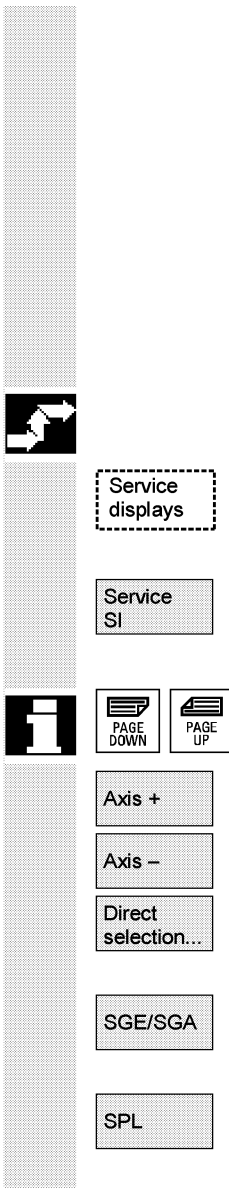
You can page up and down with the "Page keys".

The service values of the next (+) and the previous (-) axis are displayed.

An axis can be directly selected from the available axes.

This softkey is used to display safety relevant input and output signals.

This softkey is used to display safe programmable logic signals.



SGE/SGA

SGE/SGA		AX1: X1 (DR1:SRM)
SGE		
Safe input signals NCK bit 0...15		0000 0000 0000 1011
Safe input signals drive bit 0...15		0000 0000 0000 1011
Safe input signals NCK bit 16...31		1111 0000 0001 1100
Safe input signals drive bit 16...31		1111 0000 0001 1100
SGA		
Safe output signals NCK bit 0...15		0000 0000 1000 0100
Safe output signals drive bit 0...15		0000 0000 1000 0000
Safe output signals NCK bit 16...31		0000 0000 0010 0001
Safe output signals drive bit 16...31		0000 0000 0010 0000

The available signals are shown in the above display.

The vertical softkey status SI is the control key in the status display SI, while SPL is the control key in the display for safe programmable logic.

SPL

Variable	Bit	Current values	Format
\$A_INSE(P)	01...08	NCK	0000 0111
		PLC	0000 0111
\$A_OUTSE(P)	01...08	NCK	0000 0001
		PLC	0000 0001
\$A_INSI(P)	01...08	NCK	0000 0000
		PLC	0000 0000
\$A_OUTSI(P)	01...08	NCK	0011 1111
		PLC	0011 1111
\$A_MARKERSI(P)	01...08	NCK	1100 0011
		PLC	1100 0011

Signal	Value
X-ch fill level	0
Cross-checking status	No errors occurred
X-ch control word	KDV tolerance time 1s
SPL booting state	0000 0110 1111 1111

The following can be selected from the "Variable" dropdown lists:

\$A_INSE(P) corresponds to the simultaneous selection of

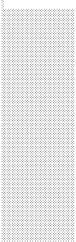
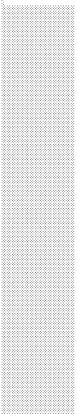
\$A_INSE top row; NCK origin

\$A_INSEP bottom row; PLC origin

The following should then be selected for the remaining variables:

	<p>\$A_OUTSE(P) \$A_INSI(P) \$A_OUTSI \$A_MARKERSI(P)</p>									
Storage	<p>Under "Bit" an 8-bit range can be requested for the selected signal. The variables selected and the assigned bit ranges are saved and taken into consideration when selecting the screen. In addition to the actual values, the origin of the displayed NCK/PLC signals is displayed.</p>									
Format	<p>The following formats can be selected in the variable rows using the</p>									
Pre-assigned format...	<table border="0"> <tr> <td>select key</td> <td>B</td> <td>Binary,</td> </tr> <tr> <td></td> <td>H</td> <td>Hexadecimal</td> </tr> <tr> <td></td> <td>D</td> <td>Decimal</td> </tr> </table> <p>The same formats can be selected in the submenu using the softkey "Pre-assigned format...". The selected format is applied to all variable displays in the screen. Individually set or fixed format settings are saved and taken into consideration in the following displays.</p>	select key	B	Binary,		H	Hexadecimal		D	Decimal
select key	B	Binary,								
	H	Hexadecimal								
	D	Decimal								
Selectable variables	<p>\$A_INSE(P) \$A_OUTSE(P) \$A_INSI(P) \$A_OUTSI(P) \$A_MARKERSI(P)</p>									
Displayed signals	<p>KDV level (KDV = cross-checking) KDV status KDV control word SPL ramp-up status SPL ramped up SPL interfaces are parameterized "SAFE.SPL" SPL program file loaded NCK waits until the PLC has started Interrupt for SPL start should be assigned Interrupt for SPL start has been assigned Interrupt processing for SPL start called Interrupt processing for SPL start exited NCK cross-checking has been started PLC cross-checking has been started Cyclic SPL checksum test active All SPL safety mechanisms active</p>									

8.3.5 Configuration data



Service displays

Config. data



Function

The configuration data of a machine (HMI version, NCU version, axis configuration, drive configuration, bus parameters, active bus nodes) can be written into a file and subsequently read out or printed.

Configuration data is output in 2 stages:

1. Creation of the configuration data file CONFIGURATION_DATA in the "Diagnosis" operating area by pressing the softkey "Config. data".
2. Reading out the file CONFIGURATION_DATA in the "Services" operating area via the RS-232 interface.

The configuration file CONFIGURATION_DATA is created in the "Services" operating area for this purpose.

Sequence of operations

Select the menu headed "Service displays".

The horizontal softkey bars change.

Press softkey "Config. data".

The system gathers the configuration data, writes it into the file CFGDAT.TXT and displays it.

The path and name of the file are output in the info line.

Additional notes

Configuration data can be read out via the RS-232 interface in the "Services" operating area.

8.3.6 Communication error log



Function

Errors that occur during communication between the HMI and NCK/PLC are registered in a communication error log.

You can display this log by pressing the  key.

The error log file is principally used by the control manufacturer (Siemens) as a diagnostic tool for communication errors.

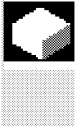
8.3.7 Action log



Action
log

An existing action log can be displayed. For additional information on configuring the contents of the action log, see Installation and Start-Up Guide HMI Advanced (IM4).

8.3.8 Version



Function

The version data of the installed system software are output in the Version display.



Sequence of operations

Service displays

Press softkey "Service displays".
The horizontal softkey bars change.

Version

Open the "Version" window in the "Service displays" menu.
The following softkeys are displayed:

NCU version

"NCU version" for the version data of the NCU and

MMC version

"MMC version" for the version data of the HMI.

Option for compile cycles

If "compile cycles" are available on the NCK, the version is displayed.

Cycle version

"Cycle version" Version data for each of the cycle packages (SW 6.3 and higher)

Definitions

"Definitions" The version data for definitions



Use the "Page" keys to scroll up and down.

8.3.9 Displaying the version screen for cycles (SW 6.3 and higher)



Function

The following version screens can be displayed and used to diagnose the cycle states in SW 6.3 and higher:

- Overview of the cycle packages available on the control.
- Details of the individual packages.
- Overview of all available cycles.
- Separate overview for user, manufacturer and standard cycles.
- Definitions.



Service displays

Version

Package overview

Cycle version

Package overview



Predefined package lists

Details of packages

Cycle version

Details

Sequence of operations

Press softkey "Service displays".
The horizontal softkey bars change.

Open the "Version" window in the "Service displays" menu.
The following softkeys are displayed:

The cycle packages available on the NCK can be displayed in a separate "Version data cycles" overview by pressing the horizontal softkey "Cycle version". The vertical softkey "Package overview" is included automatically.

Related cycles can be grouped together in packages and listed in a package list. The complete package is assigned a package name and a version identifier. The file type of the cycle package list is **.cyp** (cycle package). Package lists are named `cyc_xxx.cyp` and the following standard names are defined:

Package list	Cycle package
<code>cyc_sc.cyp</code>	Standard cycles
<code>cyc_scs.cyp</code>	Cycle support (for standard cycles)
<code>cyc_mc.cyp</code>	Measuring cycles
<code>cyc_mcs.cyp</code>	Measuring cycle support
<code>cyc_mj.cyp</code>	Measuring in JOG mode
<code>cyc_sm.cyp</code>	ShopMill
<code>cyc_st.cyp</code>	ShopTurn
<code>cyc_mt.cyp</code>	ManualTurn
<code>cyc_c950.cyp</code>	Extended stock removal
<code>cyc_c73.cyp</code>	Pocket surfaces with islands
<code>cyc_iso.cyp</code>	ISO compatibility cycles
<code>cyc_cma.cyp</code>	Manufacturer cycles (predefined name)
<code>cyc_cus.cyp</code>	User cycles (Predefined name)

Select a package in the package overview and press the vertical softkey "Details". The following details for the selected package appear in the "Version data" overview:

- Package name Name
- Package type Type
- Load status Loaded
- Package length Length
- Storage directory Directory (in data management)
- Version entry Version

Overview ofCycle
version**all cycles**All
cyclesCycle
version**User**User
cycles**Manufacturer**Manufac-
turer cycles**Standard
cycles**Standard
cycles**Definitions**Defi-
nitions

Press the vertical softkey "All cycles". The horizontal softkey "Cycle version" is included automatically.

All available cycles of the types .com and .spf from the user cycle (CUS.DIR), manufacturer cycle (CMA.DIR) and standard cycle (CST.DIR) directories are displayed in the "Version data" overview irrespective of the packages.

If a file is stored in several directories, the text color of the active version appears in black in the editor. Inactive files are grayed out.

Press the desired vertical softkey

- "User cycles" or
- "Manufacturer cycles" or
- "Standard cycles"

The horizontal softkey "Cycle version" remains active.

All files of the type .com and .spf from the user, manufacturer or standard cycle directory are displayed in the "Version data" overview without the package lists.

The definition files available on the NCK can be displayed in a separate "Version data definitions" overview by pressing the horizontal softkey "Definitions". All the definition files from the DEF.DIR directory in the data management then appear in this overview.

You can change to another version display by pressing a different horizontal softkey.

8.3.10 Exporting cycle versions (SW 6.3 and higher)



Function

You can save the contents of the version displays for cycles in a separate log file.



Sequence of operations

Service displays

Version

Cycle version

Save versions

To create a separate log file for the following version contents, press the "Save versions" softkey in the "Version data cycles" or "Definitions" version displays:

CYP.COM	Package overview
CYP_DET.COM	Details
ALLCYCLE.COM	All cycles
CUS.COM	User cycles
CMA.COM	Manufacturer cycles
CST.COM	Standard cycles
DEF.COM	Definitions

When the data have been saved, the following message appears:

"File stored!"

⇒ Services:\Diagnosis\Log files\Version\xxx.com.
xxx stands for the version contents.

8.3.11 Displaying loadable compile cycles (SW 6.3 and higher)



Function

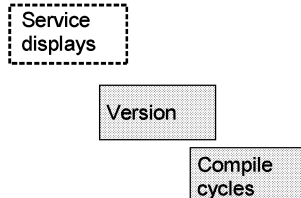
Loadable compile cycles (CC) are an optional extra. The functionality must be enabled explicitly. If loadable compile cycles are available on your NCK, you can display them in a separate "Version data cycles" display under Service displays/Version/Compile cycles.

Loadable compile cycles are available for the following software versions:

- NCK SW 6.3 and higher
- HMI Advanced SW 6.3 and higher



Loadable compile cycles



Sequence of operations

Compile cycles are loaded when you initiate an NCU reset and can then be displayed under Service displays/Version/Compile cycles.

Open the "Version" window in the "Service displays" menu.

The vertical softkey "Compile cycles" appears.

- Press the "Compile cycles" softkey.

Display loaded CC

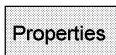
All loaded files of the type `.elf` are displayed in the "Version data compile cycles" overview.



You can check the storage location of the loadable compile cycles in the Services operating area in the main "Manage data" window. You can copy the available loadable compile cycles from here onto an installed NC card. Other storage locations, such as any external drive, e.g. disk drive or network drives 1 to 4, are supported.

Display non-loaded CC

All non-loaded files of the type `.elf` can also be displayed in the main "Manage data" window. To do this, select the appropriate directory. You can display all the main properties of an individual compile cycle by selecting any individual file with the extension `.elf`.



Press the "Properties" softkey

The horizontal and vertical softkey menu changes and a "Properties" window, such as the one below for NC card, appears.

Properties			
Path	:\NC card\Loadable compile cycles		
Name	:CCMCSC	Date:	Time
Extension	:ELF	Length:	Loaded: <input type="checkbox"/>
Type	:Loadable compile cycle		
Access rights			
Read:	Write:	Enable:	List: Delete:
Contents: Loadable compile cycle			
Version:	MCSC Coupling axes MCS	Time	Date
Advance version of compile cycle (Preliminary)			
Interface: 001.001@Interfaces=002.000 @TChain=001.000			
Current access level:			

Version display

8.4 PLC status

8.4.1 General



Function

You can obtain information about the current states of the following memory locations of the PLC and change them if necessary:

Inputs:	Input bit (Ix), input byte (Ibx) Input word (Iwx), input double word (Idx)
Outputs:	Output bit (Qx), output byte (Qbx) Output word (Qwx), output double word (Qdx)
Bit memories:	Memory bit (Mx), memory byte (MBx) Memory word (MWx), memory double word (MDx)
Timers:	Time (Tx)
Counters:	Counter (Cx)
Data:	Data block (DBx), data bit (DBxx), data byte (DBBx), data word (DBWx), data double word (DBDx)
Format:	B = binary H = hexadecimal D = decimal G = floating comma (for doublewords)

Operand	Example	Read	Write	Format	Value	Range
Inputs	I 2.0	yes	yes	B	0	0–127
	IB 2			B	0101 1010	
				H	5A	
				D	90	
Outputs	Q20.1	yes	yes	B	1	0–127
	QB 20			B	1101 0110	
				H	D6	
				D	214	
Bit memories	M 60.7	yes	yes	B	1	0–255
	MB 60			B	1101 0110	
				H	B8	
				D	180	
Timers	T20	yes	no	B H D		0–31

Operand	Example	Read	Write	Format	Value	Range
Counters	C20	yes	yes	B H D		0–31
Data block		yes	yes			0–255
Data byte	DB3.DBB9			H D B	A 10 000 0000 0000 1010	0–255



In HMI, the "Page" keys can be used to scroll up and down.

8.4.2 Changing/deleting values



Function

The values of operands can be changed.



Sequence of operations

The "Diagnosis" operating area is selected.

Diagnosis

PLC
status

Press softkey "PLC status".

The first operand screen form appears.

The vertical softkey bar changes.

Change

Cyclic updating of the values is interrupted.

Operand
+

Operand
-

You can increase or decrease the address of the operand by 1 place at a time.

Default
format...

Softkey assignment

A selection window appears.

You can preset the format fields to "B" (binary), "H" (hexadecimal), "D" (decimal) or "None".

DeleteUndo
changesAccepti
HELP

Change the operand, the format or the value.

Delete:

The entries for the selected operand (formats and values) are deleted. A query window is opened.

Undo changes:

Cyclic updating is continued; the entered values are not transferred to the PLC.

Accept:

The entered values are transferred to the PLC. Cyclic updating is continued.

Additional notes

Press the "Information key".

A description of the permissible input syntax for the PLC status display is overlaid.

8.5 Selecting/creating operand forms for PLC status



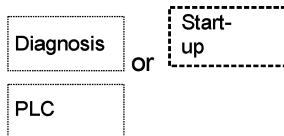
Function

You can save the operands entered in the "PLC status" window to a file or read in a back-up list of operands.



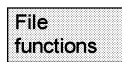
Sequence of operations

The operating area "Start-up" is selected.



Select softkey "PLC".

The horizontal and vertical softkey bars change.



Select softkey "File functions".

The vertical softkey bar changes.

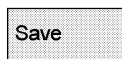
Enter the name of the file in which you wish to save the operands.

You can select existing back-up files from a list.

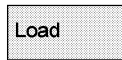
All the following functions refer to the file name entered:



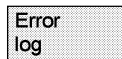
The selected operand back-up file is deleted.



The selected operands are saved to the specified file.



The selected operand file is loaded to the "PLC status" window for processing.



If errors occur during transfer of the machine data, these are entered in an error log.



Editor

The error log is transfer-specific, i.e. it is cleared before each new transfer.

The ASCII editor is called with the selected file.
You can now edit the operand back-up file.

8.5.1 File functions



Function

You can use the file functions to handle operand screens.



Diagnosis

PLC
status

File
functions

Delete

Save

Load

Sequence of operations

The "Diagnosis" operating area is selected.

Press softkey "PLC status".
The first operand screen form appears.
The vertical softkey bar changes.

Select softkey "File functions".
The "File functions" window opens.

Enter the file name of the desired operand screen form or
position the cursor on the desired operand screen form in the list.

The selected operand screen form is deleted.

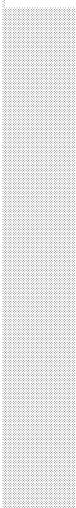
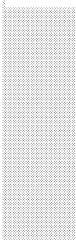
The current contents of the PLC status are saved in the selected operand screen form.

The contents of the selected operand screen form are loaded to the PLC status.

Additional notes

The operand screen forms are ASCII files.

8.6 Display NC system resources



Diagnosis

System
resources

NC

Stop

Start

Function

For the NC areas you can display the system resources (utilization display) currently being used:

Net and gross runtimes of

- position controller,
- interpolator and
- preprocessing.

Sequence of operations

The "Diagnosis" operating area is selected.

Press softkey "System resources".

The display "NC utilization" is displayed.

The following minimum/maximum total data for the servo, IPO cycle and preprocessing are displayed:

- Net runtime in ms
- Gross runtime in ms
- Level of the IPO buffer in percent
- Total capacity utilization in percent

The display update can be halted with the softkey "Stop", the displayed values are updated again with the softkey "Start".



"Start-Up" Operating Area

9.1	Machine data.....	9-436
9.1.1	Display options: Masking filter	9-438
9.2	User views.....	9-439
9.3	NC	9-441
9.4	PLC	9-441
9.4.1	PLC status	9-442
9.4.2	Setting the date/time.....	9-442
9.4.3	File functions.....	9-442
9.5	Drives/Servo.....	9-443
9.6	HMI.....	9-443
9.6.1	Changing the HMI interface	9-443
9.6.2	System settings	9-446
9.7	Tool management	9-450

Start-up basic display



Danger

Changes in the "Start-up" operating area have a major impact on the machine. Incorrect parameterization can endanger human life and cause damage to the machine.

Access to certain menus in the "Start-up" operating area can be protected by keyswitch or password.

This Chapter describes functions which the machine operator can perform on the basis of his or her access rights.

For information about start-up with respect to

- System personnel
- Machine manufacturers
- Service personnel
- Machine users (setup engineers).

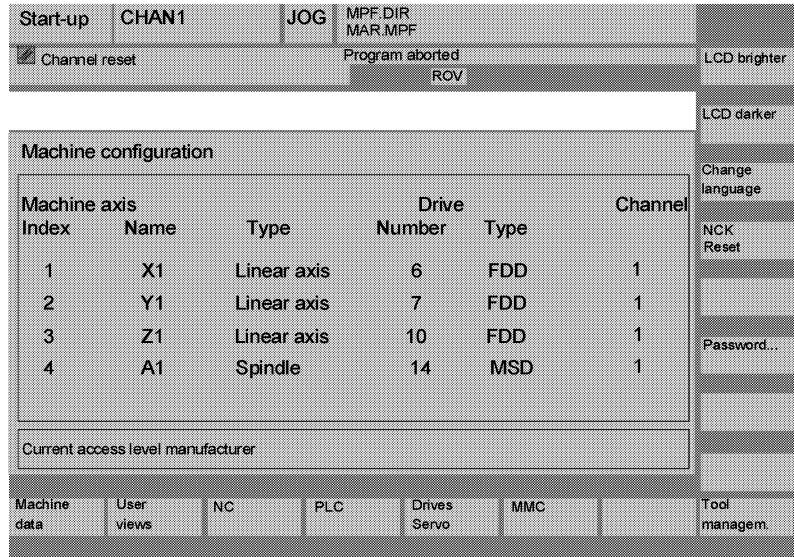
please refer to

/IAD/, Installation and Start-Up Guide, SINUMERIK 840D

/IAC/, Installation and Start-Up Guide, SINUMERIK 810D

/IAM/, IM4 Start-Up: HMI Advanced

The "Machine configuration" window is displayed in the "Start-up" basic display.



Machine data

User views

NC

PLC

Drives Servo

MMC

Tool management

Horizontal softkeys

Enables you to change the machine data for all areas.

You can create, display and modify your own views of machine data in any of the operating areas.

You can perform an NC boot in different modes.

The PLC status function is also available for updating the data and time of PLC and MMC 102/103.

You can update the date and time of the PLC and HMI.

This softkey provides access to special functions for starting up the drive/servo (e.g. function generator).

Here you can enter the basic settings for the operator panel front (e.g. color settings).

You can configure the tool magazine under this key.

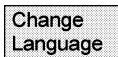


Vertical softkeys

Only for OP 010 with STN display:

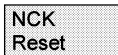
You can adjust the brightness of the screen.

You can define the boot settings in display machine data \$MM LCD CONTRAST. This display appears with each Power On boot.



You can use two languages in parallel.

When you press softkey "Change language", screen texts are displayed in the other of the two languages.



You can press this softkey to initiate NCK power ON/Reset. This softkey appears only if you have been granted appropriate access rights.



You can set, modify or delete a password.



Function

The control has a protection level system for enabling data areas. This system uses protection levels 0 to 7,

- 0 is the highest level and
- 7 the lowest.

Access to protection levels

- 0 to 3 is controlled by means of passwords and
- 4 to 7 by means of keyswitch settings.

The operator has access to information that is available on the level (and the levels below) to which he or she has access rights. Machine data are all assigned protection levels which vary depending on the nature of the data.

Protection level	Access controlled by	Range
0	Password	Siemens
1	Password	Machine manufacturer
2	Password	Start-up/service engineer
3	Password	End user
4	Keypad setting 3	Programming/set-up engineer
5	Keypad setting 2	Qualified operator
6	Keypad setting 1	Trained operator
7	Keypad setting 0	Job-trained operator

Personnel can edit data such as cycles and machine data depending on the level of authorization they have been granted.

You can alter the set password using the Password function.

If one of the above passwords is set, the keypad position is ignored.



9.1 Machine data



Danger

Changes in the machine data have a considerable influence on the machine. Incorrect parameterization can endanger human life and cause damage to the machine.



Access to the Machine data operating area can be controlled by keyswitch or password.



Areas

Function

Machine data are organized in the following areas:

- 1 General machine data (\$MN)
- 2 Channel-specific machine data (\$MC)
- 3 Axis-specific machine data (\$MA)
- 4 Feed drive machine data (\$MD)
- 5 Main spindle drive machine data (\$MD)
- 6 Display machine data (\$MM)

A separate list display in which you can view and change machine data is provided for each of these areas.

The following information about the machine data is displayed from left to right:

- Machine data number
- Machine data name (without area identifier \$MN, \$MC, \$MA, \$MD, \$MM), possibly with field index.
- Value of machine data
- Machine data unit
- Activation

If the machine data do not use units, no units are displayed.

If data are not available, the "#" symbol is displayed instead of the value.

If the value ends in an "H", it is a hexadecimal value.

The physical units of machine data are displayed on the right-hand side of the input field.

Examples:

m/s**2	m/s ² (meters/second squared): Acceleration
U/s**3	rev/s ³ (revolution/second to power of 3): Change in rate of acceleration for rotating axis
kg/m**2	kg/m ² (kilogram/meters squared): Moment of inertia
mH	mH (millihenry): Inductance
Nm	Nm (Newton meters): Torque
us	μs (microseconds): Time
uA	μA (microamperes): Unit of electric current
uVs	μVs (microvolt-seconds): Magnetic flux
userdef	User-defined: The unit is defined by the user.

The abbreviation in the right-hand column indicates the activation criterion for a machine data:

- so = active immediately
- cf = after confirmation via softkey "Set MD active"
- re = Reset
- po = POWER ON (NCK Power-On/Reset)

/IAD/, IAC/ Installation and Start-Up Guide 840D, 810D

Sequence of operations

The operating area "Start-up" is selected.

Press softkey "Machine data".

The horizontal and vertical softkey bars change.

You can select a machine data range, e.g. "General MD".



Start-up

Machine data

General MD

9.1.1 Display options: Masking filter



Function

The purpose of masking filters is to selectively reduce the number of displayed machine data. For this function, all machine data in areas

- General machine data
- Channel-specific machine data
- Axis-specific machine data
- Drive machine data

are assigned to specific groups (e.g. configuration data, etc.).

The following rules apply:

1. Each area has its own group organization.
2. Each group corresponds to one bit in the filter word ("spare" bit in previous SW)
3. Each area has a maximum of 13 groups (group 14 is reserved for Expert mode (see below), bit 15 is reserved for expansions).

Display machine data do not have any group organization.

Filter criteria

The following table shows the criteria for displaying machine data in the order in which they are evaluated:

Criterion	Check
1. Access rights	If the level of access authorization is not sufficient, the MD is not displayed. Otherwise criterion 2 is checked.
2. Masking filter active	The MD is always displayed when the filter is not active. Otherwise criterion 3 is checked.
3. Expert mode	The MD is not displayed if the expert mode bit is set and expert mode is not selected. Otherwise criterion 4 is checked.
4. Groups	If at least one group bit is both set and selected in the masking filter, criterion 6 is checked. Otherwise criterion 5 is checked.
5. All others	If none of the group bits is set and "All others" is selected in the masking filter, then criterion 6 is checked. If none of the group bits is set and "All others" is not selected in the masking filter, then the MD is not displayed.

6. Index from to	<p>If the index check is selected and the index of an array is within the chosen range, then the MD is displayed.</p> <p>If the index check is selected and the index of an array is not within the chosen range, then the MD is not displayed.</p> <p>If the index check is not selected, then the MD is displayed.</p>
------------------	--

Initialization

When you open a machine data window, the filter setting that matches the area is automatically updated.

Sequence of operations

The operating area "Start-up" is selected.

Press softkey "Machine data".

The horizontal and vertical softkey bars change.

Select the softkey "Display options", a list of all the ranges that can be displayed/hidden appears.



Start-up

Machine data

Display options

9.2 User views



Edit view

Function

User views are user-specific collections of machine data saved under one name. They are used to call all relevant machine data in a certain operating state if necessary from various areas to process in the display.

The vertical softkey bar changes.

In the "Edit view" menu, you can create your own user view and modify it if necessary.

Press softkey "Insert data". The vertical softkey bar changes.

The vertical softkey bar can be used to incorporate the following data into your user view:

- General machine data
- Channel-specific machine data
- Axis-specific machine data
- Feed spindle drive data
- Main spindle drive data

Position the cursor on the desired data. The selected data is incorporated in the user view.

Return to the "Edit view" window.

In this window you can label your user view. Entries in the "Text" field appear in the header, while entries in the "Description" field appear in the footer of your user view.

The entered text appears in the user view.

The currently selected row is deleted without confirmation.

Using the softkeys "Up" and "Down", it is possible to move the row on which the cursor is positioned one row up or down.

In this way it is possible to view and change the properties of the selected entry. In the "Designation" field you can change the name of the machine data.

In the "Description" field you can label your entries.

Only axis-specific machine data:

In the "Axis" field it is possible to enter the axis number or select the appropriate axis from a list. Axis numbers and axis names are marked with a "*" in the user view until their assignment changes.

The vertical softkey bar changes. In the "Manage views" menu, you can now work with your own user view. It is possible to assign your user views to six horizontal softkeys and call them at anytime. The currently active user view is always assigned.

Note

When you switch to "User views", the user view of the first assigned softkey is automatically displayed (usually softkey 1). If you do not enter a name in the "File" field in the "Assign softkey" menu, the currently active user view will also be deleted.

Manage
views

The view can only be assigned to the softkey if a name is entered.

The currently active user view is deleted.

You can save the currently active user view.

A specific user view can be loaded by entering the file name.

9.3 NC

Start-Up
key

Set a key position for the start-up key if you have appropriate access authorization.

NCK
address

View and edit the NCU/CCU address.

It is normally only necessary to edit the address for M : N operation.

An address modification cannot be undone by executing a general reset. It is not stored in an NC series start-up archive.

9.4 PLC



- You can only change PLC operands if you know the correct password.
- The procedure for handling PLC operands is described in subsection "PLC" in Chapter 8, "Diagnosis" Operating Area.



Danger

Changes in the states of PLC memory locations have a major impact on the machine. Incorrect parameterization can endanger human life and cause damage to the machine.

9.4.1 PLC status



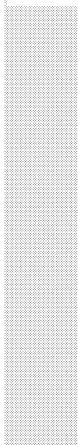
See Chapter 8, "PLC Status".

9.4.2 Setting the date/time



Function

You can change the date and time of the PLC and synchronize the date and time of the PLC and HMI.



Start-up

PLC

Set
date/clock

Accept

Sequence of operations

The operating area "Start-up" is selected.

Select softkey "PLC".

The horizontal and vertical softkey bars change.

Press the softkey "Set date/clock", the window "Set date/clock" is displayed.

Enter the correct values in the input fields.

The date and time on the HMI are transferred to the PLC.

You can check the synchronization in the output field "Current".

See /IAM/ IM4: Installation and Start-Up Guide HMI/MMC.

The set values are retained when the control is next powered up.

9.4.3 File functions



Enter the name of the file in which you wish to save the operands. You can select existing back-up files from a list.

9.5 Drives/Servo



SW 6.2 or later

This menu contains functions for

- Speed control loop
- Position control
- Circularity test
- Servo trace

Servo trace involves the recording of up to 10 bit signals during the integration period of bit-encoded Safety Integrated signals.

See: **/IAD/** Installation and Start-Up Guide
/FBA/ Description of Functions, Drive Functions
/FBSI/ Safety Integrated

9.6 HMI

9.6.1 Changing the HMI interface



Function

You can make individual settings on your HMI and store them.



Sequence of operations

Start-up

The operating area "Start-up" is selected.

MMC

Select softkey "MMC".

The horizontal and vertical softkey bars change.

Colors

The following **submenus** are available via the horizontal softkeys:

The "Colors" setting menu is opened.

You can either define the color scheme of your user interface yourself

- User



Save

or change it according to certain specifications

- VGA
- VGA positive
- Monochrome
- Monochrome positive.

The current color settings are saved.



Languages

The softkey offers the available system languages from which the first and second language can be selected. You can toggle between these languages with the "Change Language" softkey.



Operator panel

You can make the following settings in the "Operator panel front interface parameters" menu:

- Link
 - 1:1 (1NC and 1 HMI) or
 - m:n (1/several NCs and 1/several HMIs)
- Baud rate ("Bus")
 - MCP (1.5 Mbit/s)
 - MPI (187.5 Mbit/s)
- Highest bus address (15–31 available)
- Network address
 - HMI address (own address linked to bus)
 - NCK address (address used to establish communication link)
 - PLC address

NCK and PLC addresses can be changed only if you are using a 1:1 link. With m:n links, addresses are transferred from the "netnames.ini" file.

List of addresses of active nodes that can be activated with "Update".



Bus node



System settings

See 9.6.2 System settings



Select
printer

This softkey will function only if a printer is installed under Windows NT. It can be used to print displays and data from the Start-up operating area.

You select the printer on which you wish to print data/displays (default setting: Output as bitmap file)

For additional information, please see
/FBA/ Description of Functions, Drive Function.

Editor

This key opens the ASCII editor in which files can be edited at DOS level. You can select existing drives via vertical softkeys.

DOS
SHELL

You open a DOS shell.



Enter the "Exit" command to go back to the "Colors" menu.

9.6.2 System settings



System
settings

Function

This softkey provides access to settings for inquiry windows, file tree display and screen representation of the Machine, Program and Services operating areas.

File
display

You can set the file tree display for the Services, Machine and Program operating areas.

The following columns can be selected:

- Type (extension)
- Loaded
- Length
- Access protection
- Date
- Time
- Enable
- Max. display levels (branch to directory trees, a max. of 7)
- Max. name length (a max. of 25 characters)

Your settings are automatically displayed in the "Forecast" window.



Inquiry



Request confirmation before

- Deletion of data/programs,
- Deletion of directories,
- File overwrite.

Symbols

Here you can define whether keys must be represented as symbols or as text in HMI displays.

Example: Operator panel front in US layout,

e. g. the selection key as a symbol () as text (.

Templates

When creating a new workpiece, specify here whether templates for:

- job lists
- parts programs or
- initialization programs

are to be transferred to the new workpiece (directory).

The softkey provides a mask for parameterizing and activating the action log. The following settings can be made and saved:

- Logging on
- Alarm state change
- Keys (only ShopMill)
- Channel status/override
- Writing data
- File access
- Instructions (PI services program invocation)

Input fields are available for:

Write interval

File size, log file

Logged alarms

Action
log

Trace

In the event of communications errors, a trace log for communications processes can be recorded following an instruction to the Service department or to our hotline. The trace log is evaluated by the Siemens HMI Development department.



Sequence of operations

Start-up

The operating area "Start-up" is selected.

MMC

Select softkey "MMC".

The horizontal and vertical softkey bars change.

System
settings

Next select one of the desired functions:

Horizontal softkeys:

The "Settings for file display" window is opened.

File
display

Inquiry

The "Settings for inquiries" window is opened.

You can specify whether or not an inquiry window should be displayed after certain commands, e.g. Delete.

Symbols

The "Representation of keys in displays..." window is opened.

Templates

Use workpiece templates

Action
log

Log control processes

Sort...

Vertical softkey:

Sort the information into order according to a sorting criterion or sequence

A dialog box is displayed for setting the sorting sequence of **one** column.

The defined sequence applies for the display of the corresponding window in the operating areas: machine, program, services of the selected column.

You can select a sorting criterion:

- No sorting (by default the information is sorted in ascending order by column name) or
- One of the column designations

You can also select the sorting sequence:

- Decending
- Ascending

Confirm with OK.

The sorting sequence set is shown as an arrow symbol next to the column name of the column selected as the sorting criterion.

In HMI Advanced systems with an optional mouse, the following operating options are also available for column sorting:

1. A click on the column name with the arrow symbol changes the direction and sorts the information accordingly.
2. A click on another column makes this the sorting criterion.
With another click the direction can be changed, if necessary, according to (1.).

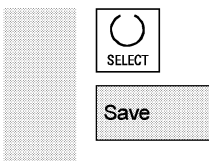
The selection of another sorting criterion in the operating area changes the sorting criterion for all operating areas (machine, program, services).

In the operating areas



Additional notes

When the sorting sequence is set using the softkey "Sort", if the sorting criterion is not available in the display image of the operating area, the information is sorted in ascending order according to the column name. The sorting sequence defined in the dialog box applies to the operating areas, in which the criterion set in the dialog box appears.



Position the cursor on the desired point and perform the settings.

Transfers your settings to system.

9.7 Tool management**Function**

/FBA/ Description of Functions, Tool Management



Maintenance



10.1 Operating data.....	10-452
10.2 Cleaning	10-453

10.1 Operating data

Operating data

	Value
Air humidity, humidity class to DIN 40040	F
Air pressure	860 to 1080 hPa
Shock protection, Protection class to DIN VDE 0160	I
Degree of protection according to DIN 40050	
• Front of operator panel front	IP 54
• Rear of operator panel front	IP 00
• Front of machine control panel	IP 54
• Back of machine control panel	IP 00

You will find the comprehensive operating data in the documentation /BH/ Operator Components Manual and in the relevant information sheets.

10.2 Cleaning

Cleaning agents

The front of the monitor and the surface of the operator panel front can be cleaned. For dirt that is relatively easy to remove, standard household washing-up liquid, or an industrial cleaner (such as "Special Swipe") can be used. These cleaners will also remove dirt containing graphite.

Cleansing agents which contain one or more of the following ingredients can be used for a short period of time:

- Diluted mineral acids
- Bases
- Organic hydrocarbons
- Detergent solutions

Plastic material used

The plastic material used on the front of the OP015, OP012 or OP015 is suitable for use on machine tools.

This is resistant to

1. Greases, oils, mineral oils
2. Bases and lyes
3. Detergent solutions and
4. Alcohol

Solvents such as chlorinated hydrocarbons, benzene, esters and ethers should be avoided!

Appendix

A Abbreviations

AS	Automation System
ASCII	American Standard Code for Information Interchange
ASUB	Asynchronous subprogram
BCD	Binary Coded Decimals
BCS	Basic Coordinate System
BIN	Binary files
BIOS	Basic Input Output System
C1 ... C4	Channel 1 to Channel 4
CAD	Computer-Aided Design
CAM	Computer-Aided Manufacturing
CNC	Computerized Numerical Control
COM	Communication
CP	Communications Processor
CPU	Central Processing Unit
CR	Carriage Return
CRC	Cutter Radius Compensation
CSF	Control System Flowchart
CTS	Clear To Send: (serial data interfaces)
CUTOM	Cutter radius compensation (tool radius compensation)

DAC	Digital-to-Analog Converter
DB	Data Block in the PLC
DBB	Data Block Byte in the PLC
DBW	Data Block Word in the PLC
DBX	Data Block Bit in the PLC
DC	Direct Control: The rotary axis is moved along the shortest path to the absolute position within one revolution.
DIN	Deutsche Industrie Norm (German Industry Standard)
DIO	Data Input/Output: Data transfer display
DIR	Directory
DOS	Disk Operating System
DPM	Dual-Port Memory
DPR	Dual-Port RAM
DRAM	Dynamic Random Access Memory
DRF	Differential Resolver Function: (handwheel)
DRY	Dry Run
DSB	Decoding Single Block
DW	Data Word
EIA Code	Special tape format: Number of perforations per character is always odd
ENC	Encoder: actual-value sensor
EPROM	Erasable Programmable Read Only Memory
FB	Function Block

FC	Function Call: Function block in PLC
FDD	Feed Drive
FEPROM	Flash EPROM Read/write memory
FIFO	First-In-First-Out: Memory which operates without address specification from which data are read in the same order as they are stored.
FIPO	Fine Interpolator
FPU	Floating Point Unit
FRA	Frame Block
FRAME	Data block (frame)
FST	Feed Stop
GUD	Global User Data
HD	Hard Disk
HEX	Abbreviation for hexadecimal
HHU	Hand-Held Unit
HMI	Human Machine Interface: SINUMERIK operator functionality for operation, programming and simulation. HMI has the same meaning as MMC
HMS	High-Resolution Measuring System
HW	Hardware
I	Input
I/O	Input/Output
I/RF	Infeed/Regenerative Feedback Unit (power supply) of SIMODRIVE 611(D)

IKA	Interpolative Compensation
IM	Interface Module
INC	Increment
INI	Initializing Data
IPO	Interpolator
IS	Interface signal
ISO Code	Special tape code, number of perforations per character is always even
JOG	Jog mode: Setup mode
K_{UE}	Transmission ratio
K_v	Servo gain factor
LAD	Ladder diagram (programming method for PLC)
LEC	Leadscrew Error Compensation
LF	Line Feed
LUD	Local User Data
MCP	Machine Control Panel
MCS	Machine Coordinate System (Machine)
MD	Machine Data
MDI	Manual Data Input
MMC	Man Machine Communication: SINUMERIK operator functionality for operation, programming and simulation. MMC has the same meaning as HMI.
MPF	Main Program File: NC parts program

MPI	Multi Point Interface
MSD	Main Spindle Drive
NC	Numerical Control
NCK	Numerical Control Kernel (with block preparation, traversing range, etc.)
NCU	Numerical Control Unit: Hardware unit of the NCK
NURBS	Non-Uniform Rational B-Spline
O	Output
OB	Organization Block in PLC
OEM	Original Equipment Manufacturer
OI	Operator Interface
OP	Operator Panel
OPI	Operator Panel Interface
OPT	Options
PCIN	Name of the software for data communication with the control
PCMCIA	Personal Computer Memory Card International Association: Memory card standardization
PG	Programming Device
PLC	Programmable Logic Controller
PMS	Position Measuring System
RAM	Random Access Memory (read-write memory)
REF	Reference point approach function
REPOS	Reposition function

ROV	Rapid Override
RPA	R Parameter Active: Memory area in NCK for R parameter numbers
RPY	Roll Pitch Yaw: Type of rotation of a coordinate system
RS-232	Serial interface (US standard), defines transmission of serial data between DTE and DCE devices
RTS	Request To Send (serial data interfaces)
SBL	Single Block
SD	Setting Data
SEA	Setting Data Active: File identifier for setting data
SK	Soft key
SKP	Skip block
SPF	Subprogram File
SRAM	Static RAM (battery-backed)
SSI	Serial Synchronous Interface
STL	Statement List
SW limit switch	Software limit switch
SYF	System Files
TEA	Testing Data Active: Identifier for machine data
TLC	Tool Length Compensation
TNRC	Tool Nose Radius Compensation
TO	Tool Offset
TOA	Tool Offset Active: Identifier (data type) for tool offsets

TRANSMIT	Transform Milling into Turning: Coordinate conversion on turning machines for milling operations
TRC	Tool Radius Compensation
UFR	User Frame: Work offset
WCS	Workpiece Coordinate System (Work)
WO	Work offset (identical with ZO – zero offset)
WOA	Work Offset Active: Identifier (file type) for work offset data
WOP	Workshop-Oriented Programming
WPD	Workpiece Directory
ZO	Zero offset (identical with WO – work offset)
ZOA	Zero Offset Active: Identifier (file type) for work offset data

B Terms

Important terms are listed below in alphabetical order, accompanied by explanations. Cross-references to other entries in this glossary are indicated by the symbol ->.

A**Absolute dimension**

A destination for an axis movement is defined by a dimension that refers to the origin of the currently active coordinate system. See also -> incremental dimension.

Acceleration with jerk limitation

To obtain the optimum acceleration gradient for the machine while at the same time minimizing mechanical wear and tear, the machining program offers a choice between instantaneous acceleration and continuous (smooth) acceleration.

Access rights

Programs and other files are protected by a 7-level system of access restrictions:

- Three password levels for system manufacturer, machine manufacturer and user, and

Four keyswitch positions that can be analyzed by the PLC (depending on keyswitch HW).

Alarms

1. All -> messages and alarms are displayed on the control panel in plain-text form and accompanied by date and time and the symbol for the appropriate deletion criterion. Alarms and messages are displayed separately.

Analog I/O module

Analog input/output modules are signal transducers for analog process signals.

Analog input modules convert analog measured signals into digital values that can be processed in the CPU.

Analog output modules convert digital values into analog manipulated variables.

Approach machine fixed-point

Approach to a predefined -> machine fixed point.

Archiving	Exporting files and/or directories to an external storage device.
A spline	The Akima spline progresses tangentially through the programmed interpolation points (3rd degree polynomial).
Asynchronous subprogram	Parts program that can be triggered asynchronously to (irrespective of) the current program status by means of an interrupt signal (e.g. "high-speed NC input" signal).
Automatic	Control operating mode (block-sequential operation to DIN): Operating mode of NC systems in which a -> parts program is selected and then processed without interruption.
Auxiliary functions	Auxiliary functions can be used to pass -> parameters to the -> PLC in the -> parts program, triggering reactions there which are defined by the machine manufacturer.
Axes	CNC axes are categorized by their functional scope as follows: <ul style="list-style-type: none">• Axes Interpolative path axes• Auxiliary axes: Non-interpolative infeed and positioning axes with axis-specific feedrates. Auxiliary axes do not participate in workpiece machining as such and include tool feeders, tool magazines, etc.
Axis address	See -> axis identifier
Axis identifier	In compliance with DIN 66217, axes are identified as X, Y, Z for a right-handed rectangular Cartesian -> coordinate system rotating in the clockwise direction. Rotary axes rotating around X, Y and Z are assigned the identifiers A, B and C. Additional axes, which are parallel to those specified, can be identified by other letters.
Axis name	See -> axis identifier

B**B spline**

The programmed B spline positions are not intermediate points but simply "checkpoints". Instead of passing directly through these checkpoints, the curve merely passes in their vicinity (1st, 2nd or 3rd degree polynomials).

Base axis

Axis whose setpoint or actual value is employed in calculating a compensatory value.

Basic coordinate system

Cartesian system of coordinates, mapped onto the machine coordinate system by transformation.

In the -> parts program, the programmer uses the axis names of the basic coordinate system. The basic coordinate system exists in parallel to the machine coordinate system when no -> transformation is active. The difference between the systems relates only to the axis identifiers.

Blank

The unmachined workpiece.

Block

All files required for programming and program execution are known as blocks.

Block

A section of a -> parts program terminated with a line feed is also called a block. There are two types of block, i.e. -> main blocks and -> subblocks.

Block search

For testing parts programs and after an interruption in machining, the block search function can be used to select a point in the parts program at which machining must be started or resumed.

Boot

Loading of the system program after Power ON.

C**C axis**

Axis about which the tool spindle describes a controlled rotational and positioning movement.

Channel

A channel can execute a -> parts program independently of other channels. A channel has exclusive control over the axes and spindles assigned to it. Parts program sequences on different channels can be coordinated by -> synchronization.

Channel structure	On account of the channel structure, it is possible to execute the -> programs of the individual channels simultaneously and asynchronously. See also - machining channel.
Circular interpolation	The -> tool is required to travel in a circle between defined points on the contour with a specified feed while machining the workpiece.
CNC	-> NC
COM	Numerical control component for the implementation and coordination of communication.
Compensation axis	Axis having a setpoint or actual value modified by a compensation value.
Compensation table	Table of intermediate (interpolation) points. This table supplies the compensation values of the compensation axis for selected positions of the base axis.
Compensation value	Difference between the axis position measured by the position sensor and the desired, programmed axis position.
Continuous-path mode	The purpose of continuous-path control mode is to prevent excessive deceleration of -> path axes at parts program block limits and to ensure the smoothest possible transition to the next block.
Contour	Outline of a -> workpiece
Contour monitoring	The following error is monitored within a definable tolerance bandwidth as a measure of contour precision. The following error might violate permissible limits, for example, on account of drive overload. In this case, an alarm is output and the axes are stopped.
Coordinate system	See -> machine coordinate system, -> workpiece coordinate system
CPU	Central processing unit of a -> programmable logic controller
Cycle	Subprogram for execution of a recurring machining process on the -> workpiece.

Cycle support

Support displays for technology cycles are provided in the editor in the "Program" operating area. Once the desired machining cycle has been selected, the parameters required to assign values are displayed in plain-text form.

See also -> Standard cycles.

D**D number**

Number for the tool offset memory

Data block

1. Data unit of the -> PLC, accessible by -> HIGHSTEP programs.
2. Data unit of the -> NC: Data blocks contain data definitions for global user data. The data can be initialized directly on definition.

**Data transfer program
PCIN**

PCIN is a routine for sending and receiving CNC user data via the serial interface. Typical data include parts programs, tool compensation data, etc. The PCIN program is executable under MS-DOS on industry-standard PCs.

Data word

A data unit, two bytes in size, within a -> data block.

**Dimensions in metric
and inch systems**

In the machining program, position and lead/pitch values can be entered in inches. The control is set to a base system irrespective of the programmable unit of measure (G70/G71).

DRF

Differential Resolver Function: An NC function which generates an incremental work offset in AUTOMATIC mode in conjunction with an electronic handwheel.

Drift compensation

When the CNC axes are in the constant motion phase, automatic drift compensation is implemented in the analog speed control. (SINUMERIK FM-NC).

Drive

- SINUMERIK FM-NC has an analog ± 10 V interface to the SIMODRIVE 611A converter system.
- The SINUMERIK 840D control system is connected to the SIMODRIVE 611D converter system by a high-speed digital parallel bus.

E**Editor**

The editor allows programs/texts/program blocks to be created, modified, extended, chained and inserted.

Electronic handwheel

Electronic handwheels are used to traverse selected axes simultaneously under manual control. The handwheel clicks are analyzed by the increment analyzer.

Exact stop

When an exact stop is programmed, a position specified in the block is approached accurately and, where appropriate, very slowly. In order to reduce the approach time, -> exact stop limits are defined for rapid traverse and feed.

Exact stop limit

When all path axes reach their exact stop limits, the control responds as if it had reached its destination point precisely. The -> parts program carries on execution with the next block.

External work offset

Work offset specified by the -> PLC

F**Feedrate override**

The current feedrate setting entered via the control panel or by the PLC is overlaid on the programmed feedrate (0-200 %). The feedrate can also be corrected by a programmable percentage factor (1-200 %) in the machining program.

File type

Possible types of files include parts programs, work offsets, R variables, etc.

Finished-part contour

Contour of the finished workpiece. See also -> Blank.

Fixed-point approach

Machine tools can execute defined approaches to fixed points such as tool-change points, loading points, pallet-change points, etc. The coordinates of these points are stored in the control. The control traverses the axes in question in -> rapid traverse if possible.

Focus

Border (bold frame) which identifies windows that can be edited.

Frame

A frame is a calculation rule that translates one Cartesian coordinate system into another Cartesian coordinate system. A frame contains the components -> work offset, -> rotation, -> scaling, -> mirroring.

G**Geometry**

Description of a -> workpiece in the -> workpiece coordinate system.

Geometry axis

Geometry axes are used to describe a 2- or 3-dimensional area in the workpiece coordinate system.

H**High-level language CNC**

The high-level language supports: -> user variables, -> predefined user variables, -> system variables, -> indirect programming, -> computation and angle functions, -> comparisons and logical gating, -> program jumps and program branches, -> program coordination (SINUMERIK 840D), -> macro programming.

High-speed digital inputs/outputs

Digital inputs can be used, for example, to start high-speed CNC program routines (interrupt routines). The digital CNC outputs can be used to trigger high-speed, program-driven switching functions (SINUMERIK 840D).

I**I/O module**

I/O modules establish the link between the CPU and the process. I/O modules are:

- -> Digital input/output modules
- -> Analog input/output modules
- -> Simulator modules

Identifier

In accordance with DIN 66025, identifiers (names) for variables (calculation variables, system variables, user variables), subprograms, vocabulary words and words can contain several address letters. These letters have the same meaning as the words in the block syntax. Identifiers must be unique. The same identifier may not be used for different objects.

Inch system of measurement

System of measurement in which distances are measured in inches and fractions of inches.

Incremental dimension

Traversing length calculated from the number of increments x increment length. The number of increments can be stored as a -> setting data or selected with keys labeled for 10, 100, 1000, 10 000.

Initialization file

Initialization files are special -> program blocks. They contain value assignments that must be implemented before program execution commences.

Initialization files are used primarily to initialize predefined data or global user data.

An initialization file can be created for each -> workpiece. In it, the various variable value instructions which apply exclusively to one workpiece can be stored.

Interpolative compensation

Interpolative compensation provides a means of compensating leadscrew errors and measuring system errors resulting from the production process (LEC, MSEC).

Interpolator

Logical unit of the -> NCK which determines intermediate values of the movements to be traversed by the individual axes on the basis of destination positions specified in the parts program.

J**Jog**

Control mode (setup): The machine can be set up in Jog mode. Individual axes and spindles can be jogged by means of manually operated momentary-contact switches. Other functions in the Jog mode are
-> reference point approach, -> Repos and -> Preset (act actual value).

K**Keyswitch**

The keyswitch on the -> machine control panel has 4 positions, each of which is assigned certain functions by the operating system of the control. The keyswitch has three keys of different colors; a key can be removed when in the designated position.

Keywords

Words of a defined notation and having a defined meaning in the programming language for -> parts programs.

K_{UE}

Transmission ratio

K_v

Servo gain factor, control variable of a control loop

L

Languages

The user interface and system messages and alarms are available in five system languages (on disk):

English, French, German, Italian and Spanish.

Any **two** of the above languages are installed and selectable in the control (Start-up operating area).

Leadscrew error compensation

Compensation for the mechanical inaccuracies of a leadscrew participating in the feed. The control uses stored deviation values for the compensation.

Limit speed

Maximum/minimum (spindle) speed: The maximum speed of a spindle can be limited by values defined in the machine data, the -> PLC or -> setting data.

Linear axis

The linear axis is an axis which, in contrast to a rotary axis, describes a straight line.

Linear interpolation

The tool is required to travel to the destination point along a straight line while machining the workpiece.

Look Ahead

The **Look Ahead** function is a means of optimizing the machining speed by looking ahead over a parameterizable number of traversing blocks.

M

Machine axes

Axes which physically exist in the machine tool.

Machine control panel

An operator panel on a -> machine tool with operating elements such as keys, rotary switches, etc. and indicators such as LEDs. It is used for direct control of the machine tool via the PLC.

Machine coordinate system

The machine coordinate system (Machine - MCS) refers to the coordinates of the machine axes, i.e. all machine axes and auxiliary axes are displayed in the machine coordinate system.

Machine fixed point

A point uniquely defined by the machine tool, for example, the reference point.

Machine zero

A fixed point on the machine tool which can be referenced by all derived measuring systems.

Machining channel	A channel structure provides a means of reducing non-productive times by paralleling operations. For example, a loader can execute its movements during a machining operation. In this respect, a channel ranks as an autonomous CNC complete with decoding, block preparation and interpolation.
Macro programming	A collection of instructions under a common identifier. The identifier in the program represents the quantity of collected instructions.
Main block	A block prefixed by ":" containing all the parameters required to start execution of a -> parts program.
Main memory	The working memory is a RAM in the -> CPU which the processor accesses as it executes the user program.
Main program	A -> parts program identified by a number or a name in which further main programs, subprograms or -> cycles can be called.
MDI	Control operating mode: Manual Data Input: In the MDI mode, individual program blocks or block sequences can be entered without reference to a main program or routine. They can then be executed immediately with the NC start key.
Messages	All messages programmed in the parts program and -> alarms detected by the system are displayed on the control panel in plain-text form with date and time and the appropriate symbol for the deletion criterion. Alarms and messages are displayed separately.
Metric measuring system	Standardized system of units: The units of measure for length, for example, are mm (millimeter) and m (meter).
Mirroring	Mirroring exchanges the leading signs of the coordinate values of a contour in relation to an axis. Mirroring can be performed simultaneously in relation to several axes.
Mode group	Technologically related axes and spindles can be combined in a mode group. Axes/spindles of the same mode group can be controlled by one or more -> channels. The same -> mode is always assigned to the channels of a mode group.

Multipoint interface (MPI)	<p>The multipoint interface (MPI) is a 9-pin D-Sub port. A parameterizable number of devices can be connected to a multipoint interface for the purpose of communicating with one another:</p> <ul style="list-style-type: none">• Programming devices• MMI systems• Other automation systems <p>The "Multipoint Interface MPI" parameter block of the CPU contains the -> parameters which define the properties of the MPI.</p>
N	
NC	<p>Numerical Control: NC control incorporates all the components of the machine tool control system: -> NCK, -> PLC, -> MMC, -> COM. Note: CNC (computerized numerical control) is a more accurate term for the SINUMERIK 840D and FM-NC controls.</p>
NCK	<p>Numeric Control Kernel: Component of the NC control which executes -> parts programs and essentially coordinates the movements on the machine.</p>
Node number	<p>The node number is the address of a -> CPU or the -> programming device or some other intelligent I/O module for communication via a -> network. The node number is assigned to the CPU or the programming device by the S7 tool -> "S7 configuration".</p>
NRK	<p>Numeric Robotic Kernel (operating system of the -> NCK)</p>
NURBS	<p>Within the NC, motion control and path interpolation are based on NURBS (Non-Uniform Rational B-Splines). This is available as a uniform procedure for all interpolation activities of the control (SINUMERIK 840D).</p>
O	
OEM	<p>The manufacturers of machine tools who prefer to generate their own user interfaces or incorporate customized, technology-related functions in the control have plenty of scope to do so (OEM applications) with the SINUMERIK 840D.</p>
Offset memory	<p>Data area in the control used to store the tool offset data.</p>

Operating area	The basic functions of the control are organized in separate operating areas.
Operating mode	An operating concept on a SINUMERIK control. The modes -> Jog, -> MDI and -> Automatic are defined.
Oriented spindle stop	Stops the workpiece spindle at a specified orientation angle, e.g. to perform an additional machining operation at a specific position.
Oriented tool retractions	RETTOOL: If machining is interrupted (e.g. by tool breakage), a program command can be issued that causes the tool to be retracted a defined distance and at a specific orientation angle.
Override	Manual or programmable control feature which enables the operator to overlay programmed feedrates or speeds to adapt them to a specific workpiece or material.
P	
Parameters	<ol style="list-style-type: none">S7-300: This system has 2 types of parameter:<ul style="list-style-type: none">Parameter of a STEP 7 statement A parameter of a STEP 7 statement is the address of the operand to be processed or a constant.Parameter of a -> parameter block A parameter of a parameter block determines the behavior of a module.840D/FM-NC:<ul style="list-style-type: none">Computation parameter, can be set any number of times or interrogated by the programmer for any purpose in the parts program.
Parts program	A sequence of instructions to the NC control which combine to produce a specific -> workpiece by performing machining operations on a -> blank. Also refers to a specific machining operation on a given -> blank.
Path axis	Path axes are all the machining axes of the -> channel which are controlled by the -> interpolator such that they start, stop, accelerate and reach their end points simultaneously.
Path feed	Path feed is applied to -> path axes. It is the geometric sum of feeds of the participating -> geometry axes.

Path velocity	The maximum programmable path velocity depends on the input resolution. If the resolution is 0.1 mm, for example, the maximum programmable path speed is 1000 mm/min.
PLC	Programmable Logic Controller: A component of the -> NC control. A controller which can be programmed to control the logic on a machine tool.
PLC program memory	<ul style="list-style-type: none">• SINUMERIK FM-NC: The PLC user memory of the CPU 314 is used to store the PLC application program and the user data together with the PLC basic program. The S7-CPU 314 has a user memory of 24 KB for this purpose.• SINUMERIK 840D: The PLC user memory is used to store the PLC application program and the user data together with the PLC basic memory. The PLC user memory can be expanded to 96 KB through the insertion of expansion modules.
Polar coordinates	A coordinate system which defines the position of a point on a plane in terms of its distance from the origin and the angle formed by the radius vector with a defined axis.
Positioning axis	An axis which performs an auxiliary movement on a machine tool (e.g. tool magazine, pallet transport). Positioning axes are axes that do not interpolate with the -> path axes.
Preset	The Preset function is a means of redefining the control zero in the machine coordinate system. Preset does not trigger movement of the axes. Instead, a new position value is entered for the current axis positions.
Program control	This function can be used in the Automatic and MDI modes to control program execution (e.g. through selection of a block to be skipped).
Programmable frames	Programmable -> frames can be used to define new coordinate system starting points dynamically during execution of the parts program. Two types of definition are used, i.e. "absolute" which uses a new frame and "additive" which uses a reference to an existing starting point.
Programmable working area limitation	Limitation of the movement area of the tool within programmed limits.

Programming key	Characters and character sequences which have a defined meaning in the programming language for -> parts programs (see Programming Guide).
Programming language CNC	The CNC programming language is based on DIN 66025 with high-level language extensions. The CNC programming language and -> high-level language extensions support the definition of macros (sequenced statements).
Protection zone	Three-dimensional zone within the -> working area and which the tool tip is not allowed to enter.
Q	
Quadrant error compensation	Quadrant error compensation is a means of largely eliminating contour errors at quadrant transitions caused by the variation in friction at guideways. Quadrant error compensation is parameterized by means of a circularity test.
R	
R variable	Arithmetic parameter. The programmer of the -> parts program can assign or request the values of the R variables as required.
Rapid lift from contour	If an interrupt is received, the CNC machining program can trigger a movement which permits the tool to lift rapidly away from the workpiece contour currently being machined. The angle of retraction and the distance can also be parameterized. The rapid lift from contour can be followed by an interrupt routine (SINUMERIK FM-NC, 840D).
Rapid traverse	Highest speed of an axis, used, for example, to bring the tool from an idle position to the -> workpiece contour or retract it from the workpiece contour.
Reference point	A point on the machine tool to which the measurement system of the -> machine axes refers.
Reference point approach	If the system of position measuring employed is not based on absolute-value encoders, the control must perform a reference point approach in order to ensure that the measured values supplied by the measuring system coincide with the machine coordinate values.

REPOS

1. Reapproach to contour, triggered by operator
The Repos function provides a means of returning the tool to the interrupt position with the aid of the direction keys.
2. Program-driven return to contour
A number of repositioning strategies driven by program commands are available: Reposition at interrupt point, reposition at start of block, reposition at end of block, reposition at a point on the path between beginning of block and interruption.

Rigid tapping

This function is used to tap holes without the use of a compensating chuck. Through interpolative traversal of the spindle as a rotary axis and the drill axis, threads are tapped precisely to the final drilling depth, for example, in blind tapped holes (precondition: spindle is operating in axis mode).

Rotary axis

Rotary axes cause the tool or workpiece to rotate to a specified angle position.

**Rotary axis,
continuously turning**

The range of motion of a rotary axis can be limited to an angle less than 360 degrees or defined as continuous in both directions, depending on the application. Continuously turning rotary axes are used, for example, for eccentric machining operations, grinding and winding.

Rotation

Component of a -> frame which defines a rotation of the coordinate system through a specific angle.

Rounding axis

Rounding axes cause the workpiece or tool to rotate to an angle position described on a graduated grid. When the grid position has been reached, the axis is "in position".

S**S7 Configuration**

S7 Configuration is a tool for parameterizing modules. S7 Configuration is used to set a variety of
-> parameter blocks of the -> CPU and the I/O modules in the
-> programming device. These parameters are uploaded to the CPU.

S7-300 bus

The S7-300 bus is a serial data bus via which modules communicate and receive their supply voltage. The connections between the modules are established by means of -> bus connectors.

Safety functions	The control incorporates monitors which are active at all times and which are designed to detect malfunctions in the -> CNC, the programmable controller (-> PLC) and the machine at an early stage in order to minimize the risk of damage to the tool, workpiece or machine. If a malfunction occurs, machining is interrupted and the drives are stopped. The cause of the malfunction is logged and an alarm is issued. At the same time, the PLC is notified that a CNC alarm has been triggered.
Scaling	Component of a -> frame which causes axis-specific alterations in the scale.
Setting data	Data which provide the NC with information on the properties of the machine tool in a way defined by the software.
Softkey	A key whose name appears on an area of the screen. The choice of softkeys displayed is adapted dynamically to the operating situation. Freely assignable function keys (softkeys) are assigned to functions defined in the software.
Software limit switch	Software limit switches define the limits of the travel range of an axis and prevent the slide contacting the hardware limit switches. Two pairs of values can be assigned per axis and activated separately via the -> PLC.
Spindles	Spindles are axes, which rotate tools or workpieces during machining.
Spline interpolation	Spline interpolation is a method by which the control can construct a smooth curve from a limited number of intermediate points defined on a target contour.
Standard cycles	Standard cycles are provided for frequently recurring machining processes: <ul style="list-style-type: none">• for drilling and milling• for turning (SINUMERIK FM-NC) The cycles available can be viewed in a list called through the "Cycle support" menu in the "Program" operating area. Once the desired machining cycle has been selected, the parameters required for value assignment are displayed in plain text and values can be entered.
Subblock	Block introduced by "N" and containing information for a step, e.g. the definition of a position.

Subprogram	Sequence of statements in a -> parts program that can be called repeatedly with differing initial parameters. -> Cycles are a type of subprogram.
Synchronization	Instructions in -> parts programs for coordination of the operations in different -> channels at specific machining points.
Synchronized actions	<ol style="list-style-type: none">1. Auxiliary function output When machining is in progress, technological functions (-> auxiliary functions) can be output to the PLC from within the CNC program. These auxiliary programs are used, for example, to control auxiliaries for the machine tool such as center sleeves, grippers, chucks, etc.2. High-speed auxiliary function output For time-critical switching functions, the acknowledgment times for the -> auxiliary functions can be minimized and unnecessary stops in the machining process avoided.
Synchronized axes	Synchronized axes require the same time for their travel as -> geometry axes for their path travel.
System variable	A variable which exists although it has not been programmed by the programmer of the -> parts program. It is defined by the data type and the variable name, which is prefixed with \$. See also -> User-defined variable.
T	
Teach In	Teach In is a means of creating or correcting parts programs. The individual program blocks are keyed in via the keyboard and executed immediately. Positions approached via the direction keys or handwheels can also be stored. Additional specifications such as G functions, feedrates and M functions can be entered in the same block.
Text editor	-> Editor
Tool	Component used to machine workpieces, e.g. turning tool, milling tool, drill, LASER beam ...

Tool nose radius compensation

When a contour is programmed, it is assumed that the tool is pointed. Since this is not always the case in practice, the curvature radius of the tool used is specified so that the control can make allowance for it. The curvature center point is guided along an equal distance to the contour at an offset corresponding to the curvature radius.

Tool offset

A tool is selected by programming a **T function** (5 decades, integer) in the block. Up to nine tool edges (D addresses) can be assigned to each T number. The number of tools to be managed by the control is defined in parameterization.

Tool radius compensation

Direct programming of a \rightarrow workpiece radius requires the control to be able to travel a path equidistant to the programmed contour, taking the radius of the tool used into account (G41/G42).

Transformation

Programming in the Cartesian coordinate system, execution in a non-Cartesian coordinate system (e.g. with machine axes as rotary axes).

Travel range

The maximum permissible travel range for linear axes is ± 9 decades. The absolute value depends on the selected precision for input and position control and the unit of measurement (inch or metric system of measurement).

U**User program**

User programs for S7-300 PLCs are written in the STEP 7 programming language. The user program is modular in structure and consists of individual blocks.

The basic types of block are as follows:

Code blocks: These contain the STEP 7 commands.

Data blocks: These contain constants and variables for the STEP 7 program.

User-defined variable

Users can define variables in the parts program or the data block for their own use (global user data). A definition contains a data type specification and the variable name. See also \rightarrow system variable.

V**Variable definition**

A variable definition includes the specification of a data type and a variable name. The variable name can be used to address the value of the variable.

Velocity control

In order to achieve an acceptable travel velocity in movements which call for very small adjustments of position in a block, the control can use -> Look ahead and thus analyze a number of blocks in advance.

W**Working area**

Three-dimensional zone into which the tool tip can be moved on account of the physical design of the machine tool.

See also -> protection zone.

Working area limitation

Working area limitation is a means of restricting axis movements over and above the restrictions imposed by limit switches. A pair of values delimiting the protected zone can be defined for each axis.

Work offset

Specification of a new reference point for a coordinate system by means of a reference to an existing zero and a -> frame.

1. Settable

SINUMERIK FM-NC: Four independent work offsets can be selected per CNC axis.

SINUMERIK 840D: A parameterizable number of work offsets is available for each CNC axis. Each of the work offsets can be selected by G functions and selection is exclusive.

2. External

All offsets which define the position of the workpiece zero can be overlaid with an external work offset

- defined by handwheel (DRF offset) or
- defined by the PLC.

3. Programmable

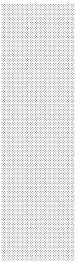
Work offsets can be programmed for all path and positioning axes by means of the TRANS instruction.

Workpiece

1. Part to be produced/machined by the machine tool or
2. A directory containing programs and other data. Workpieces are stored in directories.

Workpiece contour

Setpoint contour of the -> workpiece to be created/machined.

**Workpiece coordinate system**

The datum of the workpiece coordinate system is the -> workpiece zero. If the workpiece coordinate system is used for programming, dimensions and directions are referenced to this system.

**Workpiece zero**

The workpiece zero is the datum for the -> workpiece coordinate system. It is defined by distances from the machine zero.

**X****Y****Z**

C References**General Documentation**

- /BU/** SINUMERIK 840D/840Di/810D/802S, C, D
Ordering Information
Catalog NC 60
Order No.: E86060-K4460-A101-A9-7600
- /IKPI/** **Catalog IK PI 2000**
Industrial Communication and Field Devices
Order No. of bound edition: E86060-K6710-A101-A9-7600
Order No. of single-sheet edition: E86060-K6710-A100-A9-7600
- /ST7/** **SIMATIC**
SIMATIC S7 Programmable Logic Controllers
Catalog ST 70
Order No.: E86060-K4670-A111-A3
- /ZI/** SINUMERIK, SIROTEC, SIMODRIVE
Accessories and Equipment for Special-Purpose Machines
Catalog NC Z
Order No.: E86060-K4490-A001-A8-7600

Electronic Documentation

- /CD7/** The SINUMERIK System
DOC ON CD (11.02 Edition)
(includes all SINUMERIK 840D/840Di/810D/802 and
SIMODRIVE publications)
Order No.: 6FC5 298-6CA00-0BG3

User Documentation

/AUK/	SINUMERIK 840D/810D AutoTurn Short Operating Guide Order No.: 6FC5 298-4AA30-0BP3	(10.02 Edition)
/AUP/	SINUMERIK 840D/810D AutoTurn Graphic Programming System Operator's Guide Programming / Setup Order No.: 6FC5 298-4AA40-0BP3	(02.02 Edition)
/BA/	SINUMERIK 840D/810D MMC Operator's Guide Order No.: 6FC5 298-6AA00-0BP0	(10.00 Edition)
/BAD/	SINUMERIK 840D/840Di/810D Operator's Guide: HMI Advanced Order No.: 6FC5 298-6AF00-0BP2	(11.02 Edition)
/BEM/	SINUMERIK 840D/810D Operator's Guide HMI Embedded Order No.: 6FC5 298-6AC00-0BP2	(11.02 Edition)
/BAH/	SINUMERIK 840D/840Di/810D Operator's Guide HT 6 Order No.: 6FC5 298-0AD60-0BP2	(06.02 Edition)
/BAK/	SINUMERIK 840D/840Di/810D Short Operating Guide Order No.: 6FC5 298-6AA10-0BP0	(08.02 Edition)
/BAM/	SINUMERIK 810D/840D Operator's Guide ManualTurn Order No.: 6FC5 298-6AD00-0BP0	(11.02 Edition)
/BAS/	SINUMERIK 840D/810D Operator's Guide ShopMill Order No.: 6FC5 298-6AD10-0BP1	(11.02 Edition)
/BAT/	SINUMERIK 840D/810D Operator's Guide ShopTurn Order No.: 6FC5 298-6AD50-0BP2	(03.03 Edition)

/BNM/	SINUMERIK 840D/840Di/810D Measuring Cycles User's Guide Order No.: 6FC5 298-6AA70-0BP2	(11.02 Edition)
/CAD/	SINUMERIK 840D/840Di/810D Operator's Guide CAD Reader Order No.: (included in online Help)	(03.02 Edition)
/DA/	SINUMERIK 840D/840Di/810D Diagnostics Guide Order No.: 6FC5 298-6AA20-0BP3	(11.02 Edition)
/KAM/	SINUMERIK 840D/810D Short Guide ManualTurn Order No.: 6FC5 298-5AD40-0BP0	(04.01 Edition)
/KAS/	SINUMERIK 840D/810D Short Guide ShopMill Order No.: 6FC5 298-5AD30-0BP0	(04.01 Edition)
/KAT/	SINUMERIK 840D/810D Short Guide ShopTurn Order No.: 6FC5 298-6AF20-0BP0	(07.01 Edition)
/PG/	SINUMERIK 840D/840Di/810D Programming Guide Fundamentals Order No.: 6FC5 298-6AB00-0BP2	(11.02 Edition)
/PGA/	SINUMERIK 840D/840Di/810D Programming Guide Advanced Order No.: 6FC5 298-6AB10-0BP2	(11.02 Edition)
/PGK/	SINUMERIK 840D/840Di/810D Short Guide Programming Order No.: 6FC5 298-6AB30-0BP1	(02.01 Edition)

/PGM/	SINUMERIK 840D/840Di/810D Programming Guide ISO Milling Order No.: 6FC5 298-6AC20-0BP2	(11.02 Edition)
/PGT/	SINUMERIK 840D/840Di/810D Programming Guide ISO Turning Order No.: 6FC5 298-6AC10-0BP2	(11.02 Edition)
/PGZ/	SINUMERIK 840D/840Di/810D Programming Guide Cycles Order No.: 6FC5 298-6AB40-0BP2	(11.02 Edition)
/PI/	PCIN 4.4 Software for Data Transfer to/from MMC Module Order No.: 6FX2 060-4AA00-4XB0 (GE, EN, FR) Order from: WK Fürth	
/SYI/	SINUMERIK 840Di System Overview Order No.: 6FC5 298-6AE40-0BP0	(02.01 Edition)

Manufacturer/Service Documentation**a) Lists**

/LIS/ SINUMERIK 840D/840Di/810D/
SIMODRIVE 611D
Lists (11.02 Edition)
Order No.: 6FC5 297-6AB70-0BP3

b) Hardware

/BH/ SINUMERIK 840D/840Di/810D
Operator Components Manual (HW) (11.02 Edition)
Order No.: 6FC5 297-6AA50-0BP2

/BHA/ SIMODRIVE **Sensor**
Absolute Position Sensor with Profibus-DP
User's Guide (HW) (02.99 Edition)
Order No.: 6SN1 197-0AB10-0YP1

/EMV/ SINUMERIK, SIROTEC, SIMODRIVE
EMC Installation Guideline (06.99 Edition)
Planning Guide (HW)
Order No.: 6FC5 297-0AD30-0BP1

/PHC/ SINUMERIK 810D
Configuring Manual (HW) (09.01 Edition)
Order No.: 6FC5 297-4AD10-0BP1

/PHD/ SINUMERIK 840D
Configuring Manual NCU 561.2-573.2 (HW) (09.01 Edition)
Order No.: 6FC5 297-6AC10-0BP1

/PHF/ SINUMERIK FM-NC
Configuring Manual NCU 570 (HW) (04.96 Edition)
Order No.: 6FC5 297-3AC00-0BP0

/PMH/ SIMODRIVE **Sensor**
Measuring System for Main Spindle Drives
Configuring/Installation Guide, SIMAG-H (HW) (05.99 Edition)
Order No.: 6SN1197-0AB30-0BP0

c) Software**/FB1/**

SINUMERIK 840D/840Di/810D

Description of Functions Basic Machine (Part 1)

(11.02 Edition)

(the individual sections are listed below)

Order No.: 6FC5 297-6AC20-0BP2

- A2 Various Interface Signals
- A3 Axis Monitoring, Protection Zones
- B1 Continuous Path Mode, Exact Stop and Look Ahead
- B2 Acceleration
- D1 Diagnostic Tools
- D2 Interactive Programming
- F1 Travel to Fixed Stop
- G2 Velocities, Setpoint/Actual-Value Systems, Closed-Loop Control
- H2 Output of Auxiliary Functions to PLC
- K1 Mode Group, Channels, Program Operation Mode
- K2 Axes, Coordinate Systems, Frames
Actual-Value System for Workpiece, External Zero Offset
- K4 Communication
- N2 EMERGENCY STOP
- P1 Transverse Axes
- P3 Basic PLC Program
- R1 Reference Point Approach
- S1 Spindles
- V1 Feeds
- W1 Tool Compensation

/FB2/

SINUMERIK 840D/840Di/810D

Description of Functions Extended Functions (Part 2)

(11.02 Edition)

including FM-NC: Turning, Stepper Motor

(the individual sections are listed below)

Order No.: 6FC5 297-6AC30-0BP2

- A4 Digital and Analog NCK I/Os
- B3 Several Operator Panels and NCUs
- B4 Operation via PC/PG
- F3 Remote Diagnostics
- H1 Jog with/without Handwheel
- K3 Compensations
- K5 Mode Groups, Channels, Axis Replacement
- L1 FM-NC Local Bus
- M1 Kinematic Transformation
- M5 Measurements
- N3 Software Cams, Position Switching Signals

N4 Punching and Nibbling
P2 Positioning Axes
P5 Oscillation
R2 Rotary Axes
S3 Synchronous Spindles
S5 Synchronized Actions (up to and including SW 3)
S6 Stepper Motor Control
S7 Memory Configuration
T1 Indexing Axes
W3 Tool Change
W4 Grinding

/FB3/**SINUMERIK 840D/840Di/810D****Description of Functions, Special Functions (Part 3)****(11.02 Edition)**

(the individual sections are listed below)

Order No.: 6FC5 297-6AC80-0BP2

F2 3-Axis to 5-Axis Transformation
G1 Gantry Axes
G3 Cycle Times
K6 Contour Tunnel Monitoring
M3 Coupled Motion and Leading Value Coupling
S8 Constant Workpiece Speed for Centerless Grinding
T3 Tangential Control
TE1 Clearance Control
TE2 Analog Axis
TE3 Master-Slave for drives
TE4 Transformation Package Handling
TE5 Setpoint Exchange
TE6 MCS Coupling
TE7 Retrace Support
TE8 Path-Synchronous Switch Signal
V2 Preprocessing
W5 3D Tool Radius Compensation

/FBA/	SIMODRIVE 611D/SINUMERIK 840D/810D Description of Functions, Drive Functions (11.02 Edition) (the individual sections are listed below) Order No.: 6SN1 197-0AA80-0BP9
	DB1 Operational Messages/Alarm Reactions
	DD1 Diagnostic Functions
	DD2 Speed Control Loop
	DE1 Extended Drive Functions
	DF1 Enable Commands
	DG1 Encoder Parameterization
	DL1 Linear Motor MD
	DM1 Calculation of Motor/Power Section Parameters and Controller Data
	DS1 Current Control Loop
	DÜ1 Monitors/Limitations
/FBAN/	SINUMERIK 840D/SIMODRIVE 611 DIGITAL Description of Functions ANA MODULE (02.00 Edition) Order No.: 6SN1 197-0AB80-0BP0
/FBD/	SINUMERIK 840D Description of Functions Digitizing (07.99 Edition) Order No.: 6FC5 297-4AC50-0BP0
	DI1 Start-Up
	DI2 Scan with Tactile Sensor (scancad scan)
	DI3 Scan with Laser (scancad laser)
	DI4 Writing Milling Programs (scancad mill)
/FBDN/	SINUMERIK 840D/810D IT Solutions (01.02 Edition) System for NC Data Management and Data Distribution (DNC NT-2000) Order No.: 6FC5 297-5AE50-0BP2
/FBDT/	SINUMERIK 840D/810D IT Solutions (09.02 Edition) Sin DNC NC Data Transfer via Network Description of Functions Order No.: 6FC5 297-5AE70-0BP0

/FBFA/	<p>SINUMERIK 840D/840Di/810D ISO Dialects for SINUMERIK (11.02 Edition) Description of Functions Order No.: 6FC5 297-6AE10-0BP3</p>
/FBFE/	<p>SINUMERIK 840D/810D Description of Functions Remote Diagnosis (11.02 Edition) Order No.: 6FC5 297-0AF00-0BP2</p> <p>FE1 Remote Diagnosis FE2 Interrupt-Controlled Email Messaging: @Event</p>
/FBH/	<p>SINUMERIK 840D/840Di/810D HMI Configuring Package (11.02 Edition) Order No.: (supplied with the software)</p> <p>Part 1 User's Guide Part 2 Description of Functions</p>
/FBHLA/	<p>SINUMERIK 840D/SIMODRIVE 611 digital Description of Functions HLA Module (04.00 Edition) Order No.: 6SN1 197-0AB60-0BP2</p>
/FBMA/	<p>SINUMERIK 840D/810D Description of Functions ManualTurn (08.02 Edition) Order No.: 6FC5 297-6AD50-0BP2</p>
/FBO/	<p>SINUMERIK 840D/810D Configuring OP 030 Operator Interface (09.01 Edition) Description of Functions (the individual sections are listed below) Order No.: 6FC5 297-6AC40-0BP0</p> <p>BA Operator's Guide EU Development Environment (Configuring Package) PSE Introduction to Configuring of Operator Interface IK Screen Kit: Software Update and Configuration PS Online only: Configuring Syntax (Configuring Package)</p>
/FBP/	<p>SINUMERIK 840D Description of Functions C-PLC Programming (03.96 Edition) Order No.: 6FC5 297-3AB60-0BP0</p>

/FBR/	<p>SINUMERIK 840D/810D IT Solutions (09.01 Edition)</p> <p>SINCOM Computer Link Description of Functions Order No.: 6FC5 297-6AD60-0BP0</p> <p>NFL Host Computer Interface NPL PLC/NCK Interface</p>
/FBSI/	<p>SINUMERIK 840D/SIMODRIVE Description of Functions (09.02 Edition)</p> <p>SINUMERIK Safety Integrated Order No.: 6FC5 297-6AB80-0BP1</p>
/FBSP/	<p>SINUMERIK 840D/810D Description of Functions ShopMill (11.02 Edition) Order No.: 6FC5 297-6AD80-0BP1</p>
/FBST/	<p>SIMATIC FM STEPDRIVE/SIMOSTEP Description of Functions (01.01 Edition) Order No.: 6SN1 197-0AA70-0YP4</p>
/FBSY/	<p>SINUMERIK 840D/810D Description of Functions Synchronized Actions (10.02 Edition) Order No.: 6FC5 297-6AD40-0BP2</p>
/FBT/	<p>SINUMERIK 840D/810D Description of Functions ShopTurn (03.03 Edition) Order No.: 6FC5 297-6AD70-0BP2</p>
/FBTC/	<p>SINUMERIK 840D/840DI/810D IT Solutions SINUMERIK Tool Data Communication SinTDC (01.02 Edition) Description of Functions Order No.: 6FC5 297-5AF30-0BP0</p>
/FBTD/	<p>SINUMERIK 840D/810D IT Solutions (03.01 Edition)</p> <p>SINTDI Tool Information System (SinTDI) with Online Help Description of Functions Order No.: 6FC5 297-6AE00-0BP0</p>

/FBU/	SIMODRIVE 611 universal/universal E Closed-Loop Control Component for Speed Control and Positioning Description of Functions Order No.: 6SN1 197-0AB20-0BP5	(02.02 Edition)
/FBW/	SINUMERIK 840D/810D Description of Functions Tool Management Order No.: 6FC5 297-6AC60-0BP1	(11.02 Edition)
/FBWI/	SINUMERIK 840D/840Di/810D Description of Functions WinTPM Order No.: The document is an integral part of the software	(02.02 Edition)
/HBA/	SINUMERIK 840D/840Di/810D Manual @Event Order No.: 6AU1900-0CL20-0BA0	(03.02 Edition)
/HBI/	SINUMERIK 840Di Manual Order No.: 6FC5 297-6AE50-0BP1	(09.02 Edition)
/INC/	SINUMERIK 840D/840Di/810D Commissioning Tool SINUMERIK SinCOM NC System Description Order No.: (an integral part of the online Help for the start-up tool)	(02.02 Edition)
/PAP/	SIMODRIVE Sensor Absolute value encoder with PROFIBUS-DP User Manual Order No.: 6SN1197-0AB10-0YP1	(02.99 Edition)
/PFK/	SIMODRIVE Planning Guide 1FT5/1FT6/1FK6 Motors AC Servo Motors for Feed and Main Spindle Drives Order No.: 6SN1 197-0AC20-0BP0	(12.01 Edition)

/PJE/	SINUMERIK 840D/810D Configuring Package HMI Embedded Description of Functions: Software Update, Configuration, Installation Order No.: 6FC5 297-6EA10-0BP0 (the document CS Configuring Syntax is supplied with the software and available as a pdf file)	(08.01 Edition)
/PJFE/	SIMODRIVE Planning Guide Built-In Synchronous Motors 1FE1 Three-Phase AC Motors for Main Spindle Drives Order No.: 6SN1 197-0AC00-0BP1	(09.01 Edition)
/PJLM/	SIMODRIVE Planning Guide Linear Motors 1FN1, 1FN3 ALL General Information about Linear Motors 1FN1 1FN1 Three-Phase AC Linear Motor 1FN3 1FN3 Three-Phase AC Linear Motor CON Connections Order No.: 6SN1 197-0AB70-0BP2	(11.01 Edition)
/PJMI/	SIMODRIVE Planning Guide Motors Three-Phase AC Motors for Feed and Main Spindle Drives Order No.: 6SN1 197-0AA20-0BP5	(11.00 Edition)
/PJTM/	SIMODRIVE Planning Guide Integrated Torque Motors 1FW6 Order No.: 6SN1 197-0AD00-0BP0	(08.02 Edition)
/PJU/	SIMODRIVE 611 Planning Guide Converters Order No.: 6SN1 197-0AA00-0BP6	(08.02 Edition)
/PMS/	SIMODRIVE Planning Guide ECO Motor Spindle for Main Spindle Drives Order No.: 6SN1 197-0AD04-0BP0	(04.02 Edition)

/POS1/	SIMODRIVE POSMO A User's Guide Distributed Positioning Motor on PROFIBUS DP Order No.: 6SN2 197-0AA00-0BP3	(08.02 Edition)
/POS2/	SIMODRIVE POSMO A Installation Guide (supplied with every POSMO A)	
/POS3/	SIMODRIVE POSMO SI/CD/CA Distributed Servo Drive Systems, User's Guide Order No.: 6SN2 197-0AA20-0BP3	(08.02 Edition)
/PPH/	SIMODRIVE Planning Guide AC Induction Motors for Main Spindle Drives 1PH2-/1PH4-/1PH7 Order No.: 6SN1 197-0AC60-0BP0	(12.01 Edition)
/PPM/	SIMODRIVE Planning Guide Hollow-Shaft Motors for Main Spindle Drives 1PM4 and 1PM6 Order No.: 6SN1 197-0AD03-0BP0	(10.01 Edition)

/S7H/	SIMATIC S7-300 Manual: Assembly, CPU Data (HW) Reference Manual: Module Data Order No.: 6ES7 398-8AA03-8AA0	(10.98 Edition)
/S7HT/	SIMATIC S7-300 STEP 7 Manual, Fundamentals, V. 3.1 Order No.: 6ES7 810-4CA02-8AA0	(03.97 Edition)
/S7HR/	SIMATIC S7-300 Manual STEP 7, Reference Manuals, V. 3.1 Order No.: 6ES7 810-4CA02-8AR0	(03.97 Edition)
/S7S/	SIMATIC S7-300 FM 353 Positioning Module for Stepper Drive Order together with Configuring Package	(04.97 Edition)
/S7L/	SIMATIC S7-300 FM 354 Positioning Module for Servo Drive Order together with Configuring Package	(04.97 Edition)
/S7M/	SIMATIC S7-300 FM 357.2 Multimodule for Servo or Stepper Drive Order together with Configuring Package	(01.01 Edition)
/SP/	SIMODRIVE 611-A/611-D SimoPro 3.1 Program for Configuring Machine Tool Drives Order No.: 6SC6 111-6PC00-0AA□, Order from: WK Fürth	

**d) Installation and
Start-up**

//IAA/	SIMODRIVE 611A Installation and Start-Up Guide Order No.: 6SN 1197-0AA60-0BP6	(10.00 Edition)
//IAC/	SIMODRIVE 810D Installation & Start-up Guide (including description of start-up software for SIMODRIVE 611D) Order No.: 6FC5 297-6AD20-0BP0	(03.02 Edition)
//IAD/	SINUMERIK 840D/SIMODRIVE 611D Installation and Start-Up Guide (incl. description of SIMODRIVE 611D start-up software) Order No.: 6FC5 297-6AB10-0BP2	(11.02 Edition)
//IAM/	SINUMERIK 840D/840Di/810D HMI/MMC Installation and Start-Up Guide Order No.: 6FC5 297-6AE20-0BP2	(11.02 Edition)
	AE1 Updates/Supplements BE1 Expanding the Operator Interface HE1 Online Help IM2 Starting up HMI Embedded (PCU 20) IM4 Starting up HMI Advanced (PCU 50) TX1 Creating Foreign Language Texts	

D Index

A

Access rights 2-45
Acknowledgement symbols changed 8-409
Activate basic frame immediately 5-265
Activate skip levels 4-172
Alarm acknowledgment key 2-35
Alarm Help 2-75
Alarm messages 8-408
Alt key 2-37
Area switchover key 2-35
Arithmetic variables 5-246
Automatic 2-41, 4-149
 Program overview 4-151
Axis feedrates 4-111

B

Baud rate 7-367
Blank 2-36
Block display during program execution 4-174
Block search 4-159
Block search without calculation, external
 fast 4-162

C

Calculation of tool parameters 5-189
Cancel input 2-62
Chain rule 5-194
Changing the HMI interface 9-443
Changing the menu window 2-58
Changing the operand value 8-426
Changing/finding user data 5-268
Channel
 Active 2-69
 Interrupted 2-69
 Reset 2-69
 Switchover 2-69
Channel states 2-69

Channels 4-102
Cleaning 10-453
Close contour 6-295
Communication error log 8-419
Configuration data 8-418
Confirm 2-62
Contour
 Graphic representation 6-291
 Symbolic representation 6-290
Contour definition programming 6-285
Contour element
 Add 6-293, 6-294
 Selection 6-293
Contour elements
 Input screen forms for parameterization
 6-292
COPY 2-87
Copying 6-341
Create workpiece directory 6-333
Creating a new directory 7-390
Creating a new file 7-390
Ctrl key 2-37
Cursor 2-36
Cycle parameterization 2-66
Cycle recompilation 2-67
Cycles 6-279

D

Data bits 7-367
Data management limits 7-355
Data selection 7-359
Data structure 4-99
Defining and activating user data 7-396
Defining user data 5-267
Delete key 2-37
Delete key (backspace) 2-36
Deleting 6-344
Deleting the operand value 8-426
Determine tool offsets 5-203

- Diagnosis 8-405
 - Basic display 8-406
- Directories 7-356
- Directory structure 7-352
- Display active axes 8-411
- Display all axes 8-411
- Display machine data 9-436
- Display NC system resources 8-430
- Display options 9-438
- Display program level 4-108
- Display selected axes 8-411
- Display system frame 4-136
- Displaying auxiliary functions 4-112
- Displaying G functions 4-111
- Displaying loadable compile cycles 8-423
- Displaying spindles 4-113
- Displaying transformations 4-111
- Downtime settings
 - MMC 103 6-325
- DRF offset 4-175
- Drive diagnostics 8-410
- Dry run feedrate 5-251

E

- Edit key 2-37
- Editing a parts program 2-62
- Editing values 2-60
- Editor Help 2-75, 2-77
- Electronic gear 5-255
- Emergency stop key 2-40
- Enabling 6-347
- End of line key 2-37
- Executing job list 2-91
- Expansion key 2-34
- Exporting cycle versions 8-423
- Extended help for programming commands 2-82
- External network drive 6-349

F

- Fast external block search without calculation 4-162
- Feed control 2-42

- Feedrate override 2-42

File

- Changing its properties 7-394
- Copy 7-391
- Copying 6-341
- Delete 6-344, 7-393
- Insert 7-391
- Inserting 6-341
- Load 7-390
- Renaming 6-345
- Unload 7-390
- File functions 8-429
- File names 7-362
- File types 6-331, 7-357
- Find 2-65
- Find path for program call 6-337
- Find/Go to 2-64
- Flat D numbers 5-179
- Following axis 5-255
- Free contour programming 2-66
- Full standard keyboard 2-32

G

- Geometry axes 4-109
- Global machine status display 2-50
- Go to 2-65
- GUD 5-267

H

- Handwheel 4-114
- Hard disk 4-154, 7-353
- Hardkey assignment 2-32
- Help 2-75
- Help function 2-77
- Help in the Editor 2-78, 2-82
 - Overview 2-77
- Help with free contour programming 6-303
- Hierarchical file structure 7-358

I

- Immediate activation of tool offset 5-204
- Inc
 - Incremental dimension 4-130
- Inc keys 2-41
- Inch/metric conversion 2-74
- Inch/metric switchover (SW 5 or later) 4-121
- Incremental 2-41
- Incremental value 2-41
- Information key 2-35, 2-78, 2-82
- Initialization program
 - User data defining 5-267
- Input key 2-37
- Interface parameters 7-369
- Interfaces 7-353
- ISO program (MMC 103)
 - Import 7-384
- ISO program (MMC103) 7-384
 - Export 7-385
- ISO punched tape format 7-384

J

- Job list 2-83
- Job list syntax 2-85
- Jog 2-40, 4-126
 - Basic display 4-127
- Jog continuous 5-249
- Jog data 5-249
- Jog feedrate 5-249
- Jog spindle speed 5-249

K

- Keyswitch 2-45

L

- Leading axis 5-255
- LOAD 2-86
- Load standard 6-336
- Loading 5-224, 6-339
 - From magazine list 5-225, 5-228
- Loading list 4-153

- Log 6-348

M

- m:n communication links 2-70
- MAC
 - Changing/finding user data 5-268
- Machine area key 2-34
- Machine axes 4-109
- Machine control panel keys 2-40
- Machine control panels 2-39
- Machine functions 2-41, 4-101
- Machine Operating Area 4-97
- Machine zero 5-256
- Maintenance 10-451
- Mark (select) block 2-64
- Masking filter 9-438
- MCS 2-43, 4-109
- MCS/WCS switchover 4-109
- MD Help 2-75
- MDI 2-40, 4-142
- MF-2 keyboard 2-32
- MMC 103 7-371
 - Basic display 7-371
 - Log 7-382
 - Reading in data 7-379
 - Reading out data 7-381
 - Series start-up 7-398
 - Setting the interface 7-374
 - Software update 7-402
- Multi-channel search
 - In Program test mode 4-166

N

- NC active data 7-352
- NC card
 - Series start-up from (SW 5 or later) 7-399
 - Delete data on ... (SW 5 or later) 7-393
 - Export data to ... (SW 5 or later) 7-381
 - Import data from ... (SW 4.4 or later) 7-380

- Restoring the original state ... (SW 5 or later)
 - 7-401
- Upgrade from ... (SW 5 or later) 7-403
- NC Start 2-47
- NC Stop 2-47
- Network drive 6-349
- New parts program 6-332
- New tool 5-198
- New tool edge 5-201
- New workpiece 6-332

O

- Operand forms 8-428
- Operating areas 1-23
- Operating data 10-452
- Operating example 3-95
- Operating mode group 4-102
- Operating modes 2-40, 4-100
- Operating sequences 2-27
- Operating sequences, general 2-57
- Operator components 2-27
- Operator panel keys 2-34
- Operator panels 2-29
- Orientable Toolholder
 - Simulation 6-328
- Overstore 4-169
- Overwrite 2-64

P

- Page up 2-36, 2-37
- Parameter no. for radius calculation 5-194
- Parameters 5-177
- Parity 7-367
- Part program 6-279
- Parts program
 - Cancel 4-107
 - Continue 4-107
 - Loading, unloading 4-152
 - Select 6-332
 - Start 4-107
 - Stop 4-107
- Password 2-46

- Path name 7-360
- PC format 7-365
- PLC 9-441
- PLC status 8-425
- Pocket calculator 2-91
- Pocket calculator function 2-74
- Predefined package lists 8-421
- Preset 4-117
- Program
 - Basic display 6-280
- Program 6-277
 - Edit 6-282
 - Enabling 6-347
 - Execution 6-336
 - Loading 6-339
 - Management 6-330
 - Simulate, MMC 103 6-309
 - Store 6-280
 - Unloading 6-339
- Program control 2-47, 4-171
- Program editing 4-157
- Program management 6-340
- Program overview 2-57
- Program protection
 - Selective 6-284
- Program selection 2-57
- Program types 6-279
- Protection levels 2-45
- Protection zones 5-254
- Punched tape format 7-361

R

- R variables 5-246
 - Change 5-246
 - Delete 5-247
 - Find 5-247
- Rapid traverse overlay 2-43
- Rapid traverse override 2-42
- Recall key 2-34
- Ref point 2-41
- Reference point approach 4-123
- Renaming 6-345
- Repos 2-41, 4-131

- Reset 2-48
- Restoring the original state 7-401

- S**
- Safety Integrated 4-132
- Save file 2-65
- Save program (MMC 103) 4-145
- Scratching 4-134, 4-137
- Search destination 4-159
- SELECT 2-86
- Select operating mode 4-104
- Selecting 6-336
- Selecting a directory 2-59
- Selecting a file 2-59
- Selection key 2-36
- Service axis 8-412
- Service display 8-410
- Service drive 8-413, 8-414
- Service overview 8-410
- Set actual value 4-119
- Setting data 5-248
 - Change 5-254
 - Display 5-253
 - Find 5-253
 - Other types 5-253
- Setting defaults 6-336
- Setting the date 9-442
- Setting the time 9-442
- Settings (MMC 103 only) 2-67, 6-308
- Shift key 2-35
- Short help for program commands 2-78
- Simulation 2-67
 - Orientable toolholder 6-328
- Simulation of complete turning operations 6-309
- Simulation of drilling/milling 6-309
- Simulation settings
 - MMC 103 6-321
- Simulation user interface
 - MMC 103 6-310
- Single block 2-47
- Softkey 2-34
- Softkey assignment 2-32
- Software version 8-420

- Special axes 4-109
- Special functions 7-368
- Spindle control 2-44
- Spindle data 5-250
- Spindle number 5-194
- Spindle override 2-44
- Standardized program and data management 7-389
- Start of search in Program test mode
 - multi-channel 4-166
- Starting angle for thread cutting 5-252
- Start-up
 - Basic display 9-432
- Start-up 9-431
- Stop bits 7-368
- Storing
 - Setup data 6-335
- Storing setup data 6-335
- Subroutine 6-279
- Switch over channel 2-35
- Switching the control OFF 1-25
- Switching the control ON 1-25
- Synchronized actions
 - Status 4-115
- System settings 9-446
- System variable
 - Display 5-270
- System variables
 - Logging 5-273

- T**
- Tab key 2-37
- Teach In 2-41, 4-142, 4-146
- Tool
 - Create 5-198
 - Delete 5-201
 - Display 5-200
 - Relocation 5-230
- Tool cabinet 5-235
- Tool catalog 5-232
- Tool compensation
 - Structure 5-179
- Tool data 5-179

- Change 5-233, 5-236
- Create 5-232
- Display 5-233, 5-236
- Display/change 5-215
- Tool edge
 - Create 5-201
 - Delete 5-202
- Tool management 5-205
- Tool management, Basic display 5-207
- Tool offset 5-196
- Tool offset data
 - Create 5-236
- Tool parameters 5-179
- Tool type
 - Drill 5-181
 - Grinding tools 5-182
 - Milling tool 5-180
 - Slotting saw 5-187
 - Turning tools 5-185
- Tool types 5-179
- Tool zero 5-256
- Tool-specific parameters 5-185
- Traversing axes 4-129
- Traversing speed 4-129

U

- Undo key 2-37
- Unloading 5-229
- User data 5-267
- Utilization display 8-430

V

- Variable increment 5-249
- Variable views
 - Creating 5-271
 - Manage 5-272
 - Processing 5-271
- Version screen for cycles 8-420

W

- WCS 2-43, 4-109
- Window selection key 2-36
- Work offset 5-256
 - Activate immediately 5-265
 - Active external 5-264
 - Active programmable 5-263
 - Active settable 5-262
 - Settable 5-258
- Work offsets
 - Sum of active 5-265
- Working area limitation 5-248
- Workpiece 6-279
 - Enabling 6-347
 - Execution 6-336
 - Loading, unloading 4-152
 - Select 6-332
- Workpieces
 - MMC 103 6-335
 - Number 6-335

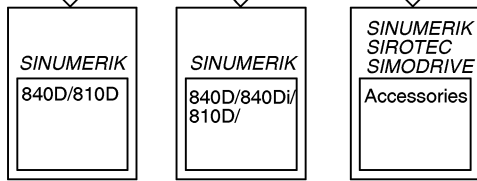
To
SIEMENS AG
A&D MC BMS
P.O. Box 3180
D-91050 Erlangen, Germany
 (Phone: ++49-(0)180-5050-222 [Hotline]
 Fax: ++49-(0)9131-98-2176 [Documentation]
 Email: motioncontrol.docu@erlf.siemens.de)

<p>From</p> <hr/> <p>Name</p> <hr/> <p>Company/Dept.</p> <hr/> <p>Address</p> <hr/> <p>Zip code:</p> <hr/> <p>Phone: /</p> <hr/> <p>Fax: /</p>	<p>Suggestions</p> <hr/> <p>Corrections</p> <p>For publication:</p> <p>SINUMERIK 840D/840Di/810D</p> <p>User Documentation</p> <hr/> <p>Operator's Guide HMI Advanced</p> <p>Order No.: 6FC5298-6AF00-0BP2 Edition: 11.02</p> <hr/> <p>Should you come across any printing errors when reading this publication, please notify us on this sheet. Suggestions for improvements are also welcome.</p>
---	---

Suggestions and/or corrections

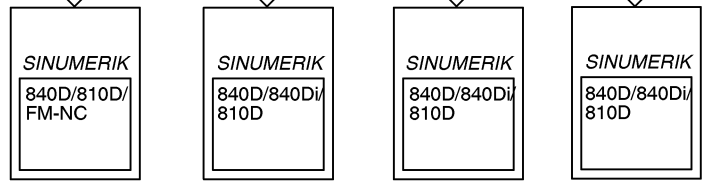
Overview of SINUMERIK 840D/840Di/810D Documentation (11.2002)

General Documentation



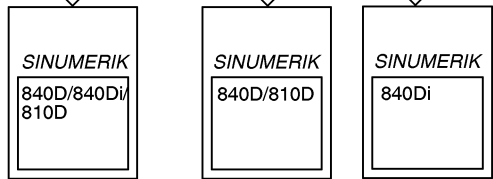
Brochure
Catalog
Catalog Ordering Info. NC 60 *)
Catalog Accessories NC-Z

User Documentation



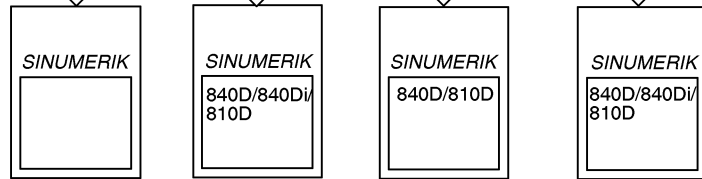
AutoTurn
 – Short Guide
 – Programming/Setup
Operator's Guide
 – HT 6
Diagnostics Guide *)
Operator's Guide *)
 – Short Guide
 – HMI Embedded
 – HMI Advanced

User Documentation



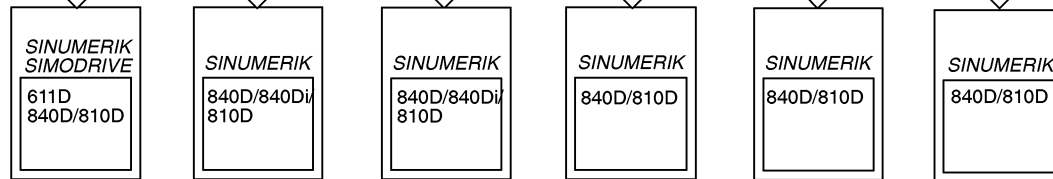
Program. Guide
 – Short Guide
 – Fundamentals *)
 – Advanced *)
 – Cycles
 – Measuring Cycles
 – ISO Turning/Milling
Operator's Guide
 – **ManualTurn**
 – Short Guide ManualTurn
 – **ShopMill**
 – Short Guide ShopMill
 – **ShopTurn**
 – Short Guide ShopTurn
System Overview

Manufacturer/Service Documentation



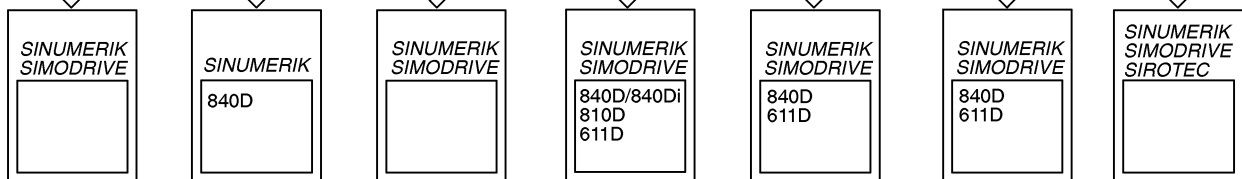
Configuring (HW) *)
 – 810D
 – 840D
Operator Components (HW) *)
 Description of Functions
 – **ManualTurn**
 – **ShopMill**
 – **ShopTurn**
 Description of Functions
Synchronized Actions

Manufacturer/Service Documentation



Description of Functions
Drive Functions *)
Description of Functions
 – Basic Machine *)
 – Extended Functions
 – Special Functions
 Description of Functions
Tool Management
Configuring Kit
 HMI Embedded
 Description of Functions
 Operator Interface
OP 030
IT Solutions
 – Computer Link
 – Tool Data Information System
 – NC Data Management
 – NC Data Transfer
 – Tool Data Communication

Manufacturer/Service Documentation



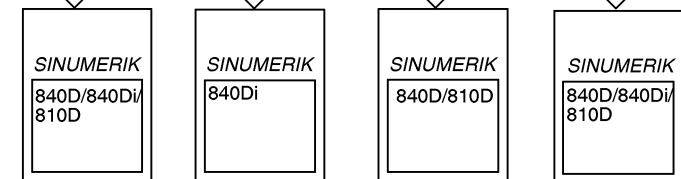
Description of Functions
 SINUMERIK
Safety Integrated
 Description of Functions
Digitizing
Installation & Start-Up Guide *)
 – 810D
 – 840D/611D
 – HMI
Lists *)
 Description of Functions
Linear Motor
 Description of Functions
 – **Hydraulics Module**
 – **Analog Module**
EMC
 Guidelines

Manufacturer/Service Documentation

Electronic Documentation



DOC ON CD *)
 The SINUMERIK System



Description of Functions
ISO Dialects for SINUMERIK
Manual (HW + Installation and Start-Up)
 Description of Functions
Remote Diagnosis
Manual @ Event

*) These documents are a minimum requirement