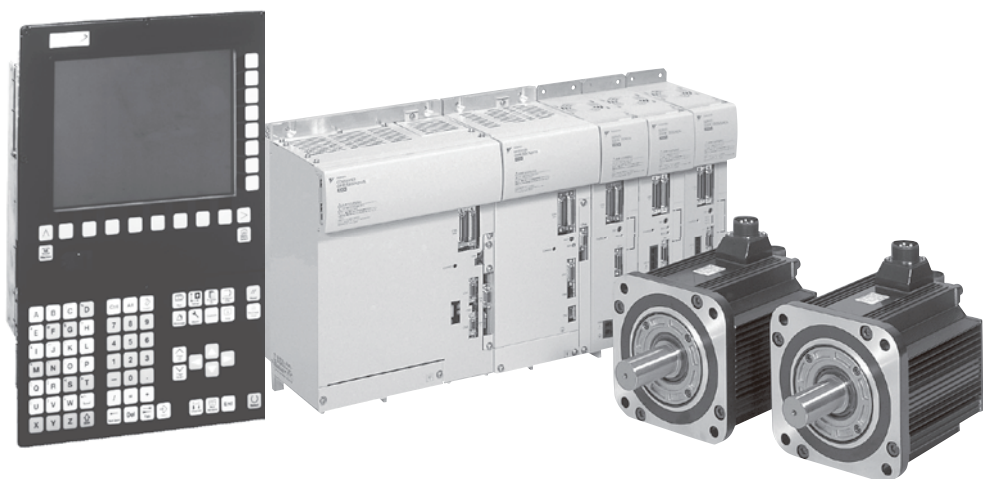


SINUMERIK 840D/840Di/810D

Operator's Guide HMI Advanced



Yaskawa Siemens Numerical Controls Corp. has been merged to Siemens K.K. and Siemens Japan K.K. as of August, 2010 respectively.

"Yaskawa Siemens Numerical Controls Corp." in this manual should therefore be understood as "Siemens Japan K.K."

This manual is made with the purpose of using in combination with "Yaskawa Siemens CNC Series Operating Manual for Standard HMI (NCSIE-SP02-24)".

For actually available functions in your machine, Please refer to the documents published by the machine manufacturer.

The Internet address of the web site for our company and products is as follows:

<http://www.siemens.co.jp/>

For EU Declaration of Conformity, please ask to the manufacturer of your machine.

SINUMERIK 840D/840Di/810D

HMI Advanced

Operator's Guide

Valid for

<i>Control</i>	<i>Software version</i>
SINUMERIK 840D powerline	7
SINUMERIK 840DE powerline (export version)	7
SINUMERIK 840Di	3
SINUMERIK 840DiE (export version)	3
SINUMERIK 810D powerline	7
SINUMERIK 810DE powerline (export version)	7

Introduction	1
Operator Components/ Sequence of Operations	2
Example of Operation	3
Machine	4
Parameters	5
Program	6
Services	7
Diagnostics	8
Startup	9
Service	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
03.04	6FC5298-6AF00-0BP3	C

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

Edition	Order No.	Remarks
03.04	6FC5298-7CA00-0BG0	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.siemens.com/motioncontrol>

This documentation was produced with Word V 9.0 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, 1995-2004. 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.

Preface



Structure of the Documentation

Audience

Standard scope

Validity

Note

This HMI Advanced Operator's Guide describes the user interfaces and the operating sequences for the main operations on systems with HMI Advanced 6.4

The SINUMERIK documentation is organized in 3 parts:

- General Documentation
- User Documentation
- Manufacturer/Service Documentation

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

This Operator's Guide describes only the functionality of the standard scope. Extensions or changes made by the machine tool manufacturer are documented by the machine tool manufacturer.

More detailed information about other publications concerning SINUMERIK 840D and 810D 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.

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 included in the export version:

Function	810DE	840DE
Machining package 5 axes	–	–
Handling transformation package (5 axes)	–	–
Multi-axis interpolation (> four axes)	–	–
Helical interpolation 2D+6	–	–
Synchronous actions step 2	–	O ¹⁾
Measurement level 2	–	O ¹⁾
Adaptive control	O ¹⁾	O ¹⁾
Continuous dressing	O ¹⁾	O ¹⁾
Using the compile cycles (OEM)	–	–
Sag compensation, multi-dimensional	–	O ¹⁾

– Function not possible

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 meaningful and practicable. The various levels of information have been organized such that you can selectively access the information you need for the task in hand.



1. Function

The theoretical section is primarily intended as learning material for the NC entry-level user and includes important information to assist the user to understand the operator functions.

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



2. Sequence of operations

This section contains the sequence of keys required for the operation at a glance. If inputs 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 functions are disabled to protect them from unauthorized access. The machine manufacturer can customize or modify the described functionality. Please follow the instructions of the machine-tool manufacturer.



Explanation of symbols




Function




Operating sequence




Other Information




Cross-references to other documentation or sections



Danger notes




Additional notes or background information



Ordering data supplement



Explanation



Description of syntax

Programming examples

**Notes**

The following notes used in the documentation are of special significance:

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



In this documentation, you will find this symbol with a reference to an ordering option. The described function can only run if the control contains the designated option.

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 documentation.

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.

Supplementary devices

The applications of SIEMENS controls can be expanded for specific purposes through the addition of special add-on devices, equipment and expansions supplied by SIEMENS.

Personnel

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

The areas of **responsibility** assigned to personnel involved in setting up, operating and maintaining the equipment must be clearly **specified** and their compliance **verified**.

Procedure

Before the control is started up, it must be ensured that the Operator's Guides have been read and understood by the personnel responsible. Operation of this equipment is dependent on continuous monitoring of the overall technical condition of the control (with a view to identifying externally visible defects and damage as well as changes in the operating behavior of the control).

Service



Repairs to equipment may only be carried out by **personnel specially trained and qualified** for the application in question in accordance with the provisions specified in the maintenance and servicing guides. All relevant safety regulations must be followed.

The following are deemed as **improper usage** and **exclude the manufacturer from all liability**:

Any usage or application incompatible with or beyond the scope of the items specified above.

If the control is **not in technically perfect condition**, or is operated without due regard for safety regulations and accident prevention instructions given in the Instruction Manual.

Cases where faults which could affect safety are not remedied **before** starting up the control.

Any **modification, bypassing or disabling** of functions or components on the control whose intended purpose is to ensure proper functioning, unrestricted use of equipment and/or 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.



Notes

Contents

Introduction	1-21
1.1 The SINUMERIK 840D/810D system	1-22
1.2 Notes on handling	1-24
1.3 Activation/deactivation.....	1-25
Operator Components/Sequence of Operations	2-27
2.1 Operator panels	2-29
2.1.1 OP 010 operator panels	2-29
2.1.2 OP 010S operator panel	2-30
2.1.3 OP 010C operator panel	2-30
2.1.4 OP 012 operator panel.....	2-31
2.1.5 OP 015 operator panel.....	2-31
2.1.6 Full standard keyboard.....	2-32
2.2 Operator panel keys.....	2-34
2.3 Machine control panels	2-39
2.4 Machine control panel keys.....	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.....	2-46
2.5 Screen layout	2-47
2.5.1 Overview	2-47
2.5.2 Global machine status display	2-48
2.5.3 Program control display	2-52
2.6 General sequence of operation.....	2-54
2.6.1 Program overview and program selection.....	2-54
2.6.2 Switch menu window.....	2-55
2.6.3 Select directory/file	2-56
2.6.4 Edit input/values	2-57
2.6.5 Confirm/cancel input	2-58
2.6.6 Editing part programs in the ASCII editor.....	2-59
2.6.7 Switch channel	2-65
2.6.8 m:n Communication links.....	2-66
2.6.9 Pocket calculator.....	2-70
2.7 Help	2-71
2.7.1 Editor help	2-73
2.7.2 Quick help for program commands.....	2-74
2.7.3 Extended help for program commands.....	2-77
2.8 Job list	2-78

2.8.1	Syntax description for job lists	2-80
2.8.2	Example of a job list with two-channel 1:1 connections	2-83
2.8.3	Example of a job list with multi-channel m:n connections	2-84
2.8.4	"Execute job list" operating sequence	2-85
2.8.5	Rename workpieces with job lists	2-87
2.8.6	Copy workpieces with job lists	2-88
2.8.7	Archive workpieces with job lists in case of m:n.....	2-88

Example of Operation 3-89

3.1	Typical sequence of operation.....	3-89
-----	------------------------------------	------

Machine Operating Area 4-91

4.1	Data structure of the NC control.....	4-93
4.1.1	Operating modes and machine functions.....	4-94
4.1.2	Mode groups and channels	4-96
4.1.3	Channel-wide status display with symbols (SW 6.2 and higher)	4-97
4.1.4	Dual-channel display (SW 6.4 and higher).....	4-98
4.1.5	Operating mode selection/change.....	4-99
4.2	General functions and displays	4-102
4.2.1	Stop/start/cancel/continue part program	4-102
4.2.2	Display program level	4-103
4.2.3	Toggling between Machine/Work (MCS/WCS).....	4-104
4.2.4	Display axis feeds.....	4-106
4.2.5	Display G functions, transformations and swivel data.....	4-106
4.2.6	Display auxiliary functions	4-107
4.2.7	Display spindles.....	4-108
4.2.8	Handwheel.....	4-109
4.2.9	Status of synchronized actions.....	4-110
4.2.10	Preset	4-112
4.2.11	Setting the actual value	4-113
4.2.12	Inch/metric switchover.....	4-114
4.3	Reference point approach	4-116
4.4	JOG mode	4-119
4.4.1	Function and main screen	4-119
4.4.2	Traverse the axes.....	4-122
4.4.3	Inc: Incremental dimensions.....	4-123
4.4.4	Repos (repositioning)	4-124
4.4.5	SI (Safety Integrated): User agreement	4-125
4.4.6	Scratching (SW 6 or higher).....	4-126
4.4.7	Scratching (SW 5 and lower).....	4-130
4.4.8	Display system frames	4-133
4.5	MDI mode.....	4-135
4.5.1	Function and main screen	4-135
4.5.2	Save program, file function.....	4-137
4.5.3	Teach-in	4-138

4.6	Automatic mode	4-140
4.6.1	Function and main screen.....	4-140
4.6.2	Program Overview	4-142
4.6.3	Loading and unloading the workpiece/part program	4-143
4.6.4	Log: Loading list of the programs	4-144
4.6.5	Executing programs from the hard disk	4-145
4.6.6	Access to external network drive	4-146
4.6.7	Program editing.....	4-148
4.6.8	Block search/setting the search destination.....	4-150
4.6.9	Accelerated external block search	4-153
4.6.10	Block search in program testing mode, multi-channel	4-156
4.6.11	Overstore	4-158
4.6.12	Program control.....	4-160
4.6.13	DRF offset.....	4-164

"Parameters" Operating Area

5-165

5.1	Tool data	5-167
5.1.1	Tool offset structure	5-167
5.1.2	Tool types and tool parameters.....	5-167
5.2	Tool offsets.....	5-182
5.2.1	Function and main screen for tool offsets	5-182
5.2.2	New tool	5-184
5.2.3	Display tool.....	5-185
5.2.4	Delete tool	5-186
5.2.5	New cutting edge.....	5-187
5.2.6	Delete cutting edge	5-188
5.2.7	Determine tool offsets	5-188
5.2.8	Activate tool offset immediately.....	5-190
5.3	Tool management	5-191
5.3.1	Basic tool management functions	5-193
5.3.2	Modifying/changing tool data.....	5-201
5.3.3	Change of meaning/representation of tool wear data	5-205
5.3.4	Grinding parameters extension in SW 6.2 or later	5-207
5.3.5	Loading.....	5-210
5.3.6	Unload	5-214
5.3.7	Relocation	5-216
5.3.8	Tool master data in the tool catalog	5-217
5.3.9	Tool offset data in the tool cabinet	5-220
5.3.10	Batch processing of tools	5-224
5.4	R parameters (arithmetic parameters).....	5-232
5.4.1	Function	5-232
5.4.2	Editing/deleting/finding R parameters	5-232
5.5	Setting data	5-234
5.5.1	Working area limitation	5-234
5.5.2	Jog data	5-235

5.5.3	Spindle data.....	5-236
5.5.4	Dry run feedrate for dry run DRY.....	5-237
5.5.5	Starting angle for thread cutting	5-238
5.5.6	Other types of setting data	5-239
5.5.7	Protection zones.....	5-240
5.5.8	Electronic gears (SW 6.3 and higher)	5-241
5.6	Work offset.....	5-242
5.6.1	Function.....	5-242
5.6.2	Changing the settable work offset (G54 ...)	5-244
5.6.3	Global work offset/frame	5-244
5.6.4	Display active settable work offset	5-247
5.6.5	Display active programmable work offset.....	5-248
5.6.6	Display active work offset external	5-249
5.6.7	Display sum of the active work offsets	5-249
5.6.8	Immediately activate work offset and basic frame	5-250
5.6.9	Actual-value display: settable zero system, ENS	5-250
5.7	User data/user variables (GUD, PUD, LUD)	5-251
5.7.1	General.....	5-251
5.7.2	Changing/finding user data/user variables	5-252
5.8	Display system variables	5-254
5.8.1	Creating/editing variable views.....	5-255
5.8.2	Manage variable views	5-257
5.8.3	Log system variables.....	5-257
"Program" Operating Area		6-261
6.1	Program types	6-263
6.1.1	Part programs.....	6-263
6.1.2	Subroutines	6-263
6.1.3	Workpieces	6-263
6.1.4	Cycles.....	6-263
6.1.5	Templates.....	6-263
6.2	Storing programs.....	6-267
6.2.1	HMI	6-267
6.3	Program main screen	6-267
6.4	Edit programs	6-269
6.4.1	Text editor	6-269
6.4.2	Undo and redo in the editor (from SW 6.3)	6-272
6.4.3	Selective program protection: *RO*	6-273
6.4.4	Reserved character strings	6-274
6.5	Free contour programming.....	6-275
6.5.1	General.....	6-275
6.5.2	Program contour.....	6-276
6.5.3	Contour elements	6-280
6.5.4	Graphic representation of the contour.....	6-281

6.5.5	Input screens for setting parameters for contour elements	6-282
6.5.6	Specifying contour elements in polar coordinates, closing the contour	6-285
6.5.7	Undercuts in turning technology	6-290
6.5.8	Help	6-293
6.5.9	Parameter description of straight line/circle contour elements	6-294
6.5.10	Examples of user-defined contour programming	6-295
6.5.11	Cycle support	6-298
6.6	Display in the step editor (option from SW 6.4)	6-299
6.7	Program relationship in the dual editor (from SW 6.4)	6-302
6.7.1	Views in the dual ASCII editor	6-302
6.7.2	Synchronized view in the dual ASCII editor	6-302
6.7.3	Synchronized display in the dual-step editor	6-303
6.8	Multi-channel step display (option from SW 6.4)	6-304
6.9	Program simulation	6-311
6.9.1	Simulation operation	6-312
6.9.2	Simulation settings	6-323
6.9.3	Setting downtimes	6-328
6.9.4	Display and colors	6-329
6.9.5	Section by section simulation	6-330
6.9.6	Multi-channel, section by section program simulation	6-331
6.9.7	Simulation with orientable toolholder	6-334
6.9.8	Quick display in the simulation for mold making	6-335
6.9.9	Simulation with external network drive	6-337
6.10	Manage programs	6-338
6.10.1	Overview	6-338
6.10.2	NC data types and directories	6-339
6.10.3	New workpiece/part program	6-340
6.10.4	Saving setup data	6-343
6.10.5	Select program for execution	6-344
6.10.6	Load/unload program	6-347
6.10.7	Load/unload program	6-348
6.10.8	Copy/paste	6-349
6.10.9	Delete	6-352
6.10.10	Rename	6-353
6.10.11	Enabling	6-355
6.10.12	Log	6-356
6.11	Accessing an external network drive/computer	6-357
"Services" Operating Area		7-359
7.1	Function	7-360
7.2	Directory structure	7-360
7.2.1	NC active data	7-360
7.2.2	Hard disk	7-361
7.2.3	Directories	7-363

7.2.4	Data selection.....	7-364
7.3	Formats for saving and importing data.....	7-367
7.3.1	Punched tape format.....	7-368
7.3.2	PC format	7-371
7.4	RS-232 interface parameters	7-372
7.4.1	Interface parameters	7-375
7.5	Operation.....	7-376
7.5.1	Services main screen	7-376
7.5.2	Setting the RS-232 interface	7-379
7.5.3	Reading in data	7-383
7.5.4	Reading out data	7-384
7.5.5	Log.....	7-386
7.6	Manage data.....	7-388
7.6.1	Standardized program and data management (from SW 6.3).....	7-388
7.6.2	Creating a new file/directory	7-389
7.6.3	Loading/unloading	7-389
7.6.4	Copy/paste	7-390
7.6.5	Deleting	7-392
7.6.6	Changing the properties of a file/drive/archive	7-393
7.6.7	Defining and activating user data (GUD).....	7-395
7.7	Start-up functions	7-397
7.7.1	Series start-up	7-397
7.7.2	Restoring the original state via NC card	7-399
7.7.3	Upgrade.....	7-400
"Diagnosis" Operating Area		8-403
8.1	Diagnostics main screen	8-404
8.2	Alarms/messages/alarm log	8-406
8.3	Service display	8-408
8.3.1	Service overview (from SW 6).....	8-408
8.3.2	Service axis	8-410
8.3.3	Service drive.....	8-411
8.3.4	Service Safety Integrated	8-412
8.3.5	Configuration data	8-416
8.3.6	Communication error log	8-416
8.3.7	Action log.....	8-417
8.3.8	Version	8-417
8.3.9	Sorting and saving version information	8-418
8.3.10	Displaying the version screen for cycles (from SW 6.3).....	8-419
8.3.11	Display cycle versions (from SW 6.3).....	8-421
8.3.12	Displaying loadable compile cycles (from SW 6.3)	8-422
8.4	PLC status	8-424
8.4.1	General.....	8-424
8.4.2	Changing/deleting values	8-425

8.4.3	Symbolic PLC addresses	8-427
8.5	Selecting/creating operand screens for PLC status	8-432
8.5.1	File functions	8-433
8.6	Displaying NC system resources	8-434
"Startup" Operating Area		9-435
9.1	Machine data	9-441
9.1.1	Display options: suppression filter	9-443
9.2	User views	9-445
9.3	NC	9-447
9.4	PLC	9-447
9.4.1	PLC status	9-447
9.4.2	Set time/clock	9-448
9.4.3	File functions	9-448
9.5	Drives/servo	9-449
9.6	HMI	9-451
9.6.1	Changing the HMI interface	9-451
9.6.2	System settings	9-454
9.6.3	Transferring display MD from HMI Embedded	9-457
9.7	Tool management	9-458
Maintenance		10-459
10.1	Operating data	10-460
10.2	Cleaning	10-461
Appendix		A-463
A	Abbreviations	A-464
B	Terms	A-474
C	References	A-497
D	Index	I-511
E	Commands, identifiers	I-516

Introduction

1.1	The SINUMERIK 840D/810D system	1-22
1.2	Notes on handling	1-24
1.3	Activation/deactivation	1-25

1.1 The SINUMERIK 840D/810D system

General

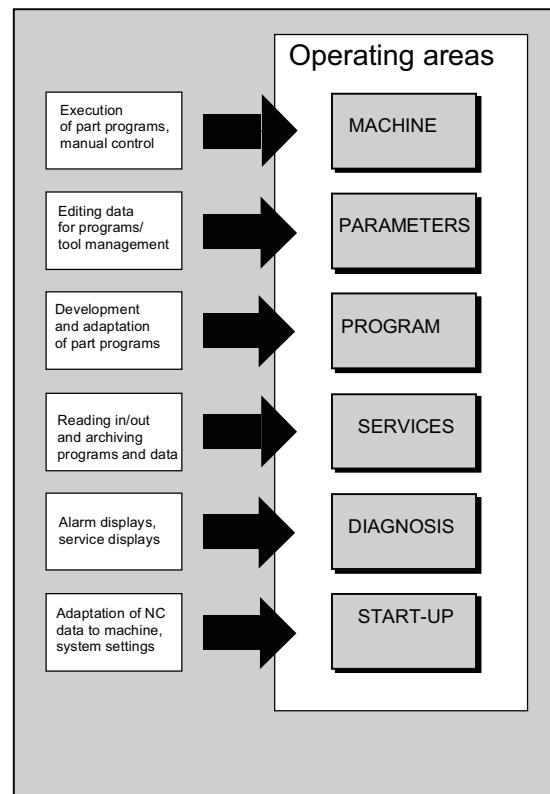
SINUMERIK 840D, 810D is a CNC control system (**C**omputerized **N**umerical **C**ontrol) for machine tools.

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

- Create and adapt part programs,
- Execute part programs
- Manual control,
- Read in/out part programs and data,
- Edit data for programs,
- Display and troubleshoot alarms,
- Edit machine data,
- Set up 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 components such as screen, LEDs
- Control elements such as keys, switches, handwheels

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 Notes on handling**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 Activation/deactivation



Activation



Function

A variety of methods can be employed to switch on the power supply to the control system or to the whole plant.

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	0.000 %
			Power [%] <input type="text"/>
			Feedrate mm/min
			Act. 0.000 0.000 %
			Set 0.000
			Tool
			preselected tool :
			G0 G91

Deactivation



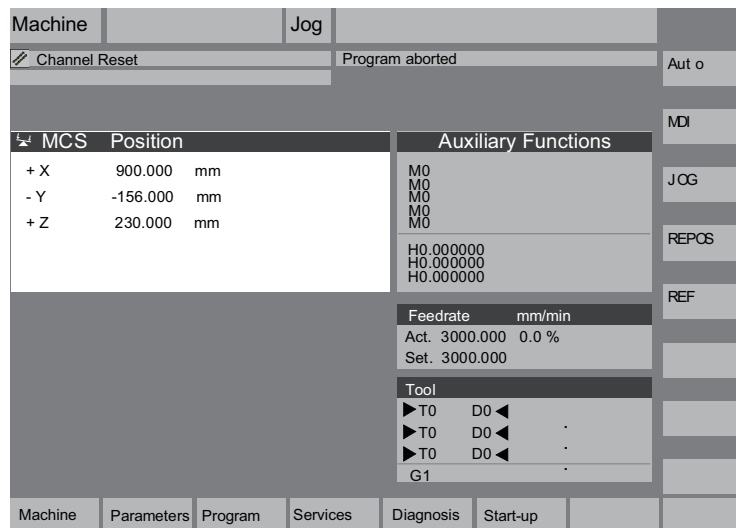
Please follow the instructions for switching off the control or the entire system!

Machine manufacturer

Please follow the machine manufacturer's instructions!

Operating sequence

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.



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/Sequence of Operations

2.1	Operator panels	2-29
2.1.1	OP 010 operator panels	2-29
2.1.2	OP 010S operator panel	2-30
2.1.3	OP 010C operator panel	2-30
2.1.4	OP 012 operator panel.....	2-31
2.1.5	OP 015 operator panel.....	2-31
2.1.6	Full standard keyboard	2-32
2.2	Operator panel keys.....	2-34
2.3	Machine control panels	2-39
2.4	Machine control panel keys	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.....	2-46
2.5	Screen layout	2-47
2.5.1	Overview	2-47
2.5.2	Global machine status display	2-48
2.5.3	Program control display	2-52
2.6	General sequence of operation.....	2-54
2.6.1	Program overview and program selection	2-54
2.6.2	Switch menu window.....	2-55
2.6.3	Select directory/file.....	2-56
2.6.4	Edit input/values.....	2-57
2.6.5	Confirm/cancel input	2-58
2.6.6	Editing part programs in the ASCII editor	2-59
2.6.7	Switch channel	2-65
2.6.8	m:n Communication links.....	2-66
2.6.9	Pocket calculator.....	2-70
2.7	Help	2-71
2.7.1	Editor help	2-73
2.7.2	Quick help for program commands.....	2-74
2.7.3	Extended help for program commands.....	2-77
2.8	Job list	2-78
2.8.1	Syntax description for job lists	2-80
2.8.2	Example of a job list with two-channel 1:1 connections.....	2-83
2.8.3	Example of a job list with multi-channel m:n connections	2-84
2.8.4	"Execute job list" operating sequence.....	2-85
2.8.5	Rename workpieces with job lists	2-87

2.8.6	Copy workpieces with job lists	2-88
2.8.7	Archive workpieces with job lists in case of m:n.....	2-88

2.1 Operator panels

2.1.1 OP 010 operator panels

A Display

B Alphanumeric keypad
Correction/cursor keys

1 Machine area key

2 Recall (return)

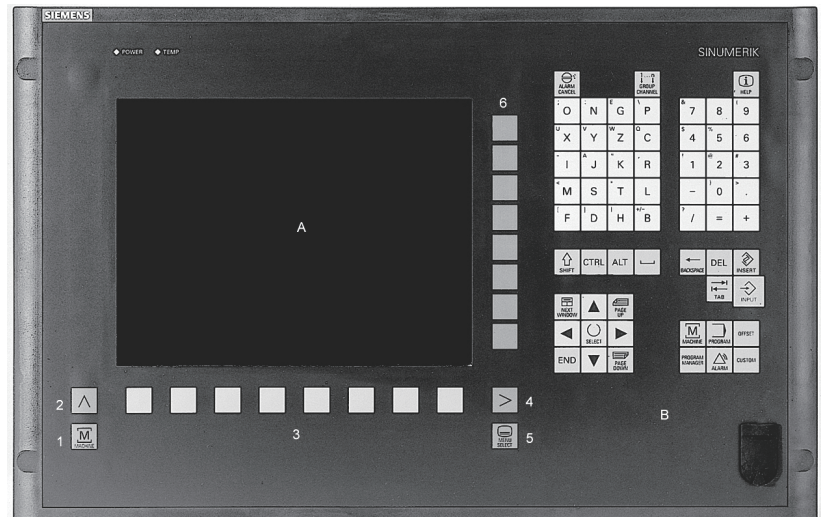
3 Softkey bar (horizontal)

4 ETC key

(menu extension)

5 Area switchover key

6 Softkey bar (vertical)



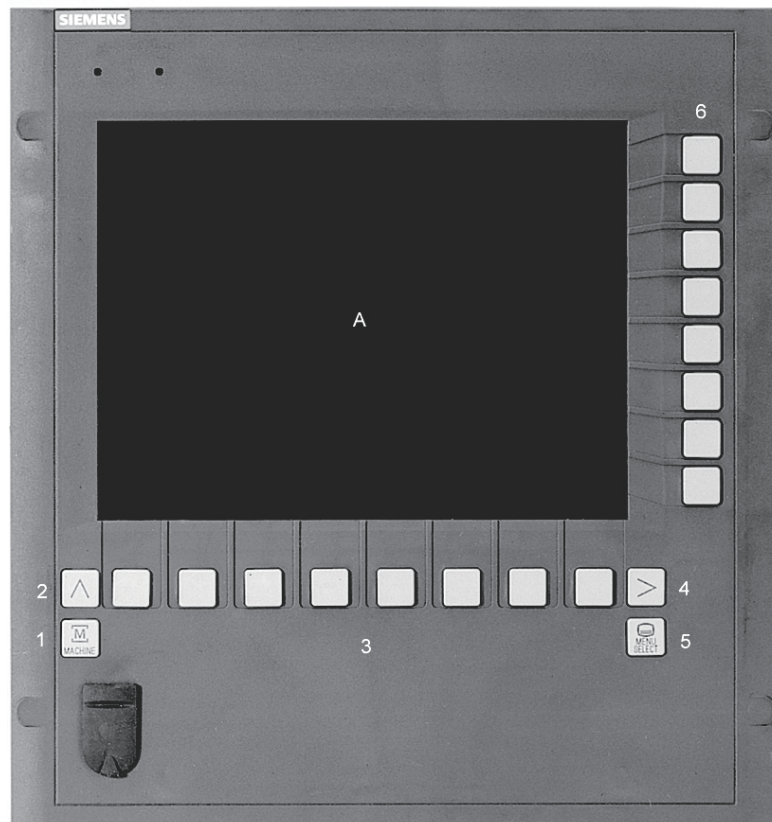
All keys are described in the following sections.

2.1 Operator panels

2.1.2 OP 010S operator panel

A Display

- 1 Machine area key
- 2 Recall (return)
- 3 Softkey bar (horizontal)
- 4 ETC key
(menu extension)
- 5 Area switchover key
- 6 Softkey bar (vertical)

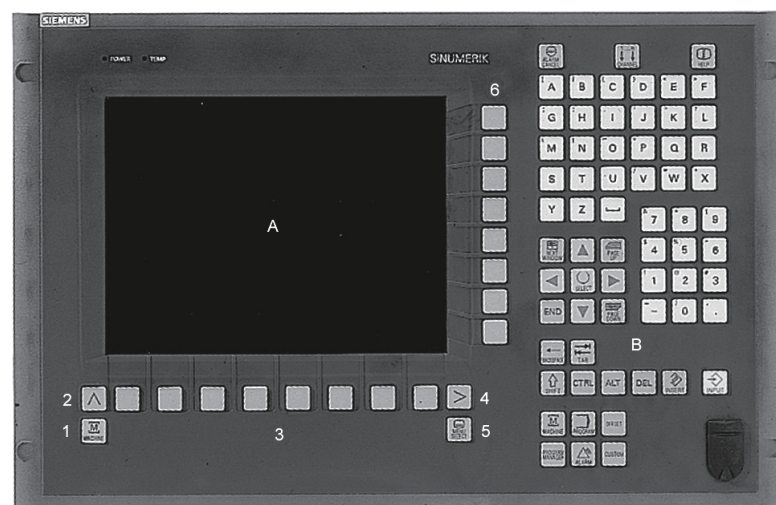


2.1.3 OP 010C operator panel

A Display

B Alphanumeric keypad Correction/cursor keys

- 1 Machine area key
- 2 Recall (return)
- 3 Softkey bar (horizontal)
- 4 ETC key
(menu extension)
- 5 Area switchover key
- 6 Softkey bar (vertical)



2.1.4 OP 012 operator panel

- A** Display
- B** Alphanumeric keypad
Correction/cursor keys
- C** Mouse and mouse keys
- 1** Machine area key
- 2** Recall (return)
- 3** Softkey bar (horizontal)
- 4** ETC key
(menu extension)
- 5** Area switchover key
- 6** Softkey bar (vertical)



2.1.5 OP 015 operator panel

- A** Display
- 1** Machine area key
- 2** Recall (return)
- 3** Softkey bar (horizontal)
- 4** ETC key
(menu extension)
- 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 in addition to this keyboard.

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	>	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	RESET	PROGRAM MANAGER	ALARM	CUSTOM	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 note

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 **OP010** and **OP010C**, are located on the MF II keyboard in the alpha/numeric keypad with NumLock off:

- | | |
|---------------|-----------------|
| • END | PROGRAM |
| • PAGE DOWN | OFFSET |
| • HOME (Pos1) | PROGRAM MANAGER |
| • PAGE UP | ALARM |

Hardkeys on the MF II keyboard

NumLock off

END

NumLock off

PAGE DOWN

NumLock off

HOME

NumLock off

PAGE UP

SHIFT F12

SHIFT F10

F10

Hotkeys on operator panels OP 010 and OP010C



Direct access
to the Program operating area



The Parameters operating area
is where the last tool offsets are directly called.



Direct access to Program overview
to show the last selected program management.



In the Diagnostics operating area, the last alarms,
messages, service displays or PLC status can be called.



Can be configured by the customer.



Direct access
to the Machine operating area.

Direct access to the last selected operating area:
Machine, Parameters, Program, Services, Diagnostics or
Startup

2.2 Operator panel 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

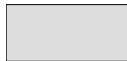
Direct branch to the "Machine" operating area.

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. Diagnostics
6. Startup

**Shift key**

Switches between functions on keys with double assignment.

**Channel switchover**

In a configuration with several channels, it is possible to switch between channels (switch from channel 1 through 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")

**Acknowledge alarm key**

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

**Information key**

This key displays explanatory information about the current operating status (e.g. support 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 down (PAGE DOWN)

You "page" down by one display.

In a part program, you can "page" the display down (to end of program) or **up (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

Space

Cursor to the left

Toggle key

- Selection key for values set in input fields and selection lists labeled with this key symbol.
- Activate/deactivate a field:

☒ = active

☐ = not active

☒ = 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 to the right

Edit key/Undo key

- Switch over to edit mode in tables and input fields (in this case the input field is in insert mode) or
- UNDO function on table elements and input fields (the value is not validated when you exit a field using the edit key; instead it is reset to the previous value = UNDO).



End of line key

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



Cursor down



Page up (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

Takes you directly to the tool offset



Program management

Program overview
A program can be opened in the text editor.



Alarm

Takes you directly to the Alarms screen



User key

Can be configured by the customer

**SW 6.3 and higher****Note**

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

PROGRAM hardkey

Press this hardkey in any operating area to open and display the part 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 part 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

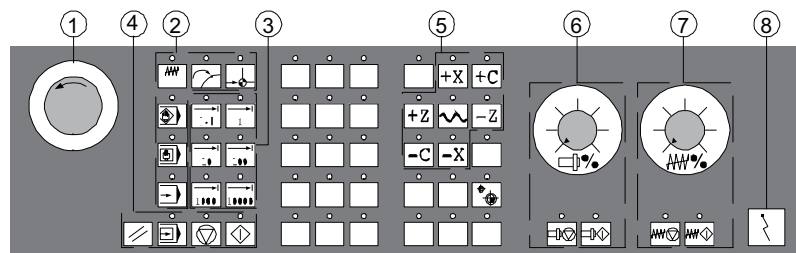
Actions on the machine tool, for example traversing the axes or program start, can only be initiated via a machine control panel.

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.

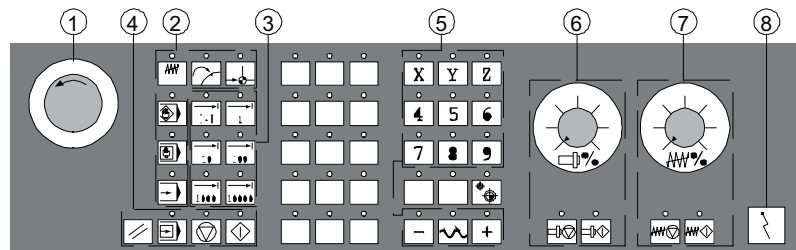
The following description applies to the 19" machine control panel supplied by SIEMENS. If you are using another machine control panel, please consult the operating instructions of the machine-tool manufacturer.

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

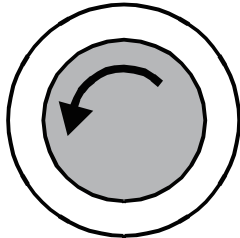
- 1 Emergency Stop button
- 2 Operating modes (with machine function)
- 3 Incremental mode
- 4 Program control
- 5 Direction key with rapid traverse override
- 6 Spindle control
- 7 Feed control
- 8 Keyswitch



Machine control panel for turning machines



Machine control panel for milling machines

2.4 Machine control panel keys**2.4.1 Emergency STOP****EMERGENCY STOP key**

Press this red key in emergency situations:

1. If life is at risk,
2. If there is a risk of damage to the machine or the workpiece.

An EMERGENCY STOP generally shuts down all drives with the greatest possible braking torque in a controlled manner.

For details of other or additional reactions to an EMERGENCY STOP: Please read the information supplied by 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 travel in jog mode via:

- 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:

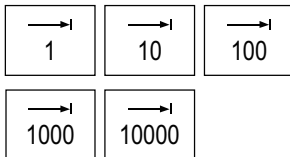
- "JOG" operating mode
- "MDI/Teach In" operating mode



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 increment value is evaluated according to the setting in the machine data.

Machine functions



Teach In

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



Repos

Repositioning

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

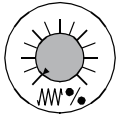


Ref

Approaching a reference point

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

2.4.3 Feed control

**Rapid feed 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 brought to a standstill under control,
- 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)

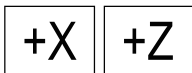
Example:

- In "MDI" mode an error is detected during execution of a block.
- A tool change is to be carried out.

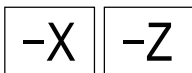
**Feed start**

If you press the "Feed start" key:

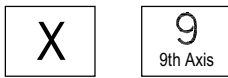
- The part program is continued in the current block,
- The feedrate is accelerated to the value specified by the program,
- The associated LED lights up as soon as feed start has been accepted by the control.

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 negative direction.



Machine manufacturer

Axis keys (for milling machines):

Select the axis (X ... 9) for traversal

in positive direction by pressing the "+" key or

in negative direction by pressing the "-" key.

Rapid traverse override

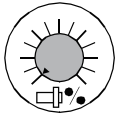
If you press this key together with key "+" or "-", the axis moves in rapid traverse mode.

- 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 (WCS)/(MCS) 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 reached.

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-tool 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-tool manufacturer).

2.4.5 Keyswitch

SIEMENS keyswitch



Machine manufacturer

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

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:

Key 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 authorization

Highest
access authorization



Changing the access authorization

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.

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 840 D or

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



2.4.6 Program control**NC Start**

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

**NC Stop**

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

After this, you can continue processing with NC start.

**Single block**

This function allows you to execute a part 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 execution is active,

- Stop is displayed on the screen in the cycle (in the program control line),
- The message "Stop: Block completed in single block" is displayed in the channel operational message line (in interrupt state).
- The current block in the part program will not be executed until you press the "NC Start" key,
- Program processing is stopped after one block is executed,
- You can execute the next block 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:

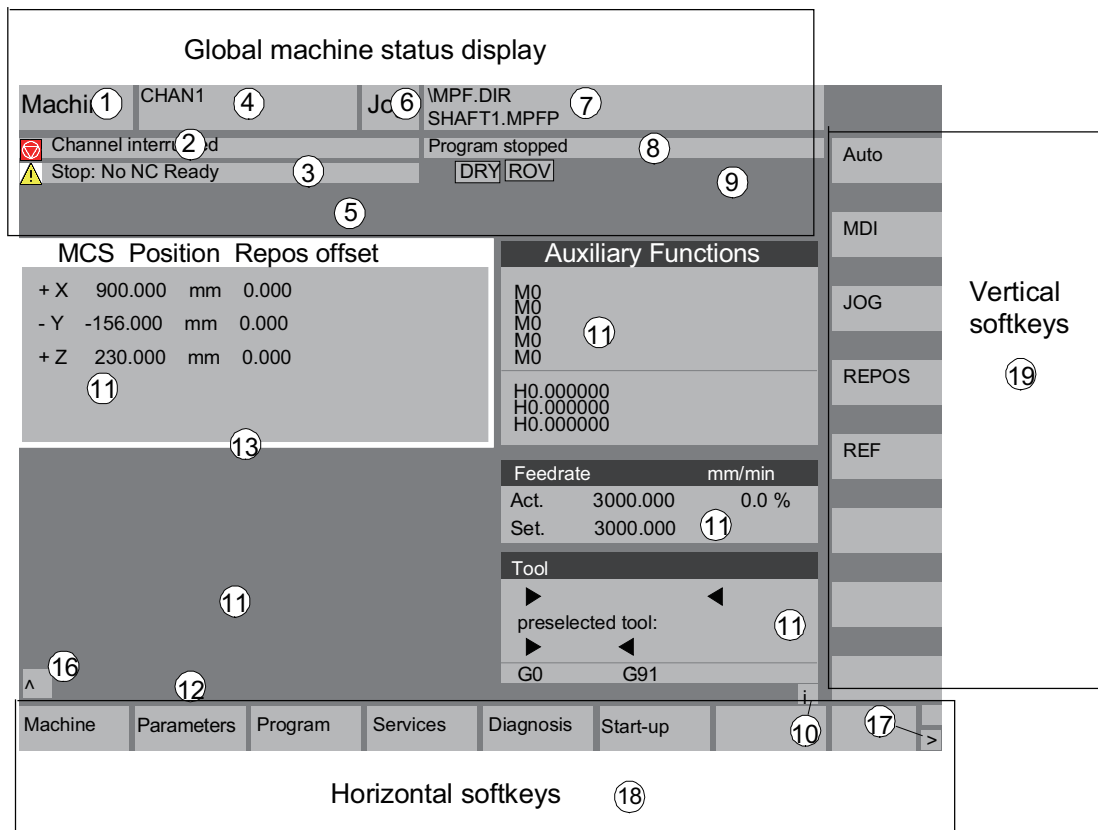
- Execution of the current part program is aborted.
- Messages issued by the monitoring function are deleted (except the alarms POWER ON, NC Start and "Acknowledge alarm").
- The channel is set to "Reset" state, i.e.,
 - The NC control remains synchronized with the machine.
 - The control is in its initial state and ready for a new 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 the selected program
- 8 Program status
- 9 Program control
- 10 Additional information (help)
 - i You can display information by pressing the i key
 - ^ Recall: Return to higher-level menu
 - > ETC.: Expansion of the softkey bar in the same menu.

2.5 Screen layout

11 Work pane, NC code and operating data

The work pane (program editor) and NC data (feedrate, tool) available in the selected operating area are displayed here.

From software Version 6.2

The unit for positional data is preceded by the diameter symbol \varnothing in the work pane if the axis is currently the transverse axis and if the tool coordinate system is set. If diameter programming is disabled with DIAMOF, the symbol preceding the unit is no longer visible.

12 Dialog line with operator information

Here operator information is displayed for the selected function (if available).

13 Focus

The selected pane is highlighted by a frame. The pane title is displayed in reverse highlighting. 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/vertical softkey bar (corresponding to F1 to F8 on the full keyboard).

Other Information

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

From **SW 6.2**: For displaying the channel operational messages with symbols, see Subsection 4.1.2.



If the status shows this symbol , operating input is required.

-  1 Stop: No NC ready
- 2 Stop: Mode group ready
- 3 Stop: Emergency Stop active
- 4 Stop: Alarm active with stop
- 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: Feed 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.

If the status shows this symbol , operation is generally not required.

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

2.5 Screen layout

	29 Wait:	For punching
	30 Wait:	For safe operation
	32 Stop:	Oscillation 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 slave axis
	36 Wait:	AXCT axis active as master axis
	37 Wait:	AXCT axis changing to follow-up
	38 Wait:	AXCT internal status change in axis
	39 Wait:	AXCT axis drive disable
	40 Wait:	AXCT axis overlaid motion active
	41 Wait:	AXCT axis axis replacement active
	42 Wait:	AXCT axis interpolator active
	43 Wait:	WAIT_FOR_CC_ENABLE: Waiting 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 data match (New-Config)
	49 Wait:	For axis replacement: Axis currently coupled
	50 Wait:	For axis replacement: Liffast 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 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 that was programmed in the part program using the MSG command (if no alarms pending)
6 Operating mode display	<p>The currently selected mode JOG, MDI or Auto (Automatic) is displayed.</p> <p>From software Version 6.2:</p> <p>The active submode is displayed next to the operating mode. An active increment is also displayed below. For example:</p> <p>JOG Repos 1000</p>
7 Program name	<p>Press NC Start to execute this program.</p> <p>From software Version 6.2:</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>From software Version 6.2:</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	<p>Functions that have been activated are visible can be set via "Program control", see Section 4.6 Automatic mode, "Program control"</p>

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 ...").

From SW 6.4 you can hide up to 10 program levels (e.g. "/6N100 .."; the 7th program level is hidden)

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 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. Part program blocks are

- Traversing blocks
- Switching and auxiliary functions
- Blocks generated within 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

Traverse movements are performed with the feedrate value set in the setting data "Dry run feedrate".

This dry run feedrate function replaces the programmed motion commands.

ROV

Rapid traverse override

The override switch for the feedrate also applies to the rapid traverse feedrate.

M01

Programmed stop 1

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. Press the NC Start key to resume execution. If the function is not activated, special function M01 (from the part program) is **not** taken into account.

Additional M fct.

Programmed stop 2

As of SW 6.3:

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.

References: /FB1/ Description of Functions Basic Machine, Predefined Auxiliary Function "Associated auxiliary functions for M0, M1"

DRF

DRF selection

When the "DRF" function is active, DRF offsets are applied.

PRT

Program test

In program test mode, setpoint outputs to the axes and spindles are disabled. The setpoint display "simulates" the traverse movements.

FST

Feed stop

The activated feed stop is displayed.

As of SW 6.2:

"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 set/deactivated under Program Control, but via the Feed Start/Feed Stop keys on the machine control panel.

2.6 General sequence of operation**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.

**Other Information**

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/Parameters, see
User-specific operator 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.

**Operating sequence**

"AUTO" mode 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.

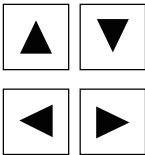
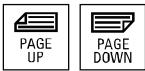


Program
selection

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 then loaded.

2.6.2 Switch 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 Select 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**

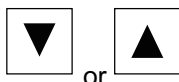
This key is used to select a file. It is possible to select multiple files. The symbol appears next to the cursor bar when a file is selected.

**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 Edit input/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 "Select" key to activate or deactivate a selection field.

Multiple-selection boxes

(you can select several options or none)

Single-selection radio buttons

(only one option can be active at a time)

☒ = active

☐ = not active

☒ = 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 lists

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.

2.6 General sequence of operation



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).

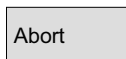
- The editor displays only the characters which can be input via the operator panel front keyboard.
- A part program opened by the editor cannot be simultaneously started in the NC (enable is canceled); an alarm (14014) is indicated in this case. If the control is switched off while the editor is open,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

2.6.5 Confirm/cancel 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 call menu level.



Cancel input:

Your inputs are rejected if you select the "Abort" softkey. 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 "Insert" key can also have 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 part programs 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/replace text
- Save file
- Create contour (programming support)
- Configure cycle parameters (drilling, milling, turning)
- Start simulation
- Recompile (cycles, free contour programming)
- Renumber blocks
- Change settings
- Open 2nd file.

Additional information

A part program selected in the NC can generally only be edited when the channel is in the reset state.

When a part program is selected and the relevant channel 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 not displayed as " L_F "; instead it is displayed as "¶".

Operating sequence

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 "Input" 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 "Delete" key to delete the character to the left of the cursor.

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

Vertical softkeys**Overwrite**

The softkey toggles between insert and overwrite modes.

Mark 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.

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.

It remains stored in the buffer even if another part program is selected.

The selected block is deleted.

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

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

Paste block

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

Over-
write

Mark
block

Copy
block

Delete
block

Mark
block

Paste
block

Find/
Go to...

Program
start

Program
end

Go to...

Find

OK

Abort

Find
next



or

Replace



or

Replace
all texts



Find/Go to...

The Find/Go to... dialog box opens.

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

The following search methods are available:

- to the beginning of the part program (cursor on the first character in the program),
- the end of the part program (cursor on the last character in the program) and
- to position on a particular NC block with "Go to.."
- or search for a particular character string with "Find".

"Go to..." "...": Enter the required block number.

- If the line being searched for contains an "N" or ":", you are taken to that line.
- A message is output if there is no 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.

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).

2.6 General sequence of operation



Abort

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

Save
file

Save file

Changes are saved in the file loaded in the editor.

Other Information

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

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.

Contour

Generate
contour

Horizontal softkeys

Free contour programming

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

Accept
contour

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

Setting parameters for cycles

The following functions are provided as programming support:

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

Drilling

Milling

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

Turning

Enter the new values for the cycle parameters.

OK

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

Example:

CYCLE81 (110, 100, 2, 35)

/PGC/, Programming Guide Cycles

Simulation

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.



Recompile

Recompile

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.

OK

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

Renumber

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

Settings

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)

2.6 General sequence of operation



Setting
contour

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").

Other information

- 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



Channel status



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.

The following three channel states can occur in each mode:

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-tool 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 running 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**

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 change to the channel menu in all operating areas by activating the channel switchover key. The only change is to the horizontal and vertical softkeys.

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.

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

Machine		Jog					
✎ Channel Reset		Program aborted			Channel 11		
					Channel 12		
MCS Position		Auxiliary Functions			Channel 13		
+ X	900.000 mm	M0			Channel 14		
- Y	-156.000 mm	M0			Channel 15		
+ Z	230.000 mm	M0					
		M0					
		M0					
		M0					
		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					
M I L L 1	M I L L 2						

Activate the channel changeover key. The currently existing connection is displayed by means of the highlighted softkeys (horizontal, vertical) if the channel menu is active.

Channel switchover

You can change to other channels by means of the vertically arranged softkeys.

Group switchover

Use the horizontal softkeys to change over to a different group; the channels of the current group are now displayed on the vertical softkeys. Switchover to another channel (and if necessary to another NC) only takes place upon activation of a vertical softkey.

NC switchover

You can change to another NC via the vertical softkeys if the channel is not on the current NC.



Other information

- 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.

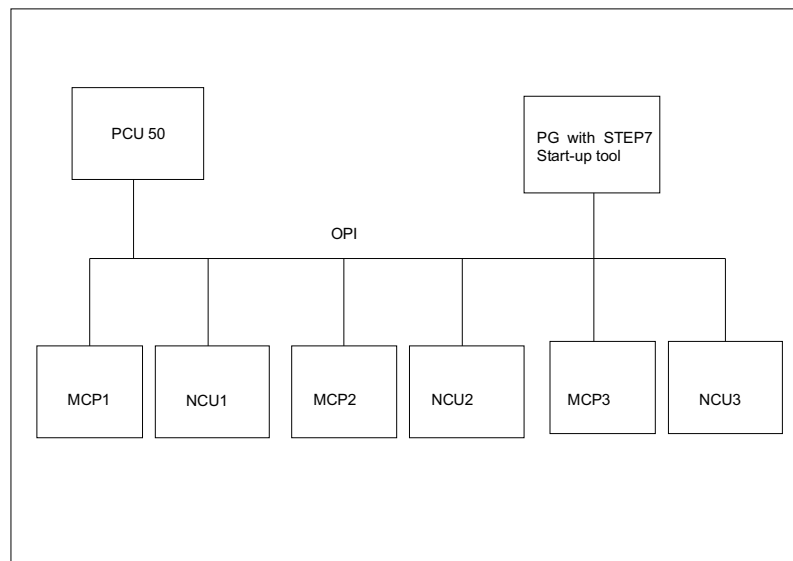
Example: 1 HMI Advanced and 3 NCUs

One HMI unit can be linked to several NCU/PLC units. The machine control panel MCP is permanently assigned to the NCU.

You can connect an additional programming device (PG) with the STEP 7 Start-up Tool.

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

- Multiple autonomous machines with multiple NCUs or
- One large machine with multiple NCUs.



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

- The NCU being operated is selected via the channel switchover key and the channel menu.
- The highlighted softkeys in the channel menu show which mode group/NC/channel the HMI is currently connected.

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

Requirement:

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



Equals key

Change to **pocket calculator mode**.

If you enter a basic arithmetic symbol (+, -, /, *) followed by a value (e.g. 13.5) in this mode and



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

The editor is in insert mode if the input/output field is opened with the input or equals key; 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" is displayed in the dialog line, you can press



the information key to display additional information.

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
down

or

Page
up

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 ...".

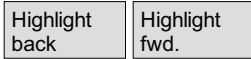
With the softkeys "Zoom +" and "Zoom -" you can increase or decrease the zoom factor in the document view,

with "Exit help" you return to the editor.

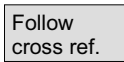


Analogous to the Windows Help, help is provided in some cases by the following softkeys:

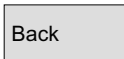
Press the "Page down" or "Page up" softkey.
The screen contents are scrolled up or down.



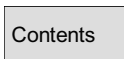
Use the "Highlight fwd." or "Highlight back" softkeys to go to the entry of your choice.



Press the "Follow cross ref." softkey.
The entry you chose appears on the screen.



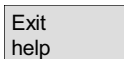
Press the "Back" softkey to return to the entry displayed beforehand.



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.



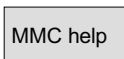
Context-independent Help call:



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 part programs in the editor via the "information" key:

- **Short help for programming commands**
Configuring see /IAM/ HE1, Help in Editor.
 - Help in part 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
- **Quick help "Parameterization screen" + extended help "pdf"**
Configurable parameterization forms from which you can jump to a particular page in the documentation (pdf file), e.g. parameterization screen for cycles, you jump to the Configuring Guide Cycles. For configuring instructions 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 Quick help for program commands



Function

You can call up a help function to provide programming support with the editing of part 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.

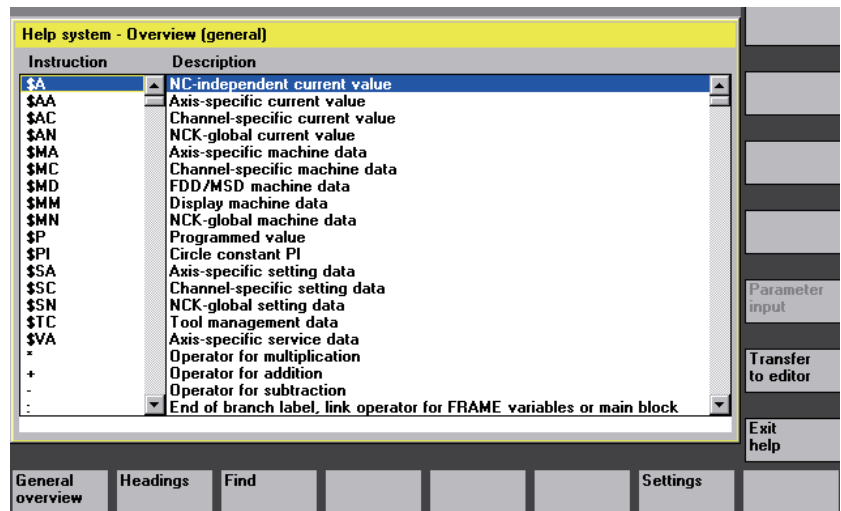
Operating sequence

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

Exit
help

General
overview

Headings



Show
heading

If transfer is possible, the instruction selected in the overview is inserted directly in the part 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 part program behind the instruction marked by the cursor.

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 part program writing screen.

Apart from context-sensitive help, it is also possible to search independently of context for instructions, descriptive texts or topics. With "General overview" a full overview of the instructions and relevant descriptive text stored in the help system is displayed.

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

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 "Show heading", the instructions belonging to the selected heading are displayed.


 Search


 Start
search

With the softkey "Search" 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).


 Settings


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 program commands



Function

You can call up a help function (short help) to provide programming support for editing part 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.

Operating sequence

Requirement:

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

You call the help function 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 part programs (in several channels if necessary):

- Parallel setup (LOAD/COPY) i.e.:
Load or copy main programs, subroutines and associated data such as
 - Initialization programs (INI)
 - R parameters (RPA),
 - User data (GUD),
 - Work offset (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 NC user memory.
- Preparations for the NC Start (SELECT) i.e.:
Select programs in different channels as well as starting preparations for execution
- Parallel clearing (reversed LOAD/COPY) i.e.:
Remove/unload main programs, subroutines and associated data from the NC working memory to the hard disk of the HMI
- Backup (in preparation for the next software version)

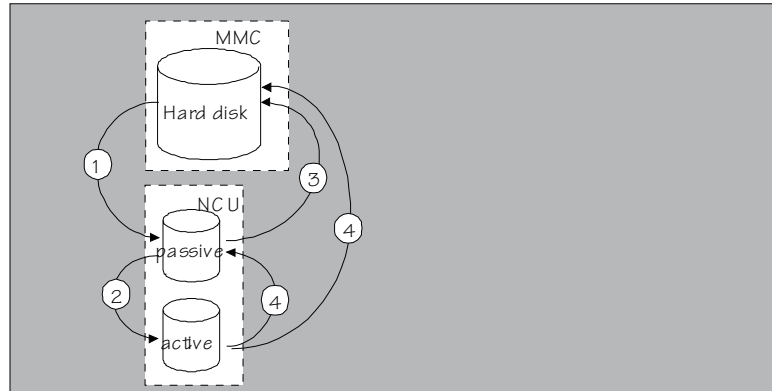
Notes

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. "Backup" (in preparation for the next software version)



Create the 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 part 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. Job list templates can be stored according to language and system.

See Programming operating area: 6.1.5 Templates.

2.8.1 Syntax description 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 the workpiece directory of the job list)

LOAD /MPF.DIR/*

(loads all files from one directory, here e.g. all files from part programs (MPF.DIR))

LOAD PART1.MPF

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

LOAD /SPF.DIR/PART1.SPF

(loads one file from one directory, here from the
subroutine 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, corresponds to the channel number)

CH=Station5

(Station5, corresponds to channel name from

NETNAMES.INI)

CH=ncu_b,1

(ncu_b, corresponds to the NCU name from NETNAMES.INI

1, corresponds to 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

(Part program PAGE1.MPF of workpiece SHAFT1.WPD
is executed from 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 it is only possible to specify the NC name with NC=.

CG= Name of the channel group, i.e. copy to each channel of this group (therefore in all NCs the channels are assigned to).
Parameter CG can be specified only if a channel menu is configured.

CH= Channel name

Channel names are only uniquely assigned across all NCs if 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 one file from the workpiece of the job list to the NC "NCU_2")

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

(Copies all files with one name from one directory e.g. PART1.* from subroutines (SPF.DIR) to one channel group, i.e. in all NCs which have channels of this group assigned to them)

COPY /MPF.DIR/* CH=CHANNEL22

(Copies all files from one directory, e.g. all files from part programs (MPF.DIR) to the NC which is assigned this channel.)

2.8.2 Example of a job list with two-channel 1:1 connections



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 with multi-channel m:n connections



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 /WORK.DIR/Part1.WPD/WpdGen.MPF NC=NCU1
    (or CH=CHANNEL1)
COPY /WORK.DIR/Part1.WPD/Channel1.MPF NC=NCU1
    (or CH=CHANNEL1)
COPY /WORK.DIR/Part1.WPD/Channel1.INI NC=NCU1
    (or CH=CHANNEL1)
COPY /WORK.DIR/Part1.WPD/Ch12.MPF NC=NCU1
    (or CH=CHANNEL1)
```

```
COPY /WORK.DIR/Part1.WPD/Channel2.MPF NC=NCU1
    (or CH=CHANNEL2)
COPY /WORK.DIR/Part1.WPD/Channel2.INI NC=NCU1
    (or CH=CHANNEL2)
COPY /WORK.DIR/Part1.WPD/Ch22.MPF NC=NCU1
    (or CH=CHANNEL2)
```

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

```
COPY /WORK.DIR/Part1.WPD/Channel3.MPF NC=NCU2
    (or CH=CHANNEL3)
COPY /WORK.DIR/Part1.WPD/Channel3.INI NC=NCU2
    (or CH=CHANNEL3)
COPY /WORK.DIR/Part1.WPD/Ch32.MPF NC=NCU2
    (or CH=CHANNEL3)
```

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

2.8.4 "Execute job list" operating sequence



Manage
data

Load

Selection

Unload



Load

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

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

- "Load"
- "Selection"
- "Unload"

from.

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.

Other Information

"Load" job list

"Load" means that all the prepared job list instructions are executed.

The data is distributed from [Source] to [Destination], to the destination 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!".

Selection**"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 which has been loaded to a destination NC with a COPY instruction is deleted in the [destination] if the time stamps are identical. If the file on the NC was changed, the user is prompted whether the version from the NC is to be saved 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



Manage
programs



Rename

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** is renamed **B.WPD**:

All files with the name **A.XXX** are renamed to **B.XXX**, i.e. the extension is retained.

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 **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
```

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

the files are not renamed.

Operating sequence

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.

2.8.6 Copy workpieces with job lists

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



Manage
programs



Copy

Insert

OK

Operating sequence

The softkey "Manage programs" in the "Program" operating area 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.

Press the softkey "Insert", specify another name if required

and confirm with "OK".

2.8.7 Archive workpieces with job lists in case of 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 joblists are executed and archiving is then started.



Data
from

Operating sequence

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.



Example of Operation

3.1 Typical sequence of operation

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 part 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 part program or read one in via an external data interface	2.6.6 7.5.3
	• Select a part program	6.10.5
	• Test a program (without a tool) <ul style="list-style-type: none"> • Start a part program (e.g. in single block) • Edit part 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 part program	6.9.5
Machine the workpiece	• Insert tool and run machining program	4.2.1
Store the program	• Save a part program <ul style="list-style-type: none"> • To hard disk • Read in/out via RS-232 interface 	6.10 6.2 7.5.3/7.5.4



3.1 Typical sequence of operation

Notes

Machine Operating Area

4.1	Data structure of the NC control.....	4-93
4.1.1	Operating modes and machine functions.....	4-94
4.1.2	Mode groups and channels.....	4-96
4.1.3	Channel-wide status display with symbols (SW 6.2 and higher)	4-97
4.1.4	Dual-channel display (SW 6.4 and higher).....	4-98
4.1.5	Operating mode selection/change	4-99
4.2	General functions and displays	4-102
4.2.1	Stop/start/cancel/continue part program	4-102
4.2.2	Display program level.....	4-103
4.2.3	Toggling between Machine/Work (MCS/WCS).....	4-104
4.2.4	Display axis feeds	4-106
4.2.5	Display G functions, transformations and swivel data	4-106
4.2.6	Display auxiliary functions.....	4-107
4.2.7	Display spindles	4-108
4.2.8	Handwheel	4-109
4.2.9	Status of synchronized actions	4-110
4.2.10	Preset	4-112
4.2.11	Setting the actual value.....	4-113
4.2.12	Inch/metric switchover.....	4-114
4.3	Reference point approach	4-116
4.4	JOG mode	4-119
4.4.1	Function and main screen.....	4-119
4.4.2	Traverse the axes.....	4-122
4.4.3	Inc: Incremental dimensions	4-123
4.4.4	Repos (repositioning)	4-124
4.4.5	SI (Safety Integrated): User agreement	4-125
4.4.6	Scratching (SW 6 or higher).....	4-126
4.4.7	Scratching (SW 5 and lower)	4-130
4.4.8	Display system frames	4-133
4.5	MDI mode.....	4-135
4.5.1	Function and main screen.....	4-135
4.5.2	Save program, file function.....	4-137
4.5.3	Teach-in.....	4-138
4.6	Automatic mode	4-140
4.6.1	Function and main screen.....	4-140
4.6.2	Program Overview.....	4-142
4.6.3	Loading and unloading the workpiece/part program.....	4-143
4.6.4	Log: Loading list of the programs.....	4-144
4.6.5	Executing programs from the hard disk	4-145
4.6.6	Access to external network drive	4-146
4.6.7	Program editing	4-148
4.6.8	Block search/setting the search destination.....	4-150

4.6.9	Accelerated external block search	4-153
4.6.10	Block search in program testing mode, multi-channel	4-156
4.6.11	Overstore	4-158
4.6.12	Program control	4-160
4.6.13	DRF offset.....	4-164

4.1 Data structure of the NC control



Environment

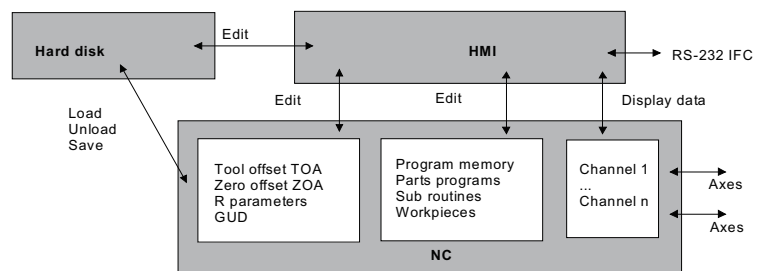
HMI

Function

The following components exist:

- NC with part 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
Part programs can be set up and processed block by block in this mode. The tested blocks can then be stored in the part 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
Part 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

or

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 documentation provided by the machine manufacturer!

**Machine functions****Machine manufacturer**

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

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)



Preparation for production

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 part program through position approach) by pressing the MCP key.

To start actual production some preparatory measures must be taken:

1. Set up the tools and workpieces,
2. Traverse the tools/workpiece to the start position specified in the setup plan,
3. Download the part program to the control memory,
4. Check/enter the work offsets,
5. Check/enter the tool offsets.

4.1.2 Mode groups and channels

**Function**

Every channel behaves like an independent NC. A maximum of one part 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.

BAG1	Channel 1 (processing)
	Channel 2 (transport)
BAG2	Channel 3 (processing)
	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).



4.1.3 Channel-wide status display with symbols (SW 6.2 and higher)



Function

The status of channels, feedrate and spindles are output with symbols in the program status line as well as the machine status if applicable - or the machine status only.

The machine manufacturer can customize the display; custom symbols can be displayed in addition to the standard symbols.

Machine manufacturer

Please follow the instructions of the machine-tool manufacturer. 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/controller stopped

Yellow Waiting for operator to take action

Green Machine/controller running

Gray Miscellaneous

Channel status

Channel interrupted

Channel active

Channel RESET

Feedrate status

Feedrate is not enabled

Spindle status

Spindle turning left or right

Spindle is not enabled

Spindle stop

Application example with 4 channels and 2 spindles:

Maschine	☐ CHAN1	AUTO	\WKS.DIR\DAUERTE_CYC950.WPD							
	◊ CHAN2		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 Dual-channel display (SW 6.4 and higher)

You can display channel information for 2 channels simultaneously with controls for which 2 or more channels have been configured.

Preconditions

- An M:N assignment is not active
- Two or more channels exist
- The display machine data is set for 2 channels

Representation

Two channels are represented side by side. One of these channels has the focus.

The 1st channel (left pane) is the channel which was explicitly selected in single-pane view. This selection can also be made outside the Machine operating area.

The right-hand pane displays:

- The associated channel configured in netnames.ini in section [MULTICHANNEL] or, if not available
- The subsequent channel configured in the same group in netnames.ini or, if not available
- The channel with the next highest number.

With the above mentioned conditions, dual-channel representation is immediately displayed when the Machine operating area is started.

Other information

Dual-channel representation is a **display** for viewing the channel information. Changes to NC channel data must still be made in the single-pane display. See switchover. As changes to the NC channel data are not entered via dual-channel display, there is no need for mode-specific softkeys to show the channels in the different operating modes in the left and right pane.

Dual-channel view

The division into two channel panes results in new boundary conditions for the information in each channel pane: In dual-channel view

- It is not possible to make entries in the relevant buffers in "MDI" and "Teach-in" modes.
You have to switch to single-channel view to make entries.
- Traversing movements of the axes are not accepted.
- The display type "program blocks" is disallowed if a part program is being executed from external in one of the displayed channels.

For selecting and changing operating modes, please refer to the following chapter.

Single-channel view

If "overstore" is active in single-channel view, the overstore function is explicitly exited when switching to dual-channel view if this is permitted by the NC. Otherwise, single-channel view is retained.

Channel switchover is effective for the pane which has the focus. If exactly two channels are configured, the channel windows do not switch over; only the focus changes.

Changeover

The M key will only switch between single-channel and dual-channel display in the current Machine operating area.

4.1.5 Operating mode selection/change**Mode change****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.

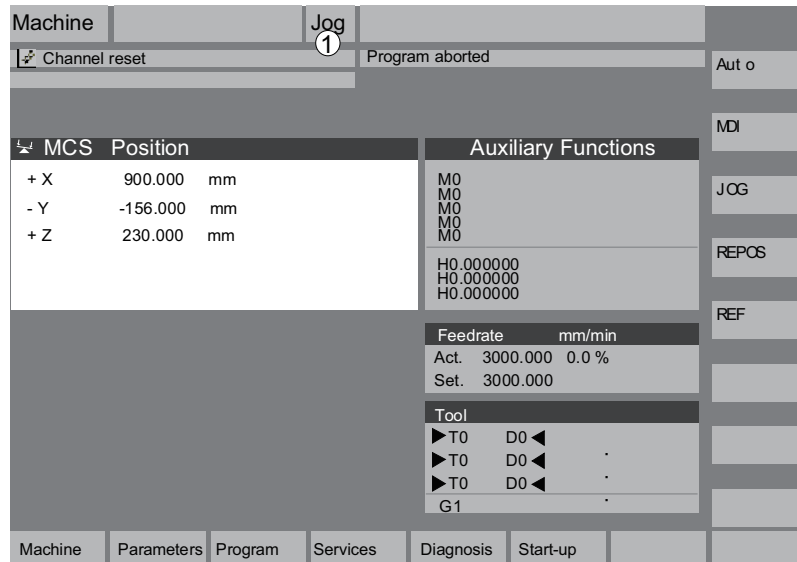
Not all mode changes are permitted.

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.

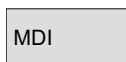
Operating sequence

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.



Other information

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-tool 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 Stop/start/cancel/continue part program



Function

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



Operating sequence

"AUTO" 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 part program:

The part program is started and executed.



Stop/abort part program:

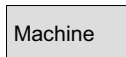
The program run is interrupted, but can be restarted with "NC Start".



The current program is aborted.

Resume part 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.

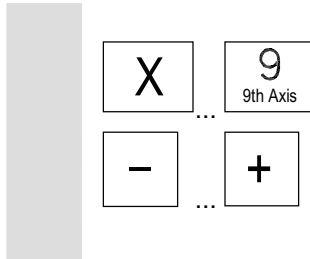


Press the "Repos" key to reposition the tool on the contour.



Turning machine:

Press the "+X" or "-X" 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

Current
block

Function

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

Operating sequence

"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".

During execution of a part 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 part program is displayed again.

4.2.3 Toggling between Machine/Work (MCS/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 part 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.

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.

Machine position	Repos offset
X	
Y	
Z	

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.

Work 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 = SZS display). Please consult the documentation provided by the machine manufacturer!



Operating sequence

"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 comprises all the physically existing 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 "WCS/MCS" key on the MCP.
- The number of places displayed after the decimal point and units of measure can be set in machine data.

/PA/, Programming Guide, Fundamentals



4.2.4 Display axis feeds



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.

Operating sequence

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

Press the "Axis feedrate" softkey:

- With "Machine" the feed window is displayed with the current feedrates and distance-to-go information as well as the associated override.
- With "Work" the feed window for the axes taking part in the interpolation is displayed with the current feed and the distance-to-go information with path override; for the remaining axes the current feed and distance-to-go information is displayed with single-axis override.

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

4.2.5 Display G functions, transformations and swivel data



G Fct.+
Transf.



or



Function

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

Operating sequence

"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.

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



References

Other information

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.

/PG/Programming Guide Fundamentals



Status display for swivel data (NCK SW 6.3 and higher)

Requirement:

The swivel function (CYCLE800) has been set up by the machine manufacturer.

The active swivel data record (orientable toolholder TOOLCARRIER) is displayed concurrently with the transformations.

The following values are displayed:

TCARR = 1 .. n* *n max. number of set up swivel data (TOOLCARRIER) that is active.

References

/PGZ/Programming Guide Cycles: Chapter 3 "Swiveling"

4.2.6 Display auxiliary functions



Display
Aux. fct.

Function

Auxiliary functions active in the selected channel can be displayed.

Operating sequence

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

The "Auxiliary functions" window appears on the screen.

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

4.2.7 Display spindles



or

**Function**

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

Operating sequence

"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.

Other information

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

SW 6.2 and higher:

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

- "Spindle Stop" 
- "Spindle not enabled" 
- "Spindle CCW direction of rotation" or "... CW" 
(=Spindle running)

4.2.8 Handwheel



Function

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

Operating sequence

"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-tool manufacturer is responsible for the design of handwheels. Operation may therefore differ from the explanation above.

Please refer to the machine-tool manufacturer's documentation!

4.2.9 Status of synchronized actions



Synchron.
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"

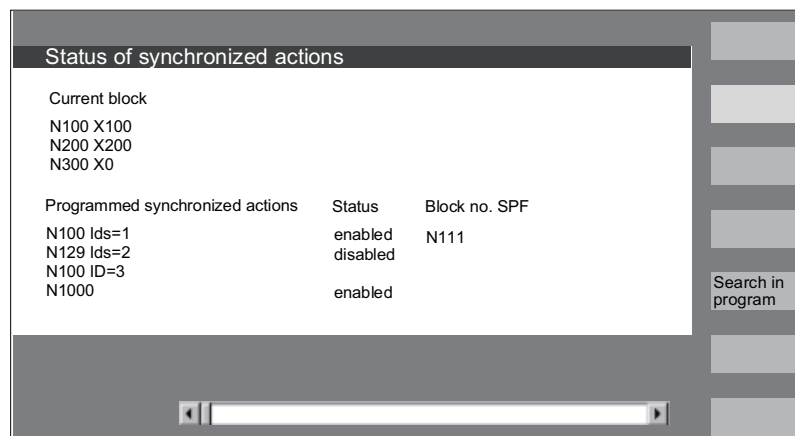
Operating sequence

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

Press the ETC key followed by

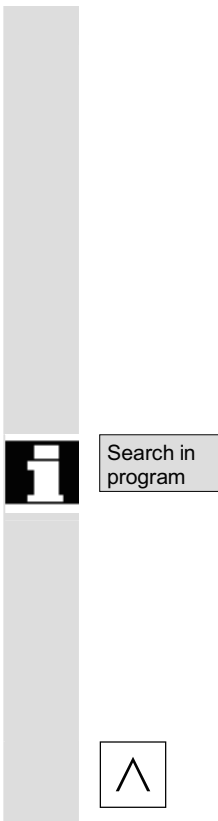
the "Synchron. actions" softkey.

The display "Status of synchronized actions" appears.



The display shows:

- **"Current block" column:**
the current extract of the selected 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)



- **"Status"** column
 - " " (no information given)
the condition is checked in the interpolation cycle.
 - "disabled"
LOCK was set from the PLC for the synchronized action
 - "active"
The action is currently being executed. 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".

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.

The preset values act on machine axes.

Axes do not move when "Preset" is active.

Machine manufacturer

See machine manufacturer's specifications.

Operating sequence

"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.

Other information**Machine manufacturer**

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

4.2.11 Setting the actual value



Set act.
value

1 9

...



Abort

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

See machine manufacturer's specifications.

For configuring references, please see
/IAM/ Installation and Start-Up Guide, Section HMI/MMC, IM4

Operating sequence

Requirement:

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.

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

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

4.2.12 Inch/metric switchover**Function**

The control system can operate with the inch or the metric system of measurement. You can switch between the inch and metric dimension systems 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 part program start or mode change are disabled for the duration of the switchover.



Switch to
metric

Operating sequence

"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 prerequisites for a switchover are not fulfilled on one of the NCUs linked, no switchover will take place on none of the NCUs. It is assumed that interpolations between several NCUs will take place on the existing NCUs whereby the interpolations can provide correct results only if the same unit systems are used.

References: /FB/, B3, Distributed Systems

4.3 Reference point approach



Ref Point

+X

-X

...

X

9
9th Axis

...

+

-

Function

The "Ref Point" 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 part 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!

Operating sequence

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

The "Ref Point" machine function is selected.

Turning machine:

Press the "Axis" keys.

Milling machine:

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





Machine manufacturer

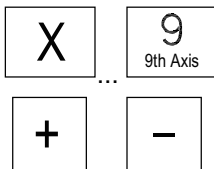
The selected axis moves to the reference point. The direction and sequence is defined by the machine-tool manufacturer in the PLC program.

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 is displayed for axes that need to be referenced.
-  This symbol is displayed next to the axis when the axis reaches 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-tool manufacturer).
- The feedrate override is active.

4.3 Reference point approach



Machine manufacturer

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



Other information

The sequence in which axes must be referenced can be defined by the machine-tool 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 main screen



Function

In manual mode you can:

1. Synchronize the control measuring system with the machine (reference point approach),
2. Set up the machine, i.e. activate manually controlled motions on the machine using the keys and handwheels provided on the machine control panel,
3. Activate manually controlled motions on the machine using the keys and handwheels provided on the machine control panel while a part program is interrupted.

The following main screen for "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			
Channel reset		Program aborted		G. fct. + transf.	
				Display aux. funct.	
MCS	Position	Repos offset	Master spindle	S1	Spindles
-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.		Axis feedrate
+V	0.000 mm	0.000	Power [%]		
		Feedrate mm/min			Zoom Act. val.
		Act. 7000.000 0.000%			
		Set 7000.000			
		Tool			Act. val. WCS
		Preselected tool:			
		G0 G91			
Preset		Scratch		Handwheel INC	

**Explanation of main
screen 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/Work (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 = SZS 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 machine (MCS) or work (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 screen** (if spindle is available)

The window contains the spindle setpoint and actual value, position,

setting of spindle override switch  and spindle output.

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

Feed**Feed 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. When G00 (rapid traverse movement) is programmed, the rapid override value is displayed.

As of SW 6.2:

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 section" and "Stroke rate" are represented inverted if you have programmed the automatic block division with "Path section length" or "Number of path sections".
- The "delay time" can only be displayed if you have set "Punching with delay time".

Zoom
act. val.

Increases the size of the actual-value display.

4.4.2 Traverse the axes

Traversing velocity

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

Traversing velocities are defined by the machine-tool manufacturer.

The default setting for the feedrate is mm/min.

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

Operating sequence

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

Traverse the 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.

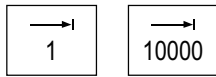
If you press "Rapid traverse override" at the same time, you can traverse the axis in rapid traverse mode.

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).





Other information

- When the control is switched on, the axes can be traversed right up to the limits of the machine as the reference points have not yet been approached and the axes referenced. Emergency limit switches might be triggered as a result.
- 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 dimensions



Function

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

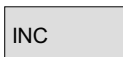


Operating sequence

"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.

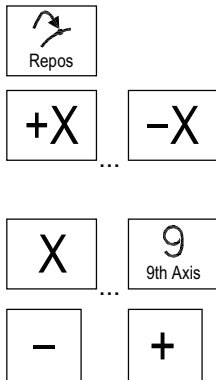
"Repos" offsets can be displayed in the machine coordinate system (MCS) or workpiece coordinate system (WCS).



Operating sequence

"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 agreement



Function

If the option "user agreement" 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 always refer to the machine (MCS).



Operating sequence

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

User
agreement

Press softkey "User agreement".

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.



Other information

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 higher)



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 provided for the "Scratching" function.

Operating sequence

1. Press softkey "Scratching":
Whether the softkey is available or not depends on the authorizations set in an MD.
 - The active plane is displayed and can be altered.
 - The active NV is displayed and can be changed.
 - The active tool is displayed. No tool is displayed if none is active (message).



Scratching

Machine	CHAN1	JOG	\MPF.DIR TEST_ULI.MPF		57 1523216
Channel reset					
Program aborted			ROV		
WCS		Position	Repos offset	Spindle	S2
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 0.000 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.

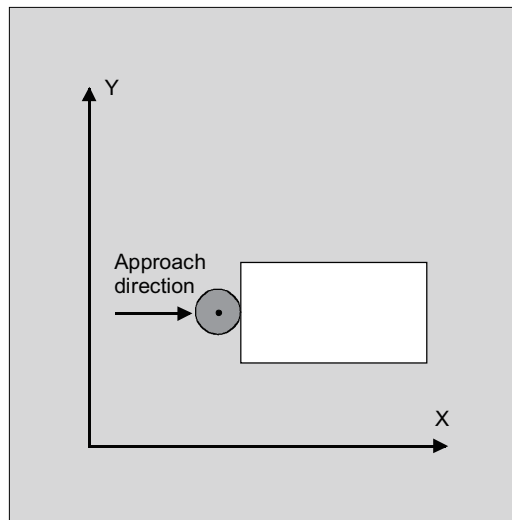
Repeat the process for other axes.

4. Press "OK" to transfer all values to the selected WO. The offset is always calculated in relation to the current work (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		
Channel reset			Program aborted		
Change active work offsets and compensations					
		X [mm]	Y [mm]	Z [mm]	
Set zero	Coarse	-20.000	-30.000	-40.000	
	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.

Tool holder

If a toolholder is active in the present system, it is displayed in the G function window as "TCARR=n" (n = 1 or 2).

Swivel
adjustment

Maschine	KANAL1	JOG	\SYF.DIR OSTORE1.SYF	
Kanal unterbrochen		Programm unterbrochen		
NC-Satz-Fehler / Anwenderfehler		ROV		
14155 Kanal 1 Satz ungültige Baseframedefinition fuer Werkzeugtraegeroffset				
WKS	Position	Repos-Versch.	Transformation + G-Funktionen	
X1	0.00 Ømm	0.00	TCARR = 1	
Z1	0.00 mm	0.00	01:G01 04:STARTFIFO	
CK1	0.00 grd	0.00	06:G18 08:G54	
CK3	0.00 grd	0.00	10:G60 12:G602	
XK3	0.00 mm	0.00		
Ankratzen				
Ebene	G18	Schneide	1	
Nullpunktverschiebung	G54		1	
Achse	Verschieb.	Sollposition	Anfahrriichtung	Werkzeugkorrektur
X1	0.00		→	- R 0 mm
Z1	0.00		→	- R 0 mm
CK1	0.00			Grad
CK3	0.00			Grad
XK3	0.00			mm
Schwenken: Abgleich nicht möglich				

With scratching in a swiveled level with rotary axes without encoder, the vertical softkey "Swivel adjustment" appears in addition to the toolholder information in the "Transformations + G functions" screen. Upon activation, the NC updates the toolholder data after scratching. If this operation is necessary, "Swivel adjustment completed" is output in the message line. If an error occurs, e.g. the channel is in "active" state or the connection with the NC is disrupted, "Swiveling: adjustment not possible" is displayed in the message line.

4.4.7 Scratching (SW 5 and lower)



Scratching



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 provided for the "Scratching" function.

Operating sequence

1. Press softkey "Scratching":

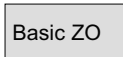
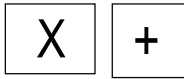
- The active plane is selected (G17).
- The active ZO is selected. If no ZO is currently active, then the first ZO (G54) is selected.
- The active tool is displayed. No tool is displayed if none is active (message).

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.

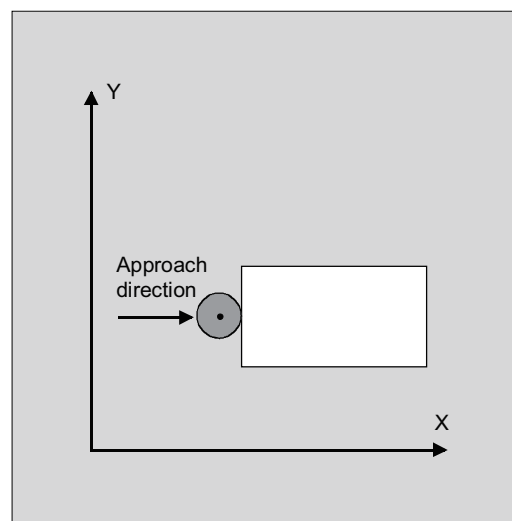


3. Move the axis up to the workpiece, enter the chosen setpoint position (e.g. "0") and press the "Input" key. The offset is then calculated.

Repeat the process for other axes.

4. Press "OK" to transfer all values to the selected WO. The offset is always calculated in relation to the current work (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.

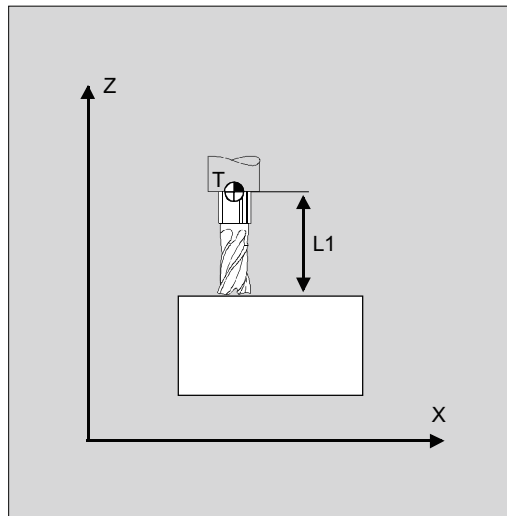
Example 1



Set "+R" with toggle key.

Example 2

Set "-L1" with toggle key.



There are three different variants depending on the system configuration:

- a) Tool selection with D No. only (flat D No.)
- b) Without tool management
- c) 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.

Other information

- 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 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 is applicable:

\$P_SETFR	System frame for actual value setting, scratching
\$P_EXTFR	System frame for work offset external
\$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

Both the offset set via frames (coarse and fine) and the rotation and mirroring defined there are displayed. Display is according to the position in the frame chain.

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	
Change active work offsets and compensations				
		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.



Other Information

As of SW 6.2:

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

As of SW 6.3:

Workpiece reference	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 main screen



Teach In

In "MDI" (Manual Data Input) mode, you can write part programs block by block and execute them. You can transfer the required motions as single part 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.

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.

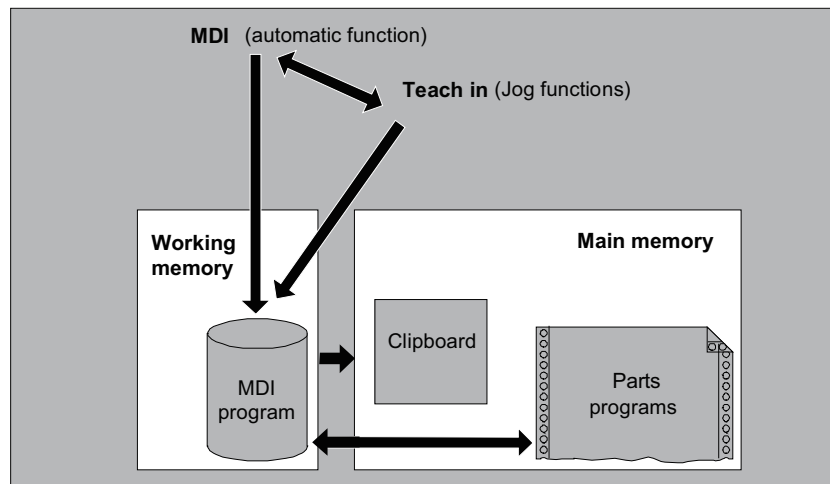
Blocks that have already been executed can only be edited in the Reset state.

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 part programs directory (MPF.DIR)



The "MDI" basic display contains values relating to position, feedrate, spindle and tool as well as the contents of the MDI clipboard.

Machine	CHAN1	MDI	VSF.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 [%]				
MDA- program			Feedrate	mm/min
N51 Y0 H55=99			Act.	7000.000 0.000 %
Y120 F7000			Set	7000.000
			Tool	
			preselected tool:	
			G0	G91
Program control		Handwheel	Editor function	MDI file fct.

Explanation of "MDI" basic display

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

Save MDI
prog.

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

Editor
functions

Press this softkey to access vertical softkeys providing editing functions:

Overwrite, Mark, Copy, Paste, Delete, Find, Find Next, Position.

MDI file fct.

You can save the MDI buffer contents or copy a part program to the MDI buffer for editing in MDI or Teach In.

Vertical softkeys

Like the Jog basic display, the MDI contains softkeys "G Fct.+Transf.", "Auxiliary functions" and "Spindle" (if spindle is configured).

Delete MDI
buffer

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 part program to the MDI buffer to modify it in MDI or Teach In mode.



Operating sequence

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

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

The new file is then displayed in the program overview.

MDI file
function

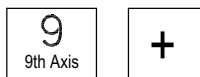
The program overview of part programs is displayed. From the list of programs select the one that you wish to copy to the MDI buffer with "Read in MDI". This can now be modified accordingly.

Read in
MDISave MDI
prog.

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 part program (MPF) under the name you have entered in the Part Programs directory.

4.5.3 Teach-in

**Function**

With the function "Teach In" part 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**Operating sequence**

"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 "Part 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

Operating sequence

"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.

Other information

- 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 part 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 main screen

You can execute part 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 part programs:

- You have already synchronized the control measuring system with the machine (i.e. "approached" reference points).
- You have already downloaded the associated part program to the control.
- You have checked or entered the necessary offset values, such as work offsets or tool offsets.
- The required 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	
<input checked="" type="checkbox"/> Channel reset	Program aborted		G fct. transf.
			Auxiliary func.
<input checked="" type="checkbox"/> MCS	Position	Dist.-to-go	Master spindle S1
-X	53.761 mm	-25.761	Act. + 0.000 rpm
+Y	30.000 mm	0.000	Set 0.000 rpm
+Z	112.704 mm	-112.704	Pos 0.000 deg.
+	0.000 mm	0.000	0.000 %
			Power [%]
Current block TEST.MPF			Feedrate mm/min
N51 Y0 H55=99			Act. 7000.000 0.000 %
Y120 F7000			Set 7000.000
			Tool
			preselected tool:
			G0 G91
Over-store	DRF offset	Program control	Block search
		Handwheel	Program overview

Explanation of basic display

Like the Jog display, the Automatic 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 part program.

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

Other information

The other softkeys are described in the sections below.

4.6.2 Program Overview



Program
overview

Selection

Alter
enable

Work-
pieces

Part
programs

Function

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

Operating sequence

"AUTO" mode 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

Display of all existing part 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	Display of all existing manufacturer cycles

4.6.3 Loading and unloading the workpiece/part program



Function

Workpieces and part 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.



Operating sequence

The "Machine" operating area is selected.

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

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

Program overview

Download
HD->NC

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

It is erased from the hard disk.


Enabling of

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

Upload
NC->HD

The highlighted workpiece/part 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.

Other information

Workpieces/part 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.

Operating sequence


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.



Program
overview



Log

4.6.5 Executing programs from the hard disk



Program
overview

Execute fr.
hard disk



External
programs



Function

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

Operating sequence

The "Machine" operating area is selected.

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).

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

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.

Other information

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 part program is selected in this channel.

From SW 6.2:

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 part 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.



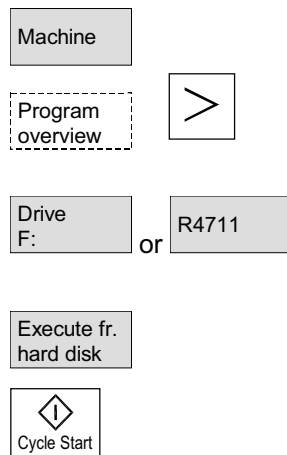
Operating sequence

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 external execution. 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 you have configured four softkeys, the fourth one is displayed in the More bar when you press the "ETC" key. 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 part program: EXTCALL

It is possible to access files on network drives from a part program using the command EXTCALL.
The programmer can set the source directory in the part program in setting data SD 42700: EXT_PROG_PATH, then specify the file name of the subroutine to be loaded in the EXTCALL command.



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

- With EXTCALL you can only call files with the MPF or SPF extension from a network drive.
- DOS naming conventions must be used for the files and paths: max. 25 characters for the name, 3 characters for the extension name.
- A program unit is found on a network drive with the EXTCALL command if
 - The search path in SD 42700: EXT_PROG_PATH points to the network drive or a directory contained on the network drive. The program must be stored directly on that level, no subdirectories are searched.
 - The correct location of the program is specified in the EXTCALL call itself by means of a fully qualified path that can also point to a subdirectory of the network drive.
 - The search path is not restricted by means of 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 part 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.



Operating sequence

"AUTO" mode 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

When you press this softkey, the correction editor is displayed, the softkey bars change and the softkey "Edit" is highlighted.

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 status:
Only program lines that have not yet been executed can be edited.
- Reset status:
All program lines can be edited.

Note:

You cannot use the "Correct program" function if you are running a program from the hard disk.

Other information

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 part 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 part 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 search with calculation, the same calculations are performed as in normal program operation. 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 made during 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 supported for parallel interaction with other functions. No axis movements are executed during the block search, however all auxiliary functions are output to the PLC. The NC will start the selected program in Program test mode automatically. 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
- by specifying a block number, a label, a program name or any character string.

References

For further information about block search, please refer to:

/FB/ K1, BAG, Channel, Program control





Block
search

Calculate
contour



Calculate
block end

Without
calculation



Reset

Operating sequence

"AUTO" mode 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.

Search
position

Program
level +

or

Program
level -

Search
pointer

0

5

...

A

Z

...

0

9

...

Interrup.
point

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 part 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 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.

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 Accelerated external block search



Function

The block search function allows you to run the part 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

- Directly positioning the cursor to the destination block, or
- Specifying a block number or a line number.

Block search sequence:

Only those programs and program parts which are needed in order to reach the specified search target and to continue program execution are transferred to the NC. This applies to the following:

- "Execution from external source" function
- Execution of EXTCALL instructions

Example:

The following search destinations are entered in the Block search menu:

Search target 1

Search target 2

Search target 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 target 1 8      EXTCALL "SUBPRG2"
9      X50
10     M30

SUBPRG2.SPF
1      X200 Y300
2      X400 Y500
3      EXTCALL "SUBPRG3"
.
.
Search target 2 4000 EXTCALL "SUBPRG3"
.
.
5000 M17

SUBPRG3.SPF
1      X20 Y50
2      .
.
Search target 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 target 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 target Line 8.
- Search target 2: The NC jumps to the search target at line 4000 of external subroutine SUBPRG2.SPF; the previous statements in the external program are skipped.
- Search target 3: The NC jumps to the search target at line 2300 of external subroutine 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 target 1: The NC jumps to line 8 of the search target.
- Search target 2: See above
- Search target 3: See above

On NC Start, the subroutine SUBPRG3.SPF is started and processed starting at the destination line.

Operating sequence

"AUTO" mode 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.

Switch to the "Search position" dialog.

Define search destination in program editor:

The current selected program level is displayed.

The block search via "Search position" is only possible for programs which have been loaded onto the NC and not for programs executed from the hard disk.

Position the cursor bar on a destination block of your choice in the part program.

If an interruption point is available on the NC, you can switch back and forth between the program levels.

Start search without calculation for external programs.

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 as search target types.

Start search without calculation for external programs.



Block
search

Search
position

Program
level +

or

Program
level -

Ext. without
calc.

Search
pointer

Ext. without
calc.

4.6.10 Block search in program testing mode, multi-channel



Function

The NC is operated in program test mode for this block search, so that interactions between channel and synchronous actions and between multiple channels are possible within an NCU.

The "Block search in Program test mode" enables a program advance up to the desired point of the part program with the following functions:

During "Block search in program testing mode", the NC outputs all auxiliary functions to the PLC and the part program commands for channel coordination (WAITE, WAITM, WAITMC, Axis replacement, Write variables, etc.) are carried out by the NC.

- This means that the PLC is updated during this block search and
- machining operations involving multiple 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 target by

- Directly positioning the cursor to the target block, or
- Using the input fields "Program name", "Search type" and "Search target".

Other information

The function "Block search in program test mode" requires NC SW 6 or later.

Operating sequence

Requirement:

- "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

Switch to the "Search position" dialog.

Search
position

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 part program.

Program
level +

or

Program
level -

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 part 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 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 above). "Search target in the program editor".

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



Overstore



Overstore

Function

In "AUTO" mode you can overwrite technological parameters (auxiliary functions, all programmable instructions ...) in the working memory of the NC. You can also enter and execute any NC block.

Operating sequence

"AUTO" mode 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.

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.

Other information

- 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 part 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	
MFct*	Programmed stop 2	from HMI SW 6.3 and NCK SW 6.4

DRF	DRF selection
PRT	Program test

SBL1	Single block with STOP after each machine function
SBL2	Single block with STOP after each block
SBL3	Stop in cycle

MFct* The M function displayed in the machine data is displayed.

With horizontal softkeys:

"Skip blocks": Skip block; with software Version 6.4 and higher, you can activate up to 10 skip levels. See machine manufacturer's specifications.

"Block display": Block display modes with 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 or /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 (10 skip levels with SW 6.4 and higher).

Blocks to be skipped are indicated in the part program by an oblique "/" before the block number. The skip levels in the part program are specified by "/0" or by "/" to "/9". You can activate/deactivate skip levels in the following window:

Program control	
SKP: Skip blocks	active
<input checked="" type="checkbox"/> Skip /	<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"/>
<input checked="" type="checkbox"/> Skip /8	<input type="checkbox"/>
<input checked="" type="checkbox"/> Skip /9	<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.



Program
control



Skip blocks, SW 6.3 and higher

Skip
blocks



Operating sequence

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

Press the horizontal softkey "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.

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".

Press the vertical softkey "Skip blocks"

The vertical softkey "Skip blocks" is only displayed 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.

Other information

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

Block display

Program execution modes

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".

Press the vertical softkey "Block display"

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 traversing 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* (from SW 6.3)

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 Startup operating area under Machine Data/Channel MD by selecting the machine data corresponding to the M function. The number stored in this machine data is then displayed. The value of this miscellaneous M function number must match the programmed stop 2.

Other information

Blocks which are too long for the display window (approx. 65 characters) are truncated and the truncated code is replaced by "...".

4.6.13 DRF offset

**Activation/deactivation****Function**

The DRF (Differential Resolver Function) offset allows an additional incremental zero offset to be set using the handwheel. "DRF" is an axial procedure which acts on the basic coordinate system. Handwheel selection is permissible via the axial interface only; i.e. you can only use machine axes.

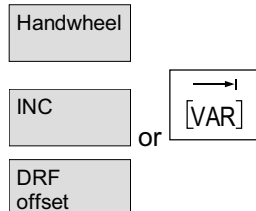
Alter DRF

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 is performed for all axes
- DRFOF (selection of DRF via part program)
- PRESETON (modification of the actual value through Preset)

You can alter the DRF offset by traversing the appropriate machine axis using the handwheel (the actual value display does not change).

**Operating sequence**

"AUTO" mode 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".



"Parameters" Operating Area

5.1	Tool data	5-167
5.1.1	Tool offset structure	5-167
5.1.2	Tool types and tool parameters	5-167
5.2	Tool offsets	5-182
5.2.1	Function and main screen for tool offsets	5-182
5.2.2	New tool	5-184
5.2.3	Display tool	5-185
5.2.4	Delete tool	5-186
5.2.5	New cutting edge	5-187
5.2.6	Delete cutting edge	5-188
5.2.7	Determine tool offsets	5-188
5.2.8	Activate tool offset immediately	5-190
5.3	Tool management	5-191
5.3.1	Basic tool management functions	5-193
5.3.2	Modifying/changing tool data	5-201
5.3.3	Change of meaning/representation of tool wear data	5-205
5.3.4	Grinding parameters extension in SW 6.2 or later	5-207
5.3.5	Loading	5-210
5.3.6	Unload	5-214
5.3.7	Relocation	5-216
5.3.8	Tool master data in the tool catalog	5-217
5.3.9	Tool offset data in the tool cabinet	5-220
5.3.10	Batch processing of tools	5-224
5.4	R parameters (arithmetic parameters)	5-232
5.4.1	Function	5-232
5.4.2	Editing/deleting/finding R parameters	5-232
5.5	Setting data	5-234
5.5.1	Working area limitation	5-234
5.5.2	Jog data	5-235
5.5.3	Spindle data	5-236
5.5.4	Dry run feedrate for dry run DRY	5-237
5.5.5	Starting angle for thread cutting	5-238
5.5.6	Other types of setting data	5-239
5.5.7	Protection zones	5-240
5.5.8	Electronic gears (SW 6.3 and higher)	5-241
5.6	Work offset	5-242
5.6.1	Function	5-242
5.6.2	Changing the settable work offset (G54 ...)	5-244
5.6.3	Global work offset/frame	5-244
5.6.4	Display active settable work offset	5-247
5.6.5	Display active programmable work offset	5-248
5.6.6	Display active work offset external	5-249
5.6.7	Display sum of the active work offsets	5-249
5.6.8	Immediately activate work offset and basic frame	5-250

5.6.9	Actual-value display: settable zero system, ENS.....	5-250
5.7	User data/user variables (GUD, PUD, LUD)	5-251
5.7.1	General.....	5-251
5.7.2	Changing/finding user data/user variables.....	5-252
5.8	Display system variables.....	5-254
5.8.1	Creating/editing variable views	5-255
5.8.2	Manage variable views.....	5-257
5.8.3	Log system variables.....	5-257

5.1 Tool data

5.1.1 Tool offset structure

	<p>A tool is selected in the program with 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. Tool management is therefore no longer 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. Tool number</p> <p>D No. Cutting edge number</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>
----------------	--

5.1 Tool data

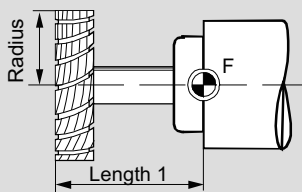
Tool types

Tool type classification:

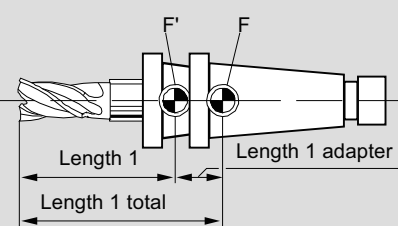
- **Group type 1xy (milling tools):**

- 100 Cutter to CLDATA
- 110 Ball-end cylindrical die-sinking cutter
- 111 Ball-end tapered die-sinking 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
- 156 Bevel cutter with rounding
- 157 Tapered die-sinking 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 Other values must be set to zero		Effect	
		G17:	Length 1 in Z Radius in X/Y
		G18:	Length 1 in Y Radius in Z/X
		G19:	Length1 in X Radius in Y/Z
		F - Toolholder reference point	

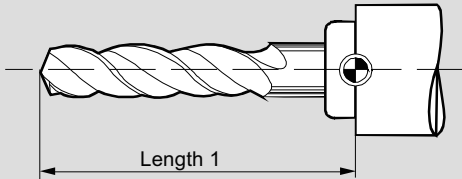
Milling tool with adapter

Entries in tool parameters			
DP1	1xy		
DP3	Length 1 -Geometry		
DP6	Radius -Geometry		
DP21	Length -Adapter		
Wear values according to requirement		F-Adapter reference point (when tool is inserted = toolholder reference point)	
Other values must be set to zero		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
		F' - Toolholder reference point	

Group type 2xy (drills):

- 200 Twist drill
- 205 Drill
- 210 Boring bar
- 220 Center drill
- 230 Countersink
- 231 Counterbore
- 240 Tap regular thread
- 241 Tap fine thread
- 242 Tap Whitworth thread
- 250 Reamer

Required offset values for a drill

Entries in tool parameters		
DP1	2xy	
DP3	Length 1	
<p>Wear values according to requirement</p> <p>Other values must be set to zero</p>		

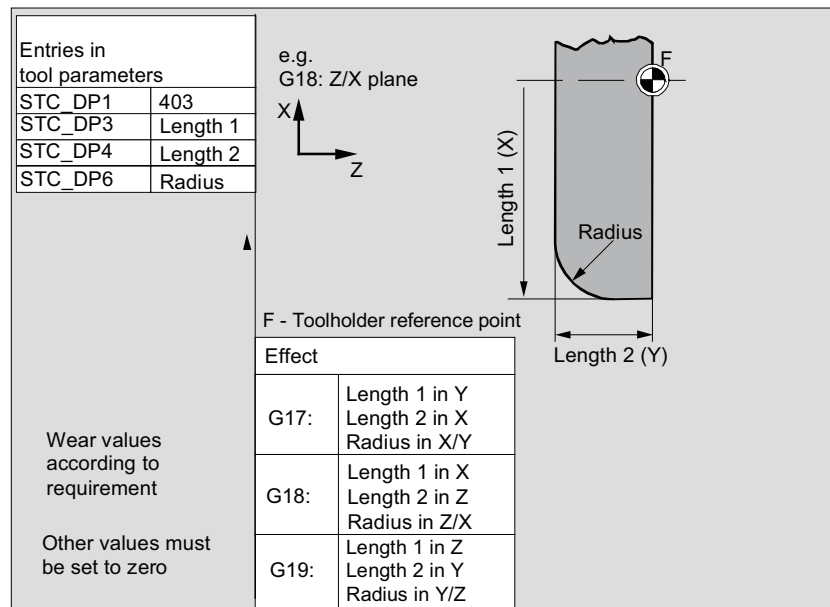
Effect	
G17:	Length 1 in Z
G18:	Length 1 in Y
G19:	Length 1 in X

F - Toolholder reference point

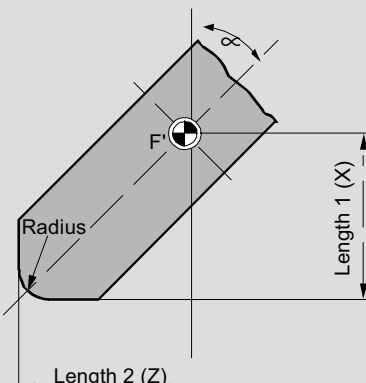
- **Group type 4xy (grinding tools):**

- 400 Surface grinding wheel
- 401 Surface grinding wheel with monitoring
- 402 Surface grinding wheel without monitoring
- 403 Surface grinding wheel with monitoring without tool base dimension for grinding wheel peripheral speed GWPS
- 410 Facing wheel
- 411 Facing wheel with monitoring
- 412 Facing wheel without monitoring
- 413 Facing wheel with monitoring without tool base dimension for grinding wheel peripheral speed GWPS
- 490 Dresser

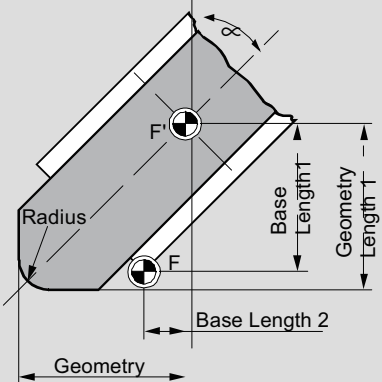
Offset values required by a surface grinding wheel



Required offset values for inclined grinding wheel with implicit monitoring selection

Entries in tool parameters		STC_TPG1	Spindle number
		STC_TPG2	Chaining rule
STC_DP1	403	STC_TPG3	Minimum wheel radius
STC_DP3	Length 1	STC_TPG4	Minimum wheel width
STC_DP4	Length 2	STC_TPG5	Current wheel width
STC_DP6	Radius	STC_TPG6	Maximum speed
Wear values according to requirement		STC_TPG7	Max. surface speed
		STC_TPG8	Angle of the inclined wheel
		STC_TPG9	Parameter no. for radius calculation
Other values must be set to zero		F: Toolholder reference point	
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		

Required offset values using the example of inclined grinding wheel with implicit monitoring selection

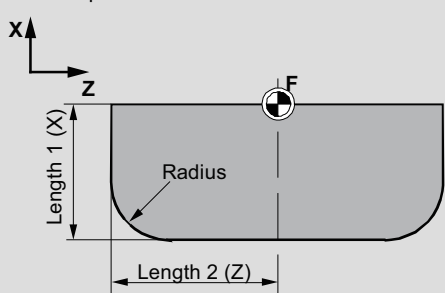
Entries in tool parameters		STC TPG1	Spindle number
		STC TPG2	Chaining rule
STC_DP1	403	STC TPG3	Minimum wheel radius
STC_DP3	Length 1	STC TPG4	Minimum wheel width
STC_DP4	Length 2	STC TPG5	Current wheel width
STC_DP6	Radius	STC TPG6	Maximum speed
		STC TPG7	Max. surface speed
		STC TPG8	Angle of the inclined wheel
		STC TPG9	Parameter no. for radius calculation
Wear values according to requirement		F: Toolholder reference point 	
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		

5.1 Tool data

Required offset values of a surface grinding wheel without base dimension for GWPS

Entries in tool parameters		STC_TPG1	Spindle number
		STC_TPG2	Chaining rule
STC_DP1	403	STC_TPG3	Minimum wheel radius
STC_DP3	Length 1	STC_TPG4	Minimum wheel width
STC_DP4	Length 2	STC_TPG5	Current wheel width
STC_DP6	Radius	STC_TPG6	Maximum speed
STC_DP21	L1 base	STC_TPG7	Max. surface speed
STC_DP22	L2 base	STC_TPG8	Angle of the inclined wheel
Wear values according to requirement		STC_TPG9	Parameter no. for radius calculation
		F: Toolholder reference point	
Other values must be set to zero			

Required offset values of a facing wheel with monitoring parameters

Entries in tool parameters		STC_TPG1	Spindle number
		STC_TPG2	Chaining rule
STC_DP1	403	STC_TPG3	Minimum wheel radius
STC_DP3	Length 1	STC_TPG4	Minimum wheel width
STC_DP4	Length 2	STC_TPG5	Current wheel width
STC_DP6	Radius	STC_TPG6	Maximum speed
Wear values according to requirement		STC_TPG7	Max. surface speed
Other values must be set to zero		STC_TPG8	Angle of the inclined wheel
		STC_TPG9	Parameter no. for radius calculation
Effect		<p>F: Toolholder reference point</p> <p>e.g. G18: Z/X plane</p> 	
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		

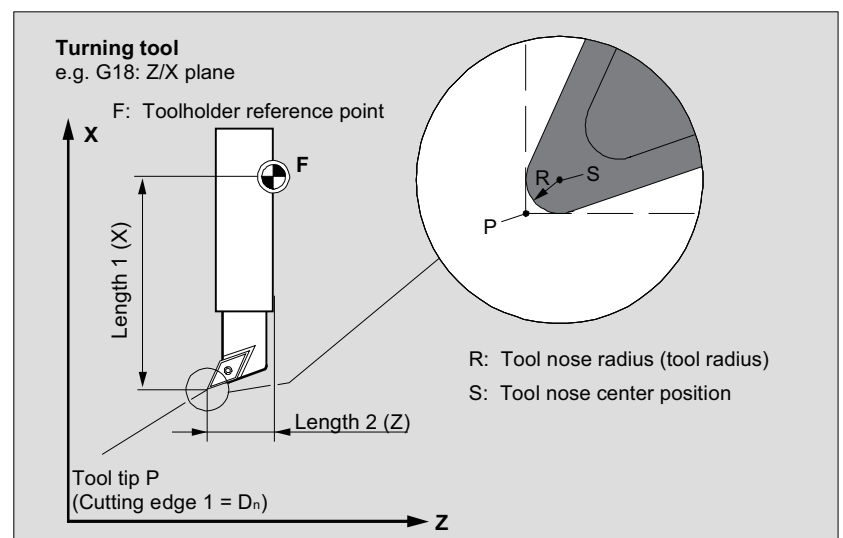
Assignment of tool-specific parameters

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 peripheral speed	Real
\$TC_TPG8	Angle of inclined wheel	Real
\$TC_TPG9	Parameter number for radius calculation	Integer
Additional parameters		
\$TC_TPC1	Angle of inclined wheel	Real
up to		
\$TC_TPC10		Real

• Group type 5xy (turning tools):

- 500 Roughing tool
- 510 Finishing tool
- 520 Plunge cutter
- 530 Parting tool
- 540 Threading tool
- 550 Steel profile
- 560 Rotary drill (ECOCUT)
- 580 Oriented probe

Required offset values for a turning tool with tool radius compensation



5.1 Tool data

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	
DP1	5xy
DP2	1...9
DP3	Length 1
DP4	Length 2
DP6	Radius

Wear values
according to
requirement

Other values must
be set to zero

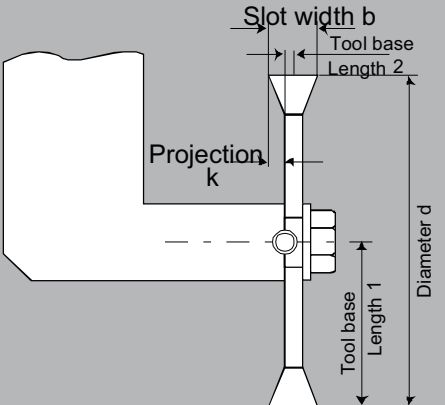
Effect	
G17:	Length 1 in Y Length 2 in X
G18:	Length 1 in X Length 2 in Z
G19:	Length 1 in Z Length 2 in Y

- Group type 7xy (special tools)**

- 700 Slotting saw
- 710 3D probe
- 711 Edge probe
- 730 Fixed 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		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)
Other values must be set to zero			
$L1 = DP3 + DP6/2$ $L2 = DP4 + DP7/2 - DP8$ $R = DP7/2$			

The offset data (TOA data) you can enter for tool type 700 "slotting saw" are as follows:

	Geometry	Wear	Base	
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 compensation				
Diameter	\$TC_DP6	\$TC_DP15		mm
Slot width b	\$TC_DP7	\$TC_DP16		mm
Projection k	\$TC_DP8	\$TC_DP17		mm

3D probe

Dead stop

Type 710

See /BNM/, User's Guide Measuring Cycles

Type 730

The stop is used to position the material in turning machines with bar feed. Only tool length compensation is important.

5.1 Tool data

Calculation of tool parameters

Types 1xy (milling cutters), 2xy (drills) and 5xy (turning tools) are calculated according to the same scheme.

Several entries exist for the geometric variables (e.g. length 1 or radius). These are added together to produce a value (e.g. total length 1, total radius) which is then used for the calculations.

Tool parameter number (P)	Meaning	Remarks
1	Tool type	For overview see list
2	Tool point direction	Only for turning tools
Geometry Length compensation		
3	Length 1	Calculation according 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		
Radius compensation		
12	Length 1	
13	Length 2	
14	Length 3	
15	Radius	
16	Reserved	
17	Reserved	
18	Reserved	
19	Reserved	
20	Reserved	

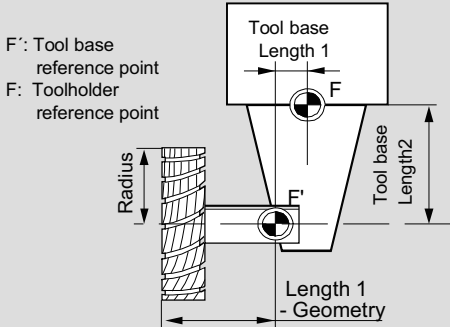
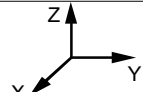
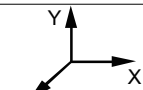
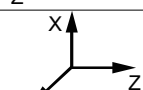
Base dimensions/ adapter		Length offsets
21		Length 1
22		Length 2
23		Length 3
Technology		
24	Clearance angle	For turning tools
25	Clearance angle	

Offsets that are not required must be assigned the value 0
(= 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.

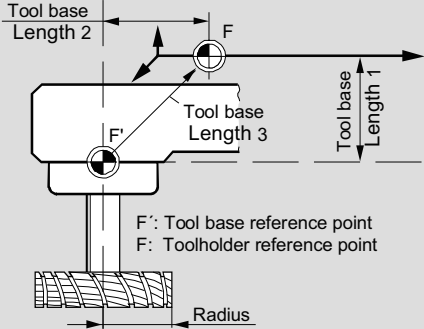
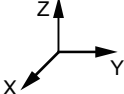
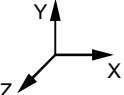
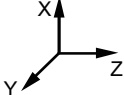
The tool offsets can be entered not only via the operator panel front
but also via the data input interface.

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			
Wear values according to requirement		Effect		
		G17:	Length 1 in Z Length 2 in Y Length 3 in X Radius in Y/Z	
		G18:	Length1 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	
Other values must be set to zero				

5.1 Tool data

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			
DP23	Length 3 - base			
Wear values according to requirement		Effect		
		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	
Other values must be set to zero				

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 Other values must be set to zero		Effect	
		G17:	Length 1 in Y Length 2 in X
		G18:	Length 1 in X Length 2 in Z
		G19:	Length 1 in Z Length 2 in Y

Turning tool with several cutting edges - length compensation:

Entries in tool parameters	
DP1	5xy
DP3	Length 1
DP4	Length 2
DP1	5xy
DP3	Length 1
DP4	Length 2

Effect	
G17:	Length 1 in Y Length 2 in X
G18:	Length 1 in X Length 2 in Z
G19:	Length 1 in Z Length 2 in Y

Wear values according to requirement

Other values must be set to zero

Grooving tool
e.g. G18: Z/X plane

Tool tip P
(Cutting edge 1 = D n)

(Cutting edge 2 = D n)

Dn: Length 1 (X)

Dn: Length 2 (Z)

F: Toolholder reference point

Calculation of tool base dimensions for turning machine:

Entries in tool parameters	
DP1	5xy
DP3	Length 1 - Geometry
DP4	Length 2 - Geometry
DP21	Length 1 - Base
DP22	Length 2 - Base

Effect	
G17:	Length 1 in Y Length 2 in X
G18:	Length 1 in X Length 2 in Z
G19:	Length 1 in Z Length 2 in Y

Wear values according to requirement

Other values must be set to zero

Turning tool
e.g. G18 Z/X plane

Tool base Length 2

Tool base Length 1

Length 1 - Geometry

Length 2 - Geometry

F': Tool base reference point

F: Toolholder reference point

5.1 Tool data

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	$\ast(2^0=1)$	Tool type	Tool type
\$TC_DP2	Tool point direction	Tool point direction	Tool point direction	Tool point direction
Geometry tool length compensation				
\$TC_DP3	Length 1	$\ast(2^2=4)$	Length 1	Length 1
\$TC_DP4	Length 2	$\ast(2^3=8)$	Length 2	Length 2
\$TC_DP5	Length 3	$\ast(2^4=16)$	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	$\ast(2^{11}=2048)$	Length 1	Length 1
\$TC_DP13	Length 2	$\ast(2^{12}=4096)$	Length 2	Length 2
\$TC_DP14	Length 3	$\ast(2^{13}=8192)$	Length 3	Length 3
\$TC_DP15	Radius	Radius	Radius	Radius
\$TC_DP16 to \$TC_DP20	Reserved	Reserved	Reserved	Reserved
Base dimension/adaptor dimension tool length compensation				
\$TC_DP21	Basic length 1	$\ast(2^{20}=1048576)$	Basic length 1	Basic length 1
\$TC_DP22	Basic length 2	$\ast(2^{21}=2097152)$	Basic length 2	Basic length 2
\$TC_DP23	Basic length 3	$\ast(2^{22}=4194304)$	Basic length 3	Basic length 3

Technology				
\$TC_DP24	Reserved	Reserved	Reserved	Reserved
\$TC_DP25	Reserved	Reserved	Reserved	Reserved
Additional parameters				
\$TC_DPC1				
up to				
\$TC_DPC10				

* Value of the chaining parameter if the compensation parameters 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 \$TC_TPG1

This parameter contains the number of the spindle to which the monitoring data and GWPS refer.

Chain rule \$TC_TPG2 \$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.

It must be noted that the minimum grinding wheel radius must be specified in the cartesian coordinate system for an inclined grinding wheel. The length compensations always specify the distances between the tool holder reference point and the tool tip in Cartesian coordinates.

The monitoring data apply to both the left-hand and the right-hand cutting edge of the grinding wheel.

The tool lengths are not automatically compensated when the angle is altered.

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 offsets

5.2.1 Function and main screen for tool offsets

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 are displayed. The input field is highlighted.

Parameter	CHAN1	AUTO	MPF.DIR TEST.MPF																																																																											
Channel active		Program running																																																																												
<div> <div> <div></div> <div>T No. +</div> </div> <div> <div></div> <div>T No. -</div> </div> <div> <div></div> <div>D No. +</div> </div> <div> <div></div> <div>D No. -</div> </div> <div> <div></div> <div>Delete...</div> </div> <div> <div></div> <div>Go to...</div> </div> <div> <div></div> <div>Overview...</div> </div> <div> <div></div> <div>New...</div> </div> <div> <div></div> <div>Determine compens...</div> </div> </div>																																																																														
Tool offset data																																																																														
T number	1	D number	1																																																																											
Tool type	100	No. of c. edges	1																																																																											
C. edge pos.	1																																																																													
<table border="1"> <thead> <tr> <th></th> <th>Geometry</th> <th>Wear</th> <th>Base</th> <th></th> </tr> </thead> <tbody> <tr> <td>Tool length comp.</td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>Length 1</td> <td>:</td> <td>0.000</td> <td>0.000</td> <td>mm</td> </tr> <tr> <td>Length 2</td> <td>:</td> <td>0.000</td> <td>0.000</td> <td>mm</td> </tr> <tr> <td>Length 3</td> <td>:</td> <td>0.000</td> <td>0.000</td> <td>mm</td> </tr> <tr> <td>Radius comp.</td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>Radius</td> <td>:</td> <td>0.000</td> <td>0.000</td> <td>mm</td> </tr> <tr> <td>DP7.18.res</td> <td>:</td> <td>0.000</td> <td>0.000</td> <td></td> </tr> <tr> <td>DPR.17.res</td> <td>:</td> <td>0.000</td> <td>0.000</td> <td></td> </tr> <tr> <td>DP9.18.res</td> <td>:</td> <td>0.000</td> <td>0.000</td> <td></td> </tr> <tr> <td>DP10.19.res</td> <td>:</td> <td>0.000</td> <td>0.000</td> <td></td> </tr> <tr> <td>DP11.20.res</td> <td>:</td> <td>0.000</td> <td>0.000</td> <td></td> </tr> <tr> <td>Length compensation</td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>Clear angle</td> <td>:</td> <td>0.000</td> <td>Deg.</td> <td></td> </tr> <tr> <td>DP25.res</td> <td>:</td> <td>0.000</td> <td></td> <td></td> </tr> </tbody> </table>					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		DPR.17.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		
	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																																																																											
DPR.17.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																																																																												
Tool offset	R variables	Setting data	Zero offset																																																																											
		User data																																																																												

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.

The maximum number of offset parameters (T and D numbers) can be set by means of machine data.



Tool wear: fine compensation

If you have at least the authorization which is entered in MD 9202: USER_CLASS_TOA_WEAR, you can change the values of the tool fine offset incrementally. The difference between the old and new values must not exceed the limit stored in MD 9450: WRITE_TOA_FINE_LIMIT.

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

OK + new
cutting edge

Function

If you create a new tool, the associated tool types are automatically displayed to assist you when you select the tool group.

Operating sequence

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 Plunge cutter
- 530 Parting tool
- 540 Threading tool
- 550 Steel profile
- 560 Rotary drill (ECOCUT)
- 580 Oriented probe

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.

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.

Operating sequence

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.

Other information

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

Delete
tool

Function

The tool is deleted together with all its cutting edges and the tool list updated accordingly.

Operating sequence

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 cutting edge



Tool
offset

New ...

New
edge



Abort

OK

OK + new
cutting edge

OK + new
tool

Function

To help you to select a new tool edge, the associated tool types are displayed automatically when you select a tool group.

Operating sequence

The "Tool offset" window is displayed automatically.

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 Plunge cutter
- 530 Parting tool
- 540 Threading tool
- 550 Steel profile
- 560 Rotary drill (ECOCUT)
- 580 Oriented probe

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 cutting edge



Delete

Delete
edge**Function**

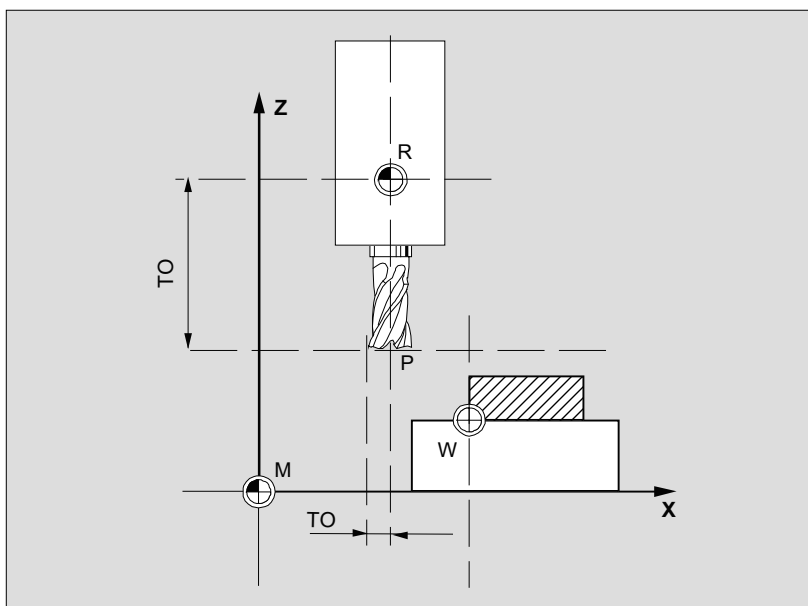
You can delete one or several edges of a tool. The tool list is updated automatically.

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 reference point
- W** Workpiece zero



Tool
offset

Determine
compens...



OK

Calculate



Operating sequence

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.

When you press softkey "OK", the current position and corresponding reference value for the selected tool parameter are calculated.

The following equation applies: $\text{Position} - \text{reference value} = \text{input value}$

The window is closed.

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 Activate tool offset immediately**Function**

Machine data can be set to ensure that the active tool offset is activated as soon as the part program is in the "Reset" or "Stop" state.

Other information

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 movement in the part program.

5.3 Tool management

Magazine list

The tool management system is organized by means of various configurable lists which show different views of the tools used.

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.



Important

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.

The TOA is assigned to multiple channels and is effective in the channel in which the tool is active. If you want to change the wear in the list of working offsets without having to call the tool again at the next NC Start, ensure that the correct channel is selected for this tool.

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
 /FBS/ Description of Functions ShopMill



5.3.1 Basic tool management functions

Tool ma-
nagement



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:

Parameter	CHAN1	AUTO	MPF.DIR LEER.MPF	
<input checked="" type="checkbox"/> Channel RESET			Program aborted	
			ROV	FST
Magazine list 1				
Magazin: 2 - Kette20 Plätze: 20 AVB: 0				
Pl	PP	PTP	WerkzeugID	Dupl. TNr. PTT W W WWW W PV WTyp xGeo-L1 xGeo-L2
1	-	1	newRack860	1 1 1 - F G M V - 0 900 11.0000 11.0000
2	-	F 1	Wzg1	1 76 1 - F - - - - 0 900 0.0000 0.0000
3	-	F 1		
4	-	1		
5	-	1	Wzg2	1 85 1 - F - - - E 0 900 0.0000 0.0000
6	-	F 1	Wzg3	1 7 1 - F - - - - 0 900 0.0000 0.0000
7	-	F 1		
8	-	F 1		
9	-	F 1		
10	-	F 1		
11	-	F 1		
12	-	F 1		
13	-	F 1		
14	-	F 1		
<div> <div>Magazine list</div> <div>Tool list</div> <div>Load</div> <div>Unload</div> <div>Relocate</div> <div>Working offset list</div> </div>				

Location (PI)	Location number
Location state (P)	Location state (one column is provided for each state) e.g. F = Location enabled G = Location disabled Z = Reserved for tool in intermediate memory B = Reserved for tool to be loaded L = Left half location occupied R = Right half location occupied O = Upper half location occupied U = Lower half location occupied I = Left half location reserved (from SW 6.4) r = Right half location reserved (from SW 6.4) o = Upper half location reserved (from SW 6.4) u = Lower half location reserved (from SW 6.4)
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 = Tool is measured V = Prewarning limit is reached W = Tool is being changed P = Tool is fixed-location-coded E = Tool has been used R = Unloading marking ("radius") B = Loading marking S = Master tool
PV	Wear group assigned to the tool.

WTy

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 – Len 1
Radius ...**

Tool offsets such as length, radius, wear, monitoring data, etc.

Horizontal softkeys

Magazine
list

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.

Tool
list

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).

Load

A magazine location is assigned to the tool.

Unload

The tool is deleted from the current magazine location.

Relocate

The tool is moved from the current magazine location to another location.

Working
offset list

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:

Tool
catalog

You can create new tool master data ("ideal" tools) and modify the existing data.

Tool
cabinet

You can create new tool offset data and tool operating data ("real" tools) and modify the existing data.

Trans-
formation

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.

Special feature:

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 tool 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" (see "Displaying/modifying tool data" 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 intermediate memory is 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.
- Positioning

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.



Vertical softkeys (tool list)

(Names assigned by
user)

Selection of a table structure configured by the user
(if configured), e.g.:

Tool
list 1

- General data

Tool
list 2

- Geometry tool data

Tool
list 3

- Wear data

Tool
details

You can display and edit the complete data of a tool (as in the magazine list).

Tool from
CC

The tool data are read from a code carrier and entered in the tool list (you can then edit the data).

Tool in cabinet

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.

Delete tool

The selected tool is removed from the list. Use the vertical softkeys to determine whether to save the tool data.

Tool in cabinet

The data are copied into the tool cabinet. The tool can be loaded later with the same data.

Data on CC

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.

Abort

The operation is canceled. The tool is not removed from the list.

OK

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.

Vertical softkeys (working offset list)

(Names assigned by user)

W. offset
list 1

W. offset
list 2

W. offset
list 3

Tool
data

Find
D number

Current
D number

Selection of a table structure configured by the user (if configured), e.g.:

- General data
- Geometry tool 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 Modifying/changing tool data



Tool management

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

Operating sequence

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.

Notice:

Please ensure that the channel in which the corrections are to become effective is displayed.

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:

- Location type
- Location coding
- Monitoring method
- 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

Other information

If twelve 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. The machine manufacturer can limit the maximum permissible number of cutting edges to a smaller number. The default setting is nine cutting edges.

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.

Press softkeys "Cutting edge data", "Mon. data" and "Local offsets" to display tables for editing the respective data. You can move freely between the individual tables. The name, Duplo number and type of tool and all defined cutting edges (#1...#12) are displayed.

Use the softkeys "Cutting edge +" and "Cutting edge -" to change between the cutting edges. Press "<<" to return to the "Tool details" input screen.

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:

- Number of workpieces
- Tool 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:

- Values can be defined for Length1, Length2, Length3 and Radius1 for each 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.

Attention! 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 Change of meaning/representation of tool wear data

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	4
2		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	 HAND	REI3	1	30	5
10		REI3	1	31	5
11		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


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" deviates from the G56 reset value ().


Symbols  

Row 9 displays the current tool for channel "HAND". There the current value of G56 is equal to the reset value ().



TRANSFORMIERT: Arbeitskorrekturen Liste 1					
Zeil	Aktuell	Werkzeug-ll	Dupl	DNo	Mag
1		REI1	1	10	4
2		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		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


The G56 reset value is TOWMCS  and
 \$SC_WEAR_TRANSFORM is set.



The current tool for channel "REV2" is indicated in row 2. The current value of G56 on channel "REV2" deviates from the G56 reset value ().

Row 11 displays the current tool for channel "HAND". There the current value of G56 is equal to the reset value ().

Symbols  

TRANSFORMIERT: Arbeitskorrekturen Liste 1					
Zeil	Aktuell	Werkzeug-Id	Dupl	DNo	Mag
1		REI1	1	10	4
2		REI1	1	11	4
3		REI1	1	12	4
4		REI1	1	13	4
5		REI2	1	20	4
6	 REV2	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		REI3	1	32	5
12		REI3	1	33	5
13		REI4	1	40	5
14		REI4	1	41	5
15	 HAND	REI4	1	42	5
16		REI4	1	43	5

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 deviates from the G56 reset value (.

Other information

Additional information can be found in
/FBW/, Description of Functions Tool Management

5.3.4 Grinding parameters 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 screen
- Tool details cutting edge data (secondary) screen
- Tool details monitoring data (secondary) screen

When this softkey is pressed, a display appears where grinding data can be:

- Display
- Modified

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	Names
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 inclined wheel	[degrees]	\$TC_TPG8
Parameter number for radius calculation/compensation parameter for GWPS	-	\$TC_TPG9



Scrollbar

Other information

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)

Boundary 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 data input (with search for empty location)



Other information

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.

Press 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

Other information

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.

Search via user-defined location (example)

The assignment of the softkeys is configured by the machine manufacturer.

normal

large

oversize

normal and
heavy

- "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

Find empty
location

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 the corresponding empty location.

The cursor bar is automatically positioned on the magazine location found in the "Magazine list".

Search via the "To loading point" softkey

To loading
point

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.

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.



Importing tool data

Data from
CC

Data from
host

Tool from
cabinet

Tool
details

Abort

Start



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:

1. Read the data in from a code carrier (if one is installed)
2. Read the data in from a host computer (if one is installed)
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.

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.

Load/input mode is canceled.

A tool created via "Tool details" or "Start" is deleted. You can search for another empty location.

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.



Tool
list

Load

Find empty
location

Abort

Start

Sequence of operations (loading from the "Tool list")

The "Tool list" is selected.
The appropriate tool is selected.

Press the "Load" softkey.
The vertical softkey bar changes.

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.

The loading operation is not initiated.
The basic display appears.

The loading operation is initiated.

5.3.6 Unload



Function

This function allows you to unload a selected tool and to save its data.



Tool ma-
nagement

Operating sequence

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.

Magazine
list

Tool
list

Unload

Tool in
cabinet

Data on
CC

From
magazine



Delete
tool



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.

The vertical softkey bar changes.

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 exit the screen **without** unloading the tool, select another display in the vertical softkey bar before activating "Start".

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



Tool ma-
nagement

Magazine
list

Tool list

Relocate

Function

This function allows you to move a selected tool from one location to another.

Operating sequence

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.

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).

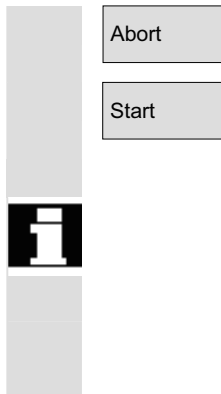
When you select the "Relocate" key, the "Relocate tool" window opens.

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 the 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".



Tool ma-
nagement



Tool
catalog

Create tool data

New

Abort

OK

Display/edit tool data

Operating sequence

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.

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.

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.



Offsets

Edge +

Edge –

New

Delete

Abort

OK

Cut. edge
user data

Tool user
data

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 "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 "Tool user data".
Up to 10 user-specific tool data are displayed here.
Enter the required settings in the table.

Additional functions

Copy

Delete

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.

Other information

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.



Tool ma-
nagement



Tool
cabinet

Create tool offset data

Abort

OK



Operating sequence

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.

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.

Please 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.

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.

5.3 Tool management

Display/Edit 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 entries 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 entries are displayed here. Enter the required settings in the table.

Abort

- Select "Abort" to discard the settings.
The data retain their old values.

OK

- Press "OK" to accept the changes.
The data are updated.

Additional functionsDelete

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).

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).

Other information

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 Batch processing of tools



Area of application

Function

The function "Batch processing of tools" allows the operator to:

- Load and unload, delete and store tools in the cabinet in one common job for several tools.
- 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. **Select a filter**
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 [BatchTools] section.

Country-specific sections are parameterized in the "language\patm_xx.ini" file in the [BatchTools] section; "xx" stands for the 2 letters user for the country code.



Precondition

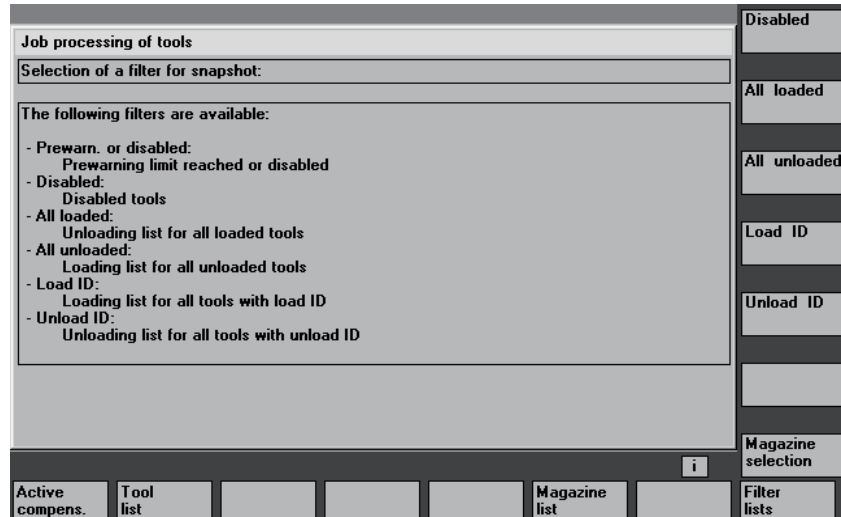
I.

Select a filter

Operating sequence

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. This data is **not** updated later.

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".

Filter 1-6

Magazine
selection

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 displayed is a **snapshot** which was created at the time of filtering; it is **not subsequently updated** when the data changes in the NC.

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 default setting, a tool selected for batch processing is displayed with a checked checkbox symbol ( and ). The color for "Cursor" and for "Selected for job processing" is identical and corresponds to the general selection display.

When selection of the tools is completed, press the softkey to start the **batch function**.

Begin job function

"Load"

No.	Sel.	Tool id.	Duplo	Mag	Loc	APD	MW	UOI
31	<input type="checkbox"/>	Test31	1	0	0			
32	<input checked="" 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			

Select all

Cancel
selection

Loading

Reactivate

Vertical softkeys

All tools in the hit list are selected for job processing.

The selection for job processing is canceled for all tools in the hit list.

"Loading" is initiated for the selected tools. You are prompted to enter the destination magazine and the load point in a dialog box.

"Reactivate" is initiated for the selected tools. When reactivating 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 to be 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"

Auftragsbearbeitung von Werkzeugen											
Entladeliste fuer alle Werkzeuge auf 1 Magazin begrenzbare											
Werkzeuge: 24, ausgewählt: 4.											
Nr.	Sel.	WZ-Bez.	Duplo	Mag	Pla	AF	GMV	EOL			
11	<input type="checkbox"/>	Test2	1	2	1						
12	<input type="checkbox"/>	Test6	1	2	2						
13	<input type="checkbox"/>	state_test1	1	2	3	F	GMV				
14	<input type="checkbox"/>	Test18	1	2	4						
15	<input type="checkbox"/>	Test22	1	2	5						
16	<input checked="" type="checkbox"/>	Test34	1	2	7						
17	<input type="checkbox"/>	Test42	1	2	9						
18	<input type="checkbox"/>	Test46	1	2	10						
19	<input checked="" type="checkbox"/>	Test48	1	3	10						
20	<input checked="" type="checkbox"/>	Test39	1	3	9						
21	<input checked="" type="checkbox"/>	Test9	1	3	2						
22	<input type="checkbox"/>	Test12	1	3	3						
23	<input checked="" type="checkbox"/>	Test27	1	3	6						
24	<input type="checkbox"/>	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.

Relocation

"Relocation" is initiated for the selected tools. A dialog box prompts you to enter the destination magazine where the tools are to be relocated. This magazine is the destination for all tools in this batch processing operation.

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 "^"





The current number of hits is discarded and the 1st screen "Filter selection" is displayed.

If batch processing has been started and the necessary entries have been made, the display switches to the 3rd screen "Batch 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": gray field, 
- "current tool in job processing": yellow/blue arrow, 
- "completed without errors": green field with tick, 
- "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	O
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 ...

Stop

Abort

OK

Active
compens.

Tool
list

Magazine
list

Filter
lists

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.

Any job processing that has been halted is continued.

This softkey can only be used while batch processing is halted.

Stop

Continue



Abort



OK

Any job processing that is halted is canceled. Non-completed jobs are discarded and display returns to the "Filter selection" screen.
This softkey can only be used while batch processing is halted.

All information relating to detail jobs is discarded and display returns 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.

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 parameters (arithmetic parameters)

5.4.1 Function



Function

Parameters are read and written by programs.
In this operating area, parameters can be altered manually.

5.4.2 Editing/deleting/finding R parameters



Function

The number of channel-specific R parameters is defined in a machine data.

Range:

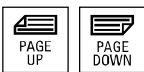
R0-999 (dependent on machine data).
There are no gaps in the numbering within the range.



Operating sequence

The "R parameters" window appears.
The channel-specific parameters are displayed.
The vertical softkey bar changes.

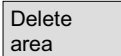
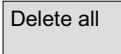

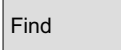
R
parameters



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

Change parameters:

Position the cursor bar on the appropriate input field and enter the new values.

Delete areaDelete allAbortOKFind**Delete parameters:**

Displays a screen in which you can specify the R parameter range Rx to Ry to be deleted.

The entire R parameter range is deleted, i.e. all values are set to "0".

Inputs are discarded.

Deletes the entered R parameter range.

Find parameters:

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.

When you press the "Input key", the cursor is automatically positioned on this parameter if it exists.

Other information

Input and deletion of parameters 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. These commands allow you to set up protection zones in the working area which are out of bounds for tool movements.



Operating sequence

Setting
data

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.



Other information

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 Revolutionary feedrate in mm (inch)/rev

JOG feedrate

Feedrate value in JOG mode

Continuous JOG

- JOG mode: The axis moves as long as the key is pressed.
- Continuous mode: The axis begins to move when the key is pressed once and continues until:
 - The key is pressed again,
 - NC Stop,
 - Reset,
 - Software/hardware 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 leading spindle
- Direction of rotation: Direction of rotation of leading spindle
- Spindle speed: Speed of the master spindle in Jog mode



Setting
data

Jog data

Operating sequence

Select softkey "Setting data".
The vertical softkey bar changes.

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

select a new value using the "Select" key.

**Other information**

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**Operating sequence**

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

select a new value using the "Select" key.

**Other information**

- 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 run DRY



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.

Operating sequence

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:

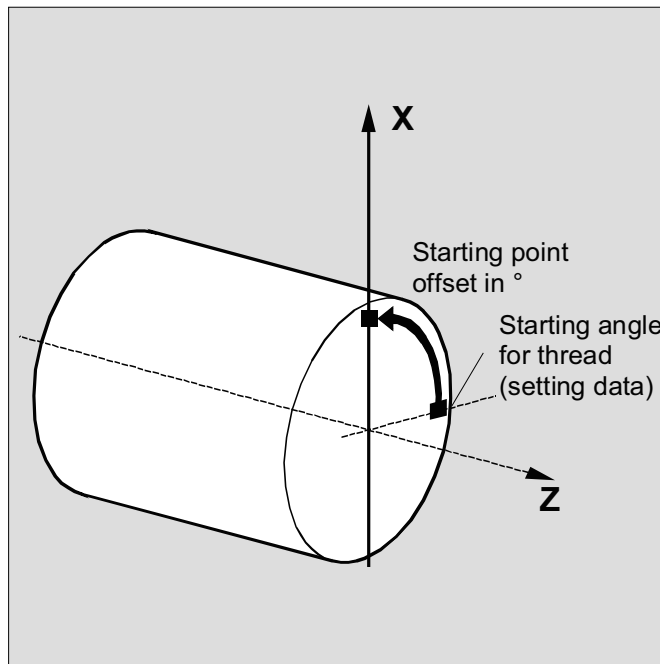
Enter a new value.

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.



Operating sequence

Setting
data

Starting
angle

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



Setting
data

Misc.

General
SD

Channel-
specific SD

Axis-
specific SD



PAGE
UP

PAGE
DOWN

Find

Find next

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.

Operating sequence

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 window "General setting data (\$SN_)" appears.
- The window "Channel-specific setting data (\$SC_)" appears.
- The window "Axis-specific setting data (\$SA_)" appears.

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.



Other information

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.



Operating sequence

Select softkey "Setting data".
The vertical softkey bar changes.

Setting
data

Protection
zones

Protection
zone +

Protection
zone -

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.

G17

G18

G19

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)

5.5.8 Electronic gears (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.

Operating sequence

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"

Coupling factor		Synchronization position	
Numerator	Denominator	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

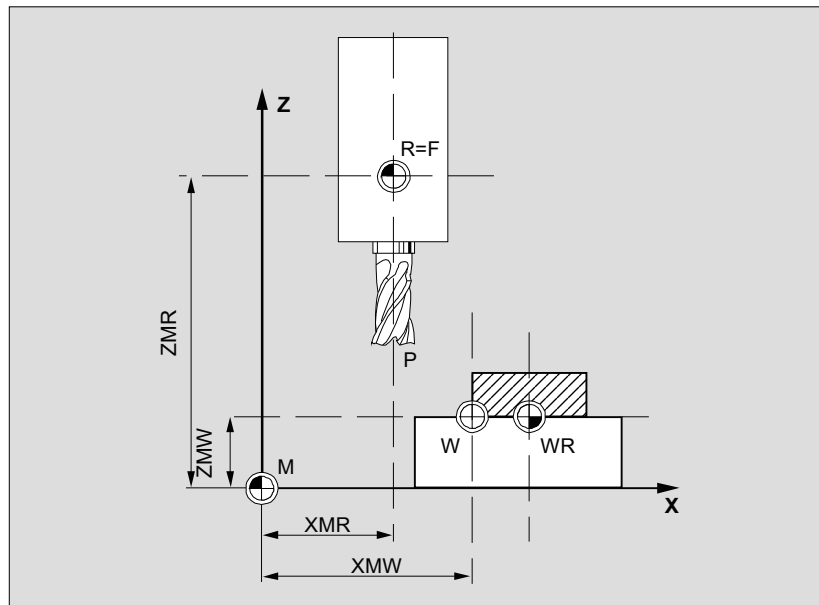
5.6.1 Function

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 part 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= . .` resulting 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 part 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: NC function which generates an incremental zero 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

- 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:

- Translation, TRANS, ATRANS
- Rotation, ROT, AROT
- Scale, SCALE, ASCALE
- Mirroring, MIRROR, AMIRROR

In the part program, all work offsets can be deselected non-modally with G53.



5.6.2 Changing the settable work offset (G54 ...)



\$P_UIFR[]

Function

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.



References

Rotation

Scale

Mirroring

Work
offset

Axes +

Axes -

Offsets

Rotation
scl, mirr

With global frames there is no geometrical relationship between the axes. It is therefore not possible to perform rotations or 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.

/FB/ K2: Axes, coordinate systems, frames

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.

The scale factor can be defined for the respective axis.

Mirroring of the relevant axis around the coordinate zero can be activated and deactivated.

Display and edit work offsets

Select softkey "Work offset".

The vertical softkey bar changes.

The display switches to the defined work offsets of the next axis.

The display switches to the defined work offsets of the previous axis.

You can use these softkeys to change the display mode of the currently displayed work offsets.

The following takes place:

- 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.

5.6 Work offset

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).

Other information

- 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 offsetActive WO
+ offset

Select horizontal softkey "Active WO + offset".

The vertical softkey bar changes.

Axes +

Displays the active work offset of the next axis.

Axes -

Displays the active work offset of the previous axis.

Offsets

Rot. dim.
mirr.

You can use these softkeys to change the display mode of the currently displayed work offsets.

Change
active WO

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 offset



Function

The active settable work offsets (selected from part program or MDI) can be displayed.



Operating sequence

Select softkeys "Work offset" and "Go to ...".
The vertical softkey bar changes.

Work offset Go to...

The "Active settable WO" window opens.
You can alter these values if necessary.

Active settable



/PGA/, Programming Guide, Advanced

Overview

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 Istwert[MKS] Istwert[ENS]		X1 [mm] 000.000 000.000	Y1 [mm] 000.000 000.000	Z1 [mm] 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	grob	399.000	100.000	000.000
	fein	1.000	0.000	0.000
	Drehung [Grad]	0.000	0.000	0.000
	Maßstab Spiegeln	1.000	1.000	1.000
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
- Overlaid movements
- DRF offsets
- Work offsets external
- 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.

Other information

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 offset



Function

The active, selected programmable work offsets (from part program or MDI) can be displayed.

Values cannot be edited in this display.



Operating sequence

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 work offset external



Function

The active, external work offsets can be displayed.
Values cannot be edited in this display.



Work
offset

Go to...

External
WO

Operating sequence

Select softkeys "Work offset" and "Go to ...".
The vertical softkey bar changes.

The "External WO" window is opened.

5.6.7 Display sum of the active work offsets



Function

The sum of the active work offsets from part program can be displayed.
Values cannot be edited in this display.



Work
offset

Go to...

Sum of
WO

Operating sequence

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:
 $\text{Sum WO} = \text{active settable WO} + \text{active programmable WO}$

The values can be changed only in the "Settable work offsets" menu.
(see Section "Settable work offsets" menu)

5.6.8 Immediately activate work offset and basic frame



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 part program is in the "Reset" state. This also occurs if the part 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 part program is continued.

Other information

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 the machine manufacturer's information.
/FB/ K2: Axes, coordinate systems, frames

Danger

The offset is applied the next time the part program is started.

5.6.9 Actual-value display: settable zero system, ENS



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 zero 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



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 user data (GUD) can be defined for HMI in the Services operating area (in the Definitions directory); reinitialization is not necessary.

The following applies:

- Definition files that are 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 as a component of their name `GUD`.

The display of global user data (GUD) can be locked by means of the keyswitch or a password.

For more information on how to protect global user data in the part program, please refer to

/PGA/, Programming Guide, Advanced
Section: File and Program Management



5.7.2 Changing/finding user data/user variablesUser
dataGlobal user
dataChannel-sp
user dataProgram
user data

GUD +

GUD -

GUD:

**Operating sequence**

Press softkey "User data".

The "Global user data" window is displayed.

The vertical softkey bar changes.

You can toggle between windows

- "Global user data" (GUD)
- "Channel-specific user data" and

- "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.

Search user data

Press softkeys "GUD +" and "GUD -" to scroll through user data from GUD 1 to GUD 9.

The "Select global user data" window is opened. The following values are permissible:

- 1 = SGUD (Siemens)
- 2 = MGUD (machine manufacturer)
- 3 = UGUD (machine user)
- 4 ... 9 = GD4 ... GD9 (additional, e.g. grinding cycles, etc.)

The selected data are displayed in the "Global user data" window.

Other information

Up to 200 characters can be entered/displayed for GUD of type String.

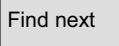
Find

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.

Other information

A description of how to define and activate user data can be found

- in Chapter "Services" Operating Area.

5.8 Display system variables



Function

System variables can be used for a wide range of different functions (e.g. as variables or in synchronized actions). The following are possible:

- View variables in a defined display (e.g. or
 - Managing variable views
 - Display variables of a view
 - Define views of variables
- Generate a log of the response of variables during a program run
 - Define logs
 - Start log
 - Manage a log
 - Display log



Operating sequence

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

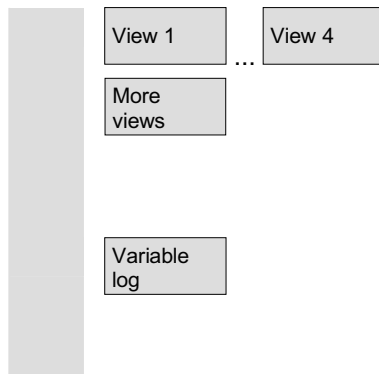
* These softkeys can be named user-specifically.

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.

Edit
view

Select
view



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 Creating/editing 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 with 2 rows) and
- Properties (e.g. name, display type, input limits).



Operating sequence

The softkey "Edit view" changes to edit mode.

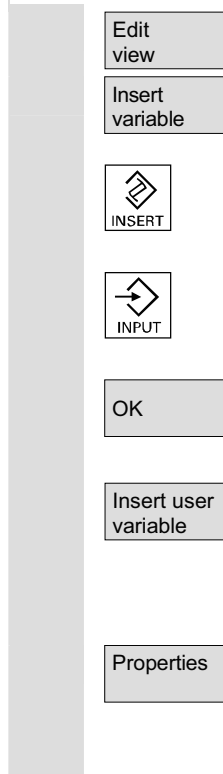
With "Insert variable", a dialog box is opened in which the required system variable can be selected from a list with the cursor and the "Edit" key. The full name of the variable appears in the info line.

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. These user variables are available in the respective GUD block even when no GUD definition files are activated. The NCK does not assign a \$ as initial character.

The softkey "Properties" opens a dialog box in which the text displayed with a variable can be altered.



The following properties can be altered:

- **Identifier** Assign variable name,
- **Displayed OPI variable** Assign a different variable,
- **Variable type**

REAL	Number of places after the decimal point:
CHAR	
STRING	text length:
BOOL	
VARIANT	Selection for format
INT	Format: Decimal B, W, D, BU, WU, DU
	Binary B, W, D
	Hex BU, WU, DU

Access level 0 to 7 are possible

- **Limit value check** Min + Max only Min or only Max or list entries
- **Text:**

Dialog line text	Long text on right next to header
Graphic text	Text on left next to I/O field
Text for units	Text on right next to I/O field
Screen text	Displayed variable text
- Text orientation and text positioning,
(when 0 values are entered, the field is automatically set to default values.)
- Text size (character size) and display type
- **Path for help screen file**

Path for help text file	
Index in the help text file	Search text

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 Manage variable views



Edit
view

Manage
views

New
file

File
contents

Delete
file

File
contents

Display
view

New
view

Delete
view

Save
view

Function

The views you create are stored and managed in file form.

Operating sequence

With the softkeys "Edit view" and "Manage views" you here can open a window in which files and views can be

- created
- display and
- deleted.

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 Log 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.

5.8 Display system variables

Recording of synchronized action variables can be limited to the event with identification number 1.

This event records variables in the IPO cycle or

- 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.

Operating sequence

The "System variable display" appears.

The softkey "Variable log" opens the screen with the heading "Logging system variables".

The softkey "Insert variable" opens a dialog box in which the system variable to be recorded can be selected.

The softkey available from SW 6.4 "Insert user variable" opens a dialog box where you can select any available user variables.

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.

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".

You can start logging by pressing the button "Start log" or with the system variable `$A_PROTO=1` in the part program.

You can stop logging by pressing the button "Stop log" or with the system variable `$A_PROTO=0`.

After stopping, the log memory is automatically downloaded to a file.



System
variable

Variable
log

Insert
variable

Insert user
variable

OK

Initial.
log

Start log

or

`$A_PROTO=1`

Stop
log

or

`$A_PROTO=0`

Manage
log

Graphic
log

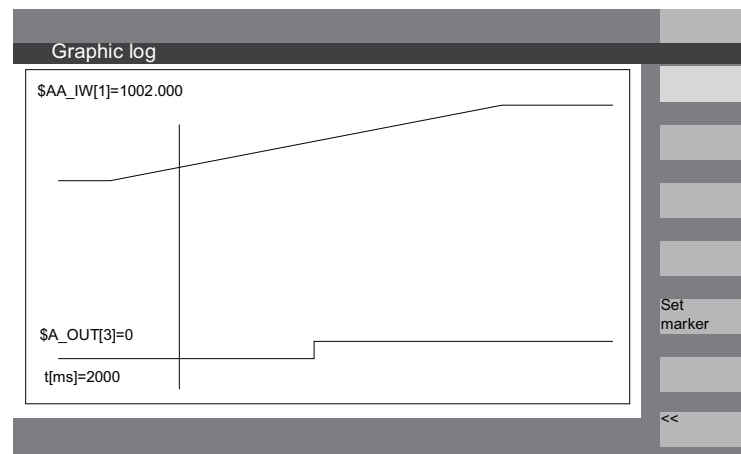
Set
marker

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.

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.



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.

Notes

"Program" Operating Area

6.1	Program types	6-263
6.1.1	Part programs	6-263
6.1.2	Subroutines	6-263
6.1.3	Workpieces	6-263
6.1.4	Cycles	6-263
6.1.5	Templates	6-263
6.2	Storing programs.....	6-267
6.2.1	HMI.....	6-267
6.3	Program main screen.....	6-267
6.4	Edit programs.....	6-269
6.4.1	Text editor	6-269
6.4.2	Undo and redo in the editor (from SW 6.3).....	6-272
6.4.3	Selective program protection: *RO*	6-273
6.4.4	Reserved character strings	6-274
6.5	Free contour programming	6-275
6.5.1	General	6-275
6.5.2	Program contour	6-276
6.5.3	Contour elements.....	6-280
6.5.4	Graphic representation of the contour	6-281
6.5.5	Input screens for setting parameters for contour elements	6-282
6.5.6	Specifying contour elements in polar coordinates, closing the contour.....	6-285
6.5.7	Undercuts in turning technology	6-290
6.5.8	Help	6-293
6.5.9	Parameter description of straight line/circle contour elements	6-294
6.5.10	Examples of user-defined contour programming.....	6-295
6.5.11	Cycle support	6-298
6.6	Display in the step editor (option from SW 6.4)	6-299
6.7	Program relationship in the dual editor (from SW 6.4).....	6-302
6.7.1	Views in the dual ASCII editor	6-302
6.7.2	Synchronized view in the dual ASCII editor	6-302
6.7.3	Synchronized display in the dual-step editor	6-303
6.8	Multi-channel step display (option from SW 6.4)	6-304
6.9	Program simulation	6-311
6.9.1	Simulation operation	6-312
6.9.2	Simulation settings	6-323
6.9.3	Setting downtimes.....	6-328
6.9.4	Display and colors.....	6-329
6.9.5	Section by section simulation.....	6-330
6.9.6	Multi-channel, section by section program simulation	6-331
6.9.7	Simulation with orientable toolholder	6-334
6.9.8	Quick display in the simulation for mold making.....	6-335
6.9.9	Simulation with external network drive.....	6-337

6.10	Manage programs	6-338
6.10.1	Overview.....	6-338
6.10.2	NC data types and directories	6-339
6.10.3	New workpiece/part program	6-340
6.10.4	Saving setup data.....	6-343
6.10.5	Select program for execution	6-344
6.10.6	Load/unload program	6-347
6.10.7	Load/unload program	6-348
6.10.8	Copy/paste	6-349
6.10.9	Delete	6-352
6.10.10	Rename	6-353
6.10.11	Enabling.....	6-355
6.10.12	Log.....	6-356
6.11	Accessing an external network drive/computer	6-357

6.1 Program types

6.1.1 Part programs



A part 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 Subroutines



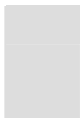
A subprogram is a sequence of instructions in a part program which can be called repeatedly with different defining parameters. Cycles are a type of subroutine.

6.1.3 Workpieces



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.

6.1.5 Templates



Templates are objects that can be used for creating new workpieces, part programs, subroutines, job lists, work offsets, etc. and can be edited and customized to suit the task at hand. With HMI Advanced, templates for job lists are stored in the file system by default. Specific user templates must first be created in template directories. This allows you to access them when creating new objects.

Templates can be designed in different languages and named to allow easy language-specific access to them. If you create a template with a wildcard in the name, the current workpiece name will be inserted into the name when the template is activated.

There is no difference between **creating templates** and creating workpieces, part programs, subroutines, job lists, work offsets, etc. You can use the administration functions and naming conventions to enter objects you want to use as templates in the template directory. Templates can be created from the Services operating area directly in the required directories. See below.

Existing templates are offered for selection when you create new objects.

Naming conventions for templates

TEMPL

If this string is assigned as part of the template name, it is replaced by the workpiece name when the template is activated. The template is not copied if the name exceeds 24 characters after replacement.

_XX

If the template name terminates in **_XX** (XX stands for the language code, e.g. **_GR** for German), only the objects are offered from the template directory where the current language matches the language code in the template name. Templates for different languages can therefore be stored together in the template directory.

1_1

If the template name starts with **1_1**, the template is proposed for standard configuration.

M_N

If the template name starts with **M_N**, the template is proposed for M:N system configurations (netnames.ini is configured).

1_1 or **M_N** can be **combined** with **_XX**.

Other information

Language-specific templates are given priority over non-language-specific templates with the same core name component. Therefore, you can create a default template which can be used for all languages for which no language-specific variants are available.

Directories for templates

The Services operating area contains the following subdirectories under Templates:

User	Empty when shipped
Manufacturer	Empty when shipped
Siemens	Contains standard job lists when shipped

Templates	Empty when shipped
-----------	--------------------

If the **Templates** subdirectory for this data type is not empty when creating a new data type, you can enter "No template" to access the templates in the user, manufacturer and Siemens directories.

The templates in User, Manufacturer, Siemens are offered for use with the individual datatypes with:

Startup→MMC→System settings→Templates

The activation differs according to:

- Job lists
- Part programs/subroutines
- Initialization programs

In the **Templates** directory, you can store:

- Individual templates for various data types and
- **Complete workpieces** as templates in a subdirectory. The full workpiece templates are offered for selection when data type WPD is newly created. If selected, **all** components belonging to the template are copied to the new workpiece in accordance with the above naming conventions.

With other data types only the selected template is used.

Creating templates

In the **Services** operating area select the directory for which you are authorized.

Templates/Users
/Manufacturers
/Siemens
/Templates



Define the name, data type and - if required - template.

Confirm with OK.

Use the editor to process the created template.

You can use the administration functions to copy and insert template objects to the directories and use the above mentioned naming conventions and rename the templates e.g. for specific languages or workpiece names.

6.1 Program types

Saving templates

You can save existing templates under Services "Data out" or read in saved templates again with "Data in".

6.2 Storing programs

6.2.1 HMI



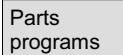
Programs can be saved in the NC memory or on hard disk. The currently available memory space is displayed in the dialog line. Cycles can also be stored in the NC Flash File System.

6.3 Program main screen



The Program basic display contains a complete overview of all workpiece and program directories.

Program		CHAN1	Mda																															
<input checked="" type="checkbox"/> Channel reset	Program aborted																																	
Program overview <table border="1"> <thead> <tr> <th>Name</th> <th>Type</th> <th>Loaded</th> <th>Length</th> <th>Date</th> <th>Enable</th> <th></th> </tr> </thead> <tbody> <tr> <td>HUG0</td> <td>WPD</td> <td></td> <td></td> <td>01.06.2000</td> <td>x</td> <td></td> </tr> <tr> <td>WKS</td> <td>WPD</td> <td></td> <td></td> <td>01.06.2000</td> <td></td> <td></td> </tr> <tr> <td>WKS</td> <td>WPD</td> <td></td> <td></td> <td>01.06.2000</td> <td></td> <td></td> </tr> </tbody> </table>							Name	Type	Loaded	Length	Date	Enable		HUG0	WPD			01.06.2000	x		WKS	WPD			01.06.2000			WKS	WPD			01.06.2000		
Name	Type	Loaded	Length	Date	Enable																													
HUG0	WPD			01.06.2000	x																													
WKS	WPD			01.06.2000																														
WKS	WPD			01.06.2000																														
Free memory: Hard disk: <input type="text"/> NCU: <input type="text"/>						New... Load HD->NC Unload NC->HD Simulation Manage programs... Select Save setup data																												
Press INPUT key for program overview																																		
Work- pieces	Parts programs	Sub- programs	Standard cycles	User cycles	Manufact. cycles																													


 Parts programs

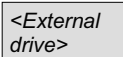
Horizontal softkeys

An overview of all part programs (main programs) stored in the selected directory is displayed.


 Sub-programs

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".


 <External drive>

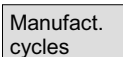
If external drives are configured, up to three softkeys can be found in the softkey menu with the configured drive/computer designation. See softkey assignment below.


 Standard cycles

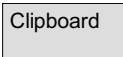
Select softkey "Standard cycles" to display a list of the cycles that Siemens has added. See softkey assignment below.


 User cycles

Select softkey "User cycles" to display a list of the user cycles that you have added.


 Manufact. cycles

Select softkey "Manufacturer cycles" to display a list of the cycles that have been integrated by the manufacturer.



 Clipboard

After pressing the "ETC" key, you can display a list of the programs/data stored in the clipboard by pressing softkey "Clipboard".



 Log

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 you have configured four softkeys, the fourth one is displayed in the More bar when you press the "ETC" key. 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


 New ...

Creates a new file for a workpiece/part program.

If activated templates exist for the created data type, they are proposed. If they are accepted, the copy of the template becomes the created object.

Load HD->NC	The highlighted program (or programs) is (are) loaded from the hard disk to the NC memory.
Unload NC->HD	The highlighted program is unloaded from the NC memory to the hard disk.
Simulation	With the function "Simulation" you can display axis movements graphically and simulate the results of machining on the screen. See also 6.6.
Manage programs...	With the softkey "Manage programs" you can manage the functions "New", "Copy", "Insert", "Delete", "Rename" and "Enable".
Select	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 Edit programs

6.4.1 Text editor



Chapter 2, "General operating sequences"

- The editor displays only the characters which can be input via the operator panel front keyboard.
- A part 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 Subsection 6.2.1 "HMI".

Product variants

- You can use the editor as standard to edit the instructions in part programs in G code, keywords, etc. The editor is turned into a "dual editor" by displaying 2 windows with part program instructions side by side".
How to use the editor with 2 work panes is described in Section 6.7 "Program relationship in the dual editor".
- As an option, the editor can be used with extended functions. The extended functions allow you to process technology-oriented program steps which are displayed with special icons in the editor for each step type.

Steps are the following:

- Groups of individual instructions: G codes, vocabulary words etc.
- Synchronization instructions (e.g. WAITM)
- Cycles
- Contour definitions

The advantages are:

- Improved readability of programs
- Operations based on steps (copy, paste, delete etc.)
- The dual editor allows transparent, straightforward use of synchronization instructions in different programs.
- Step display provides the conditions for using multi-channel step editors (Multi Channel Step Editor, MCSE)
- Recompilation from step to DIN code, contour definition and cycle display

Step representation is described in

Section 6.6 "Representation in the step editor".

- Functionality of a multi-channel step editor is available as an option.

It provides the following features:

- Simultaneous display of multiple programs for a workpiece in step representation in multiple channels
- Either fixed step length or time-proportional representation of step lengths in the channels
- Display of processing duration for steps and wait time at synchronization points

Prerequisite for time calculation:

- Actual execution of the programs in the individual channels

Advantages of multi-channel display:

- Benefits of step representation plus
- Assessment of synchronization between the channels, optimization possibilities using concrete execution times
- Fast changeover between the programs of the different channels

Important: The time calculation must be performed again after any changes were made to the program.

- For more information on multi-channel step display, see Section 6.8 "Multi-channel step display".

6.4.2 Undo and redo in the editor (from SW 6.3)

**Edit****Function**

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:

- Press "Ctrl" and "Z" for "Undo"
- Press "Ctrl" and "Y" for "Redo"

**"Ctrl" + "Z"****Operating sequence**

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 functions, certain machine-specific code lines may be protected against changes.

A read-only identifier (";*RO*") is tagged onto the code blocks as a comment. The ASCII editor recognizes these blocks, hides them or displays them in the read-only text color (gray), and inhibits changes to these blocks.

```

Editor                                     \MPF.DIR\HARR.MPF
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 change a program part protected by the read-only identifier is denied with the message "Block cannot be written".

Other information

When creating a program template, remember to position the "Read Only" identifier directly at the end of the block.

6.4.4 Reserved character strings

Reserved character strings occur in part program code lines which are generated by calling cycles and programming contour definitions. They can be viewed by setting "Display hidden lines" in the editor. The following character strings must not be used in part program lines that are directly input:

```
;#  
;#END  
;NCG  
;*RO*  
;*HD*
```

Other information

See selective program protection: *RO*

6.5 Free contour programming

6.5.1 General



Function

The free contour programming is a support tool for the editor.

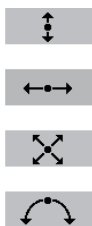
The contour programming function enables you to create simple or complex contours.

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 part 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:

- Starting 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.



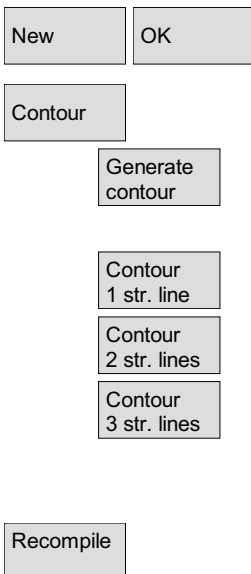
Other information

1. The valid geometry axes in the first channel are determined and used in the part 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 Program contour



New contour



Sequence

Use softkeys "Workpiece" and "Part program" in the Program operating area to select an existing program and press the "Input" key or select softkey "New" to open a new part 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

Upon recompilation only the contour elements that were created with free contour definition are regenerated. In addition, only the texts that were added using the input field "free text input" are recompiled. Any changes you made directly in the program text are lost. However, you can insert and edit user-defined afterwards, which will not be lost.



Define the starting point

The input screen form for the contour **start point** is displayed.

The screenshot shows the 'Start point' input screen. The main area is a coordinate system with X and Z axes. A red dot is placed at the starting point (X=20.0000, Z=30.0000). The 'Spec. for facing axis' section has 'Radius (DIAMOF)' selected. The 'Approach start point' is set to 'G1'. The 'Free text input' field is empty. The screen also displays 'Program: CHAN1', 'Auto: MPF0', and 'Program aborted'.

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.

Select softkey "Accept element" to store the starting point. You can add the next element by selecting the appropriate softkey:

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.

Accept
element

Alternative

1

0

...

Accept
element**Contour element
softkey texts**All
parameters

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.

The values you have entered are accepted when you select "Accept element", you can add the next element by selecting the appropriate softkey.

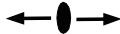
Beginning at the starting point, enter the first contour element, e.g. a straight line. Input all the data specified on the workshop drawing: Length of straight lines, end position, transition to following element, angle of pitch, etc.

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.

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.

Vertical line



Arc with any direction of rotation

Contour transition elements

Abort

Delete
value

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 is always appended to the end of a contour element. You select transition elements in the parameter input screen form for the relevant contour element.

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.

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

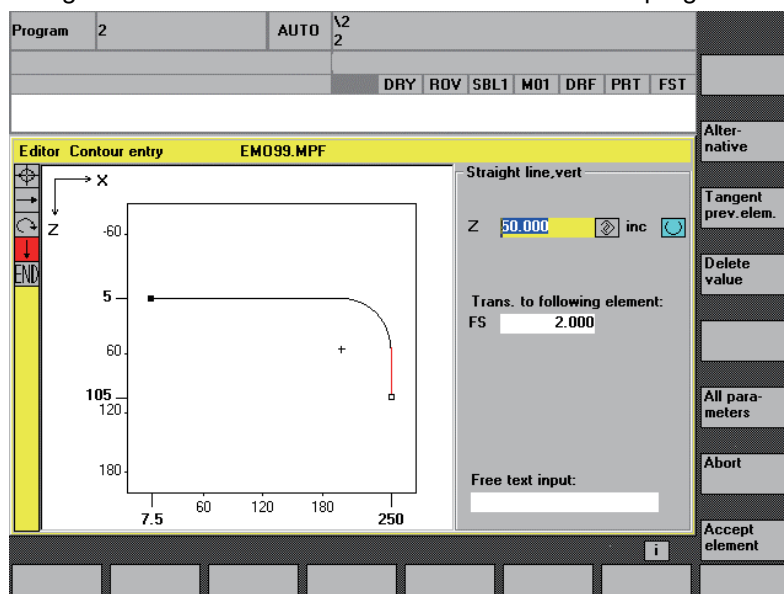


Function

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.

6.5 Free contour programming



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 screens for setting parameters for 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 0 degrees.

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.

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".

Tangent to previous element

Tangent
prev. elem.

Display additional parameters

All
parameters

Alternative

Select dialog

Select
dialog

Select
dialog

Accept
dialog

Changing a selected 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!

Clear 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.

Append 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

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.

Alter a contour element

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.

Insert contour element

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.

A rectangular button with a thin border containing the text "Accept element".

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.

Delete a contour elementA rectangular button with a thin border containing the text "Delete element".

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 inputA rectangular button with a thin border containing the text "Abort".

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



Pole

Entering polar coordinates

More

Pole

Close
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 re-specified 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.

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.

Polar coordinates

The softkey "More" in the basic plane of the contour programming accesses the "Pole" screen form and the softkey "Close contour".

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.

The contour is closed by a straight line between the last entered contour point and the starting point.

Other information

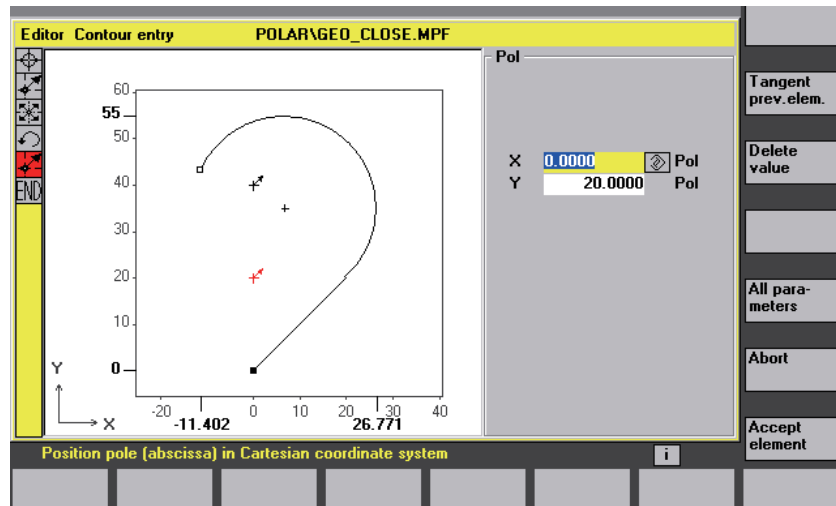
If the straight line that was generated with close contour is linked to the start element of the contour with a radius or chamfer, the radius or chamfer must be specified explicitly as follows:

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

Absolute/incremental
input



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)

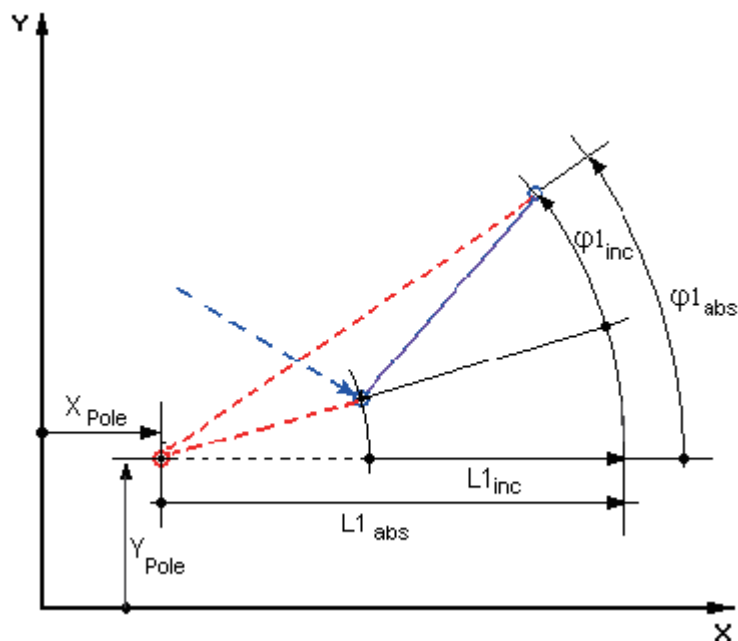
or as polar. For the Cartesian/polar switchover, additional toggle fields are displayed for "Straight line, any" and "Circular arc" in both the basic contour input view and the view with "All parameters".

A toggle field is not displayed if a pole does not exist. Input fields and display fields are then only available for Cartesian values.

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 $0^\circ \dots \pm 360^\circ$. When absolute dimensions are specified, the angular reference is based on a horizontal axis of the working plane, e.g. X axis with G17. The positive direction of rotation is counter-clockwise.

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.

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
 $X_{\text{abs}} = 8.6603$ $Y_{\text{abs}} = 5.0$

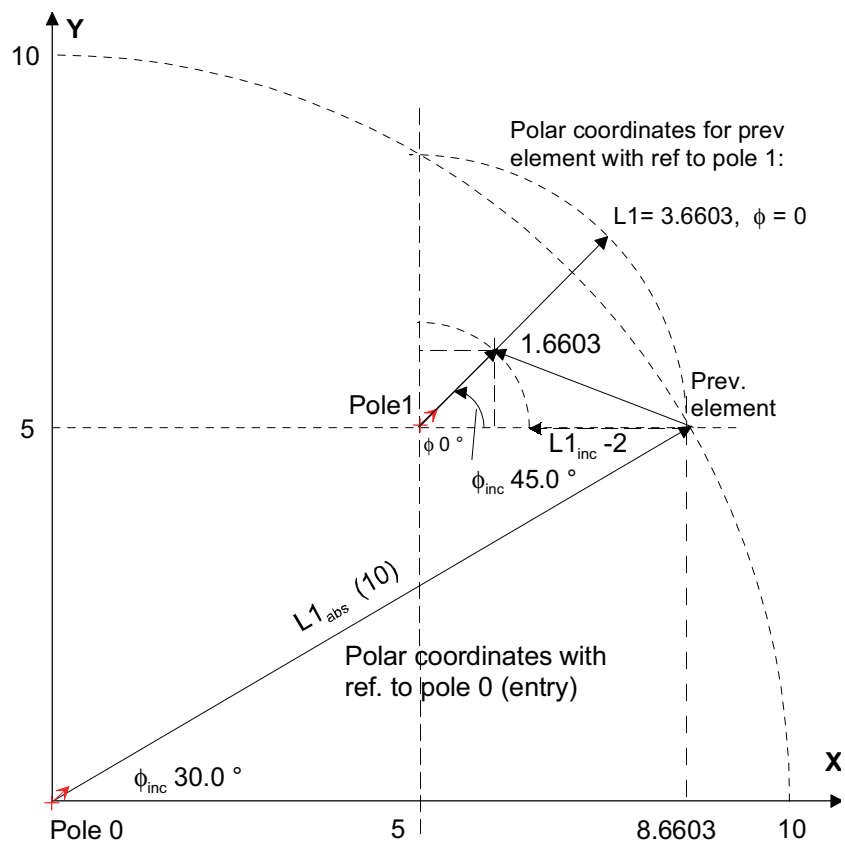
New pole:
 $X_{\text{Pole1}} = 5.0$ $Y_{\text{Pole1}} = 5.0$ (Pole 1)

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:

Program: CHAN1, Auto: MPF0, Channel reset: Program aborted, ROV: SBL1

Editor: Contour entry, POLAR\POLAR.MPF

Circular arc:

- R: 34.9132
- Polar: L1: 34.9130, φl: 200.0520
- Cartesian: I: 50.0000, J: 50.0004
- Trans. to following element: FS: 0.0000
- Free text input: : Kreisbogen links

End point distance to pole absolute

The code created in the part program is as follows:

Program: CHAN1, Auto: MPF0, Channel reset: Program aborted, ROV: SBL1

Editor: POLAR\POLAR.MPF

```

; 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

Restrictions

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°. 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).

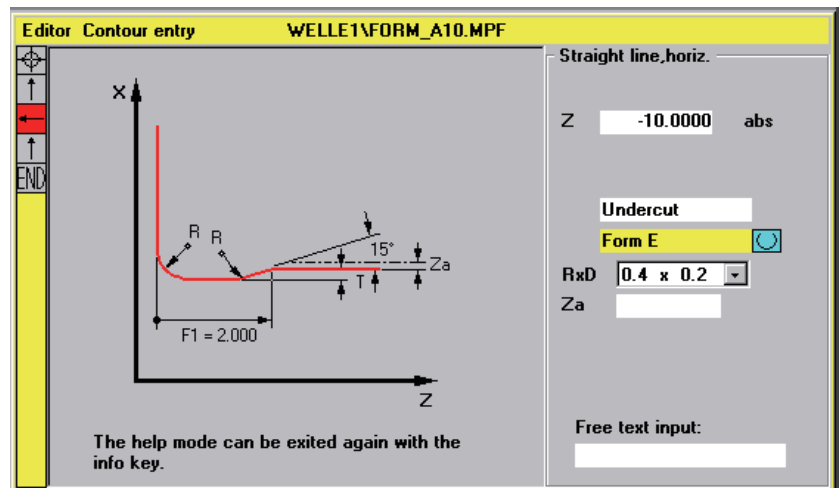
Operation

When the operator focus is on "Trans. to following element", use the Select key or softkey "Alternative" to select Undercut.

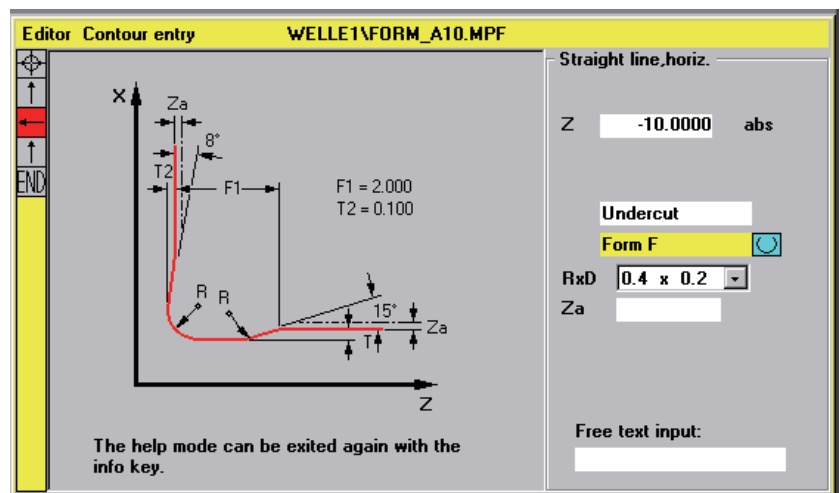
When the focus is on the following field, the undercut form can be defined. The Select key and softkey "Alternative" can be used to select the following options:

- Form E
- Form F
- DIN 76 thread
- General thread

Form E



Form F

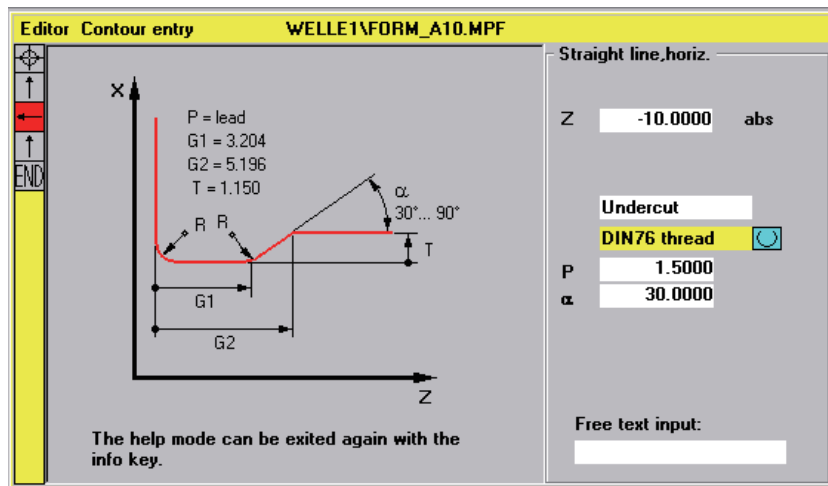


Operation (continued)

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.

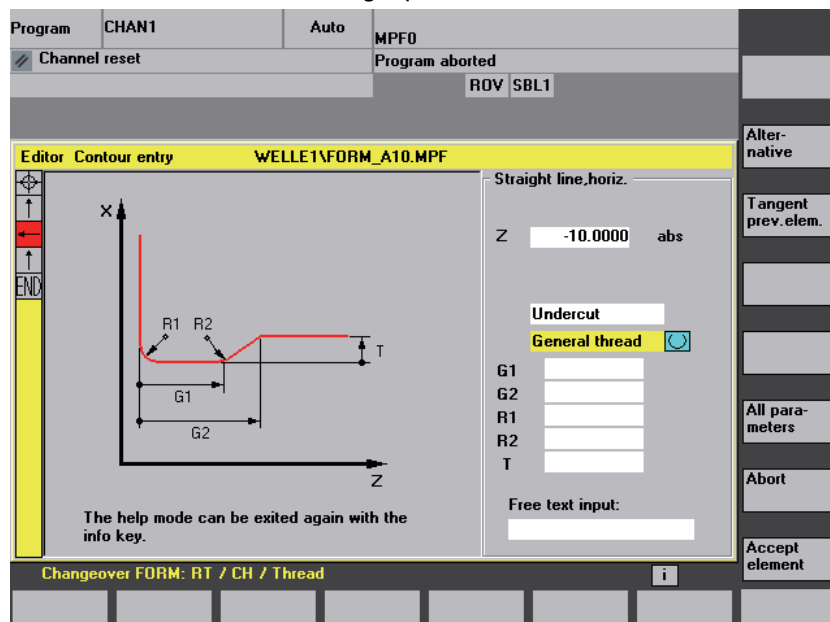
DIN thread



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:



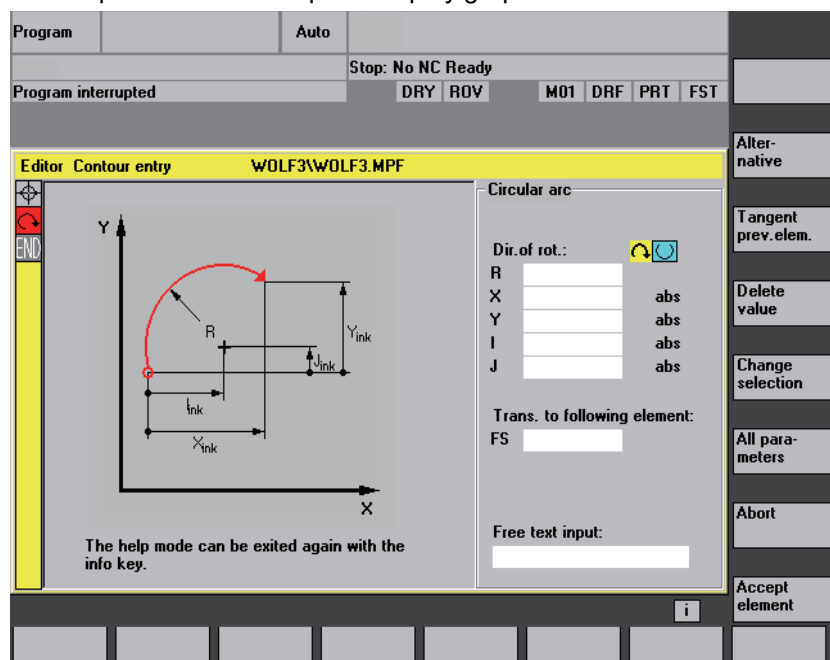
6.5.8 Help



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:

- Starting point
- Vertical line
- Straight vertical line, angle entry field
- Horizontal line
- Straight horizontal line, angle entry field
- Straight line in any direction
- Straight line any, angle entry field
- Circle
- Circle, angle entry field
- Radius/chamfer

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
$\beta 2$	Angle of aperture 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

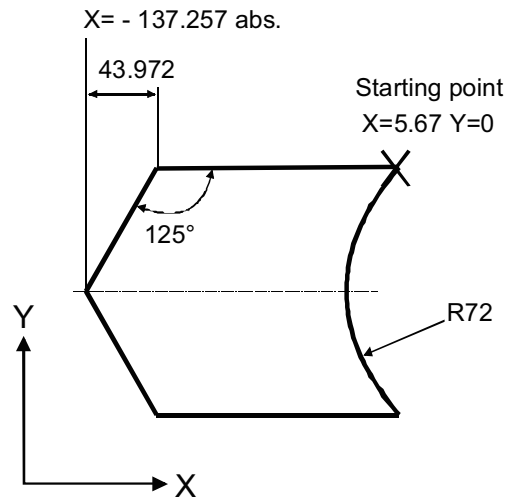


Example 1

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	Parameters	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., Select dialog	

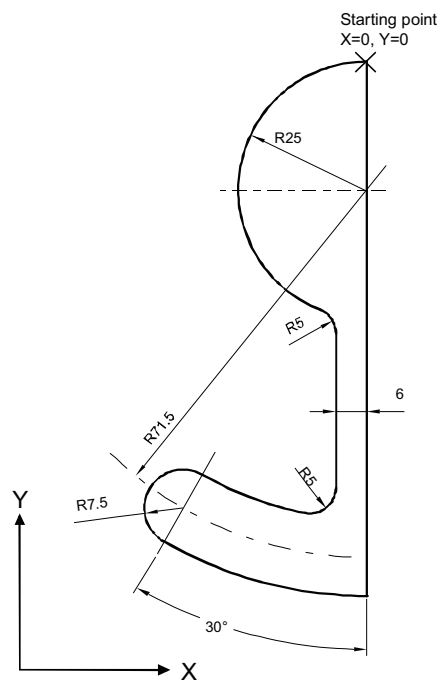


Example 2

Workpiece drawing of contour

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".



Element	Softkey	Parameters	Remarks
1		Y=-104 abs.	
2		Direction of rotation right, R=79, I=0 abs., Select dialog, all parameters, $\beta_2=30$ degrees	
3		CW direction of rotation, tangent to preced. R=7.5, all parameters, $\beta_2=180$ degrees	
4		CCW direction of rotation, 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		CW direction of rotation, R=25, X=0 abs., Y=0 abs. I=0 abs select dialog, select dialog.	

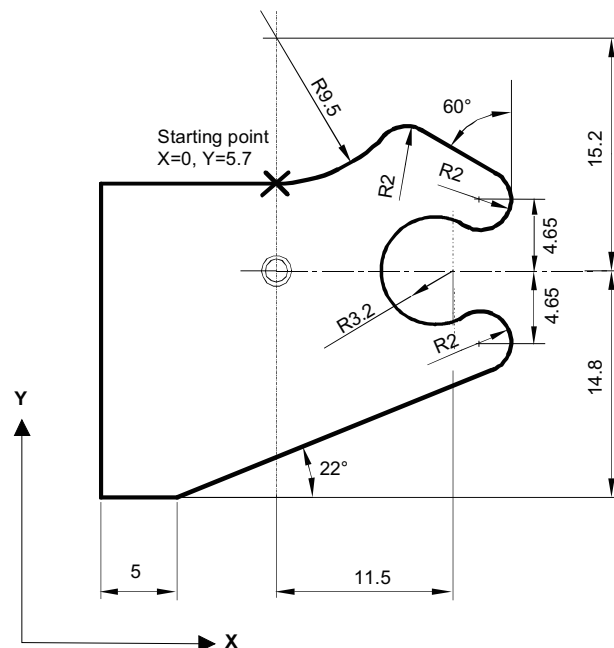











Example 3

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., select dialog	
6		Tangent to previous element $\alpha_1=-158$ degrees, $Y=-14.8$ abs., $\alpha_2=0$ degrees	Observe angles in help screen!
7		All parameters, $L=5$, select dialog	
8		$Y=5.7$ abs.	
9		$X=0$ abs.	

6.5.11 Cycle support



Additional tools in the form of prepared cycles requiring parameterization only are also available for the following technologies:

- Drilling
- Milling
- Turning

Cycles Programming Guide

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").

Other information

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

6.6 Display in the step editor (option from SW 6.4)

Introduction

You can either edit a part program in step display (step editor) or in standard display (ASCII editor). The function is available for multi-channel systems as an option: "Multi-channel sequencer programming". The file **SEEDITOR.INI** and instruction "SEFORM" can be used to structure the NC programs.

SEEDITOR.INI

Existing NC programs can be displayed in step representation using the default settings in the file **SEEDITOR.INI**.

The following are interpreted as steps:

- Standard cycles
- Contours (geometry processor, contour definition)
- G code
- Wait markers (WAITM)
- End (data end, no M17, M02, M30 command)



SEEDITOR.INI configuration see also

/IAM/ IM4: Installation and Start-up Guide, Section 5.3 Parameter Assignment with ini Files

SEFORM instruction

The function SEFORM is provided for the step editor for structuring the NC programs as step programs

("step name", level, icons) and is used for:

- Naming steps
- Assigning steps to another level (0=main level, 1=sublevel)
- To mark steps with icons for display
- Delimit steps.

They are visible in the default (standard) display. SEFORM instructions have no effect during program execution in the NC and during simulation.

Select step display



Operating
step

To display a program selected in the editor as a step program, press the 2nd horizontal softkey "Operating step" of the expansion key. A new program only displays the empty step "End". The new program steps must be inserted before this end step.

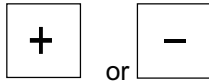


Other information

If a program contains a SEFORM instruction in the first 500 lines, step display is automatically activated. Without option, you can only edit a program with SEFORM instructions in standard display.

When you switch over in the dual editor, both editor panes switch over to step display. When you switch back to the standard ASCII editor, both panes change to ASCII display.

Step display variants



Collapsed step display

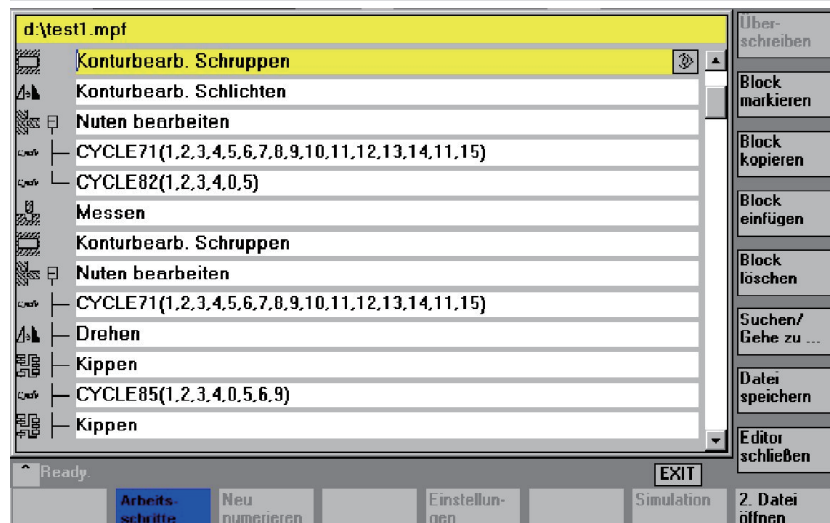
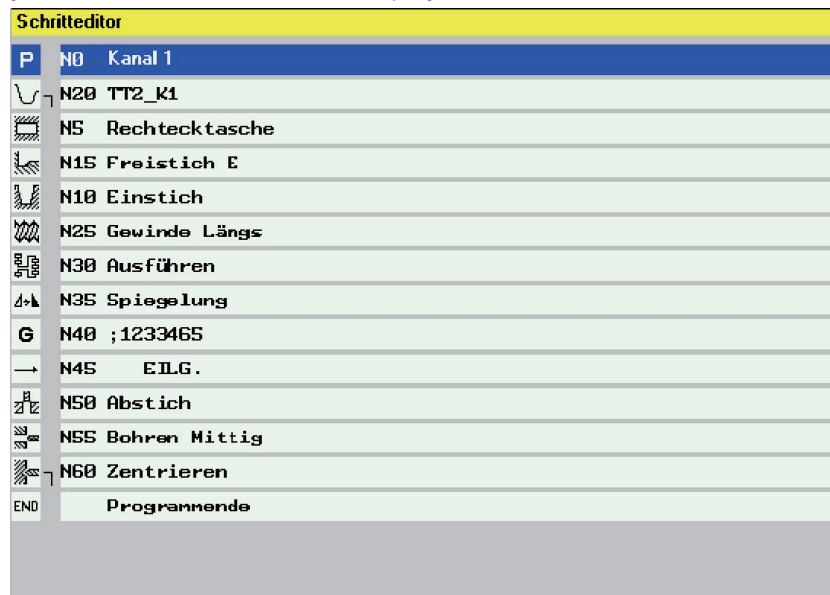
Expanded step display

Combined step and standard editor view is not supported.

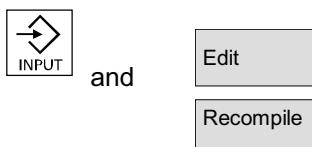
Two levels can be displayed in step view. Display is either expanded for the entire step editor (main level and sublevel) or collapsed (main level). By pressing the

- Keys + and
- Character –

you can switch between both display modes.



Step editor operation



Operation is virtually identical to that of the ASCII editor, the only different key functions for the step editor are described below:

- Steps like G code and wait markers are opened with the ASCII editor.
- Standard cycles and contours (geometry processor, contour definition) are recompiled.

The full scope of editor functions is no longer available for the ASCII editor.

6.6 Display in the step editor (option from SW 6.4)



OK

CTRL

I

In step representation

Mark
blockCopy
block

Press INSERT to insert a new step before the currently selected step. The editor window is automatically opened. The new step becomes visible in the step display via the step name and - if configured - the associated icons.

Default text is free DIN code with G icons.

New steps are also inserted via functions such as cycle support.

Pressing the keyboard shortcut CTRL and I will insert a predefined SEFORM instruction in the SEditor.INI.

The **step identifier** is defined as:

SEFORM("step name", level, "icon") with

Step name Text is step display

Level 0 Main level, 1 sublevel (default = 0).

Icon Fine name of the icons as defined in the configuration
Default = Default icon, if configured, otherwise blank
(see SEdit.ini in /IAM/, IM4)

There are two types of notation for the icon:

Without path HMI searches for the icon

With path HMI opens the icon with the specified
absolute path.

Each new work step which is started on the same level with SEFORM, is the end of the preceding work step for this level.

Instead of the step name strings you can also write the text numbers which are used in the file with the configured (alarm) texts. Enter them in the format \$8xxxx. They are then displayed in the currently active language.

With **collapsed** display, the entire step is highlighted with all substeps and can be copied or deleted as required.

With **expanded** display, the selected steps are highlighted.

Note

If you have switched between expanded and collapsed view while work steps were highlighted, the highlighting is removed for the work steps.

All selected steps are copied to the clipboard and can be inserted anywhere in the active editor window or in the second editor window. The clipboard is valid for each operating area at a time. An editor/dual editor or multi-channel editor (see Section 6.8) which is e.g. activated by the simulation or as a correction editor cannot access the clipboard which contains data created in the Program operating area.

6.7 Program relationship in the dual editor (from SW 6.4)

Find/Go
to...

Find: If the search term is located in the step identifier, the cursor is positioned on the step. If the search term is not found within a step, a message is displayed "Search text not found".

Go to: If the block number is located within a step, the cursor is positioned on the main level or sublevel of the step.

6.7 Program relationship in the dual editor (from SW 6.4)

6.7.1 Views in the dual ASCII editor

Find/Go
to...

View
subroutine

If you are positioned on a subroutine call in the **ASCII editor**, this program is loaded to the second editor when you press the "View subroutine" softkey.

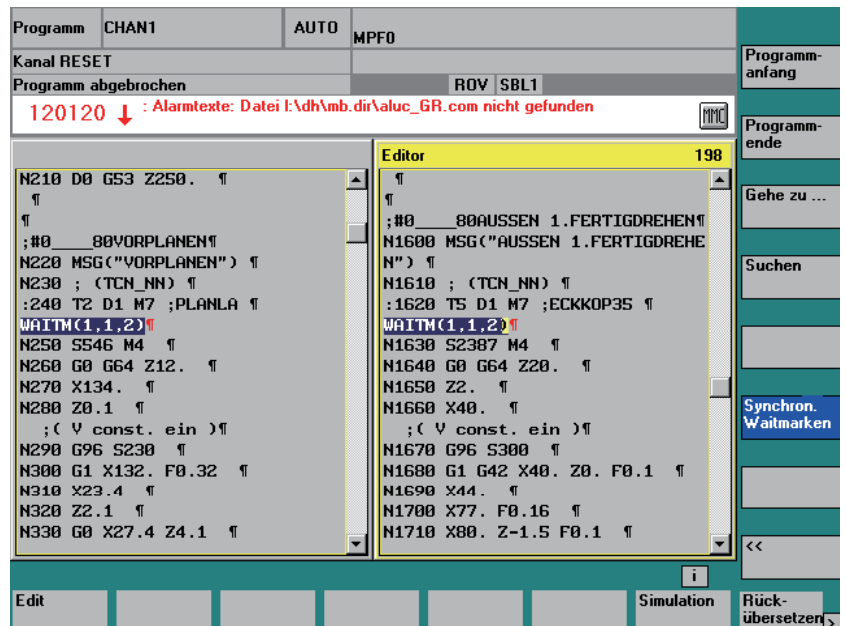
The function is only available in the ASCII editor.

6.7.2 Synchronized view in the dual ASCII editor

Find/Go
to...

Synchr.
wait markers

You can view instruction groups in the **ASCII editor**. Position the cursor on a Wait marker instruction in one of the two ASCII editors. Press the softkey "Synchr. wait markers". This string is searched in the second editor and displayed at the same height in the window with the cursor positioned in the first editor. The focus remains in the first editor. If no wait marker instruction is found together with its brackets, the next wait marker instruction is displayed in the adjacent window regardless of the brackets.

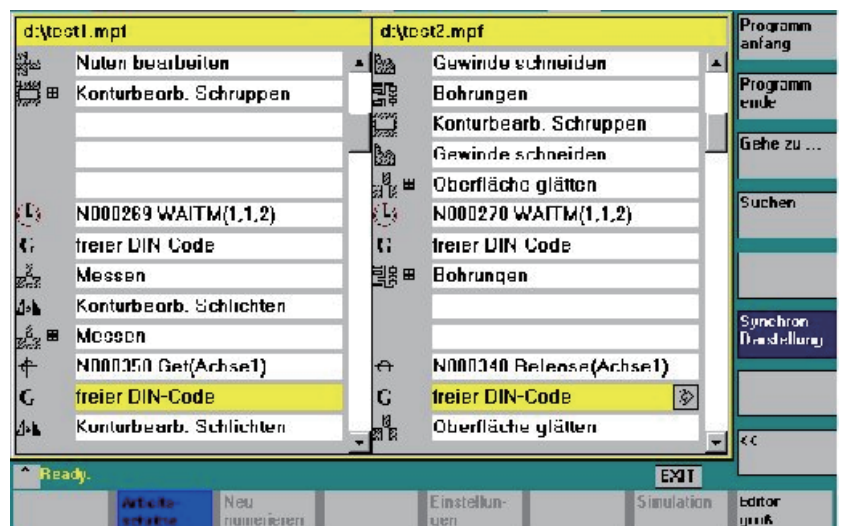


6.7.3 Synchronized display in the dual-step editor

Synchronized display

Find/Go to...

Here two programs are loaded in the step editor and displayed side by side. Both these programs are searched for synchronization points, and the points are displayed at the same level.



All synchronization marks that are to be synchronized in the step display must be stored in a SEDITOR.INI file.

In the standard file SEDITOR.INI (directory \hmi_adv) the entries for the standard wait markers are already specified in the section SyncMarks_Siemens.

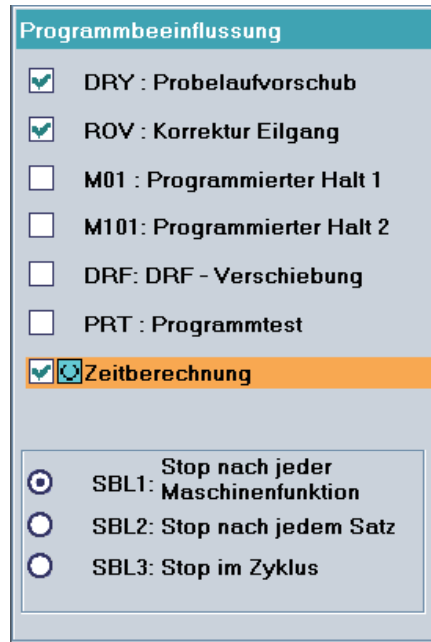
6.8 Multi-channel step display (option from SW 6.4)

The machine manufacturer or user can add entries in the SyncMarks_User section. It is also possible to add entries for cycles in the section Old_or_User_Cycles_User. You can specify as many entries as you want. See SEDITOR.INI or /IAM/ IM4: Installation and Start-up Guide, Section 5.3 Parameter Assignment with ini Files

6.8 Multi-channel step display (option from SW 6.4)

Purpose of application	Multi-channel step display is used for editing, navigating and time optimizing multi-channel workpiece programs. The function is available for multi-channel systems as an option: "Multi-channel sequencer programming".
Preconditions	<ul style="list-style-type: none"> • The individual programs for a workpiece are in step display • Channels were assigned via <ul style="list-style-type: none"> - JOB list or - START/INIT in a start program • The required coordination instruction was assigned to the individual programs
Activation	<p>Methods of activation:</p> <ul style="list-style-type: none"> - Workpiece is selected in workpiece directory WPD - JOB list is selected - <Workpiece name>.MPF <p>selected and Edit workpiece softkey is activated.</p> <p>If the above mentioned conditions are met, each of the methods or opening the start programs results in creation of a file within the control for:</p> <ul style="list-style-type: none"> - Multi-channel program display and execution, see purpose of application above - Basic for step by step Simulation, see Chapter 6 "Program simulation". - Time computation for time proportional step display <p>Multi-channel step display is activated.</p>
Activating the time computation	<p>Time computation for the steps is based on the time acquisition for individual instructions in the NC.</p> <p>The time computation is activated in the Machine operating area under program control.</p>

Edit
workpiece



Starting

The job list or start program must be selected and executed with NC Start. The required execution times for the individual instructions are added to the times per step and saved within the control.

Displaying time proportional steps for multiple channels

Edit
workpiece

The multi-channel program display is opened when you press the "Edit workpiece" softkey once the above listed preparations have been completed.

Evaluation and optimization

Reposition the steps or synchronization points (WAITM()) so that multi-channel execution leads to better parallelness and therefore shorter workpiece processing times.

Other information

After making changes to the program, you need to process the workpiece again to compute the new times.

Programs which have been modified are marked with a red X until the times have been updated.

If no time computation was performed, all steps in the program are displayed with equal time lengths.

The computed times are saved in a .DAT file for each workpiece. The file is compressed when the step editor is exited.

Multi-channel views

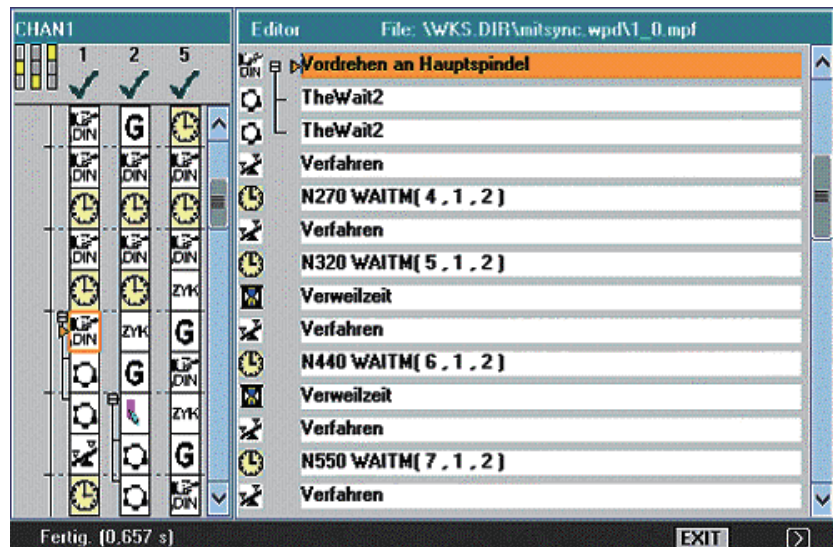
There are 3 views:

- Standard view, all steps are the same length
- Time proportional steps
- Synchronized display

Synchronized display is possible without having performed the time computation.

6.8 Multi-channel step display (option from SW 6.4)

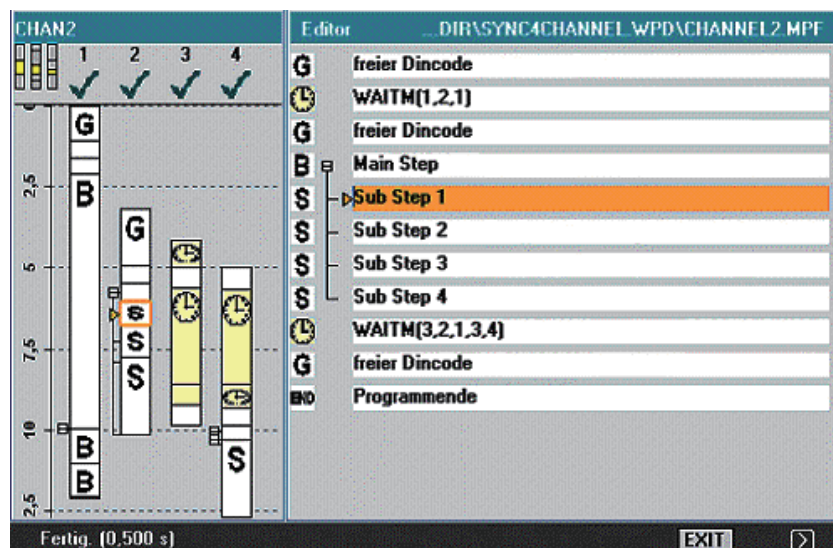
Standardized view



Features

Each step is displayed with the same length, regardless of its contents. Step identifiers are displayed for the program which currently has the focus. Status: "Program data was modified" is not displayed in the Program screen of channel 1.

Time proportional view



Features

Steps are displayed in proportion to the duration of their execution time. With wait markers, the step length is displayed proportional to the wait time.

Precondition

Time computation must have been performed by execution of the workpiece program. If the time computation was not performed, standardized view is displayed. Programs which were modified after the time computation are marked with "Data modified". Computing the time again sets all programs to "Data updated". See below.

Time computation

Preparatory phase: Message: Time computation is initialized. Please wait...

Initiation of execution: Message: Time computation is active.

Synchronized view

Synchron.
view

Features

Background color of the
synchronization icons

After completion:

Message: Recorded data is processed. Please wait....



The icons are displayed in a fixed size independently of the time required for the step. The grouped synchronization steps are displayed in parallel with the channels and filled with empty steps if required.

After switching to synchronized view, the data of all programs are initially "up-to-date". Making changes can lead to programs being marked with "Data modified". When you change back to standardized view and back to synchronized view all programs are re-synchronized and displayed as "up-to-date".

Green:

Correct synchronization step -> The synchronization syntax and parameter assignment specified in the configuration file SEDITOR.INI is recognized and all addressed partners are found.

Yellow:

Not all partners could be checked, as at least one channel must have been addressed in the synchronization syntax which was not contained in the *workpiece.DAT*.

Red:

Not all partners were found -> Indication for programmer to check the synchronization syntax or add the missing synchronization programming in the addressed channel.

In addition: Unsymmetrical channel number data at the synchronization partner(s).

6.8 Multi-channel step display (option from SW 6.4)

Cursor on synchronization icon

If you position the cursor on a synchronization icon additional states (**channel view**) are shown by means of the icon's background colors.
Background color

Green:

The channel is affected by this synchronization and synchronization partner found.

Yellow:

The channel is affected by this synchronization, but at least one of the specified channel numbers is not registered in the *workpiece.DAT* file.

Red:

In this channel the partner was not found or the channel for the synchronization icon which has the focus contains at least one channel number from *workpiece.DAT* in which the searched partner(s) was/were not found.

(Note: Programming must be modified either in the selected channel (too many/incorrect channel number(s) specified) or in the searched channel (missing synchronization)).

In addition: Channel affected by this synchronization but the specified channel number(s) do not match the selected synchronization icon.

Additional dialog box

The first synchronization that was not found (red) is output in a dialog box:

Synchronization string, channel, channel to be synchr.

Continue:

Position on block number or press

Posit. on
block no.

Cancel

Cancel softkey

Switching between work types

The following transitions between display types are available:

a) When first opened:

No time data available → standardized view

Time data available → time-related view

b) In standardized view:

only change to synchronized view possible.

c) In time-related view:

only change to synchronized view possible.

No change is possible to the standardized view as the steps and channels already have the correct relation to one another the time-related view.

d) In synchronized view:

Change to corresponding view if:

No time data available → standardized view

Time data available → time-related view

Synchron.
display

Description of layout
of views

Type of representation:

Standardized:



Time proportional:



Synchronized:



Updated state

Data is up-to-date:



Data is modified:



The time calculation must be performed again after any changes were made to the program. The icon columns are marked with the associated channel numbers. The channel which has the focus (highlighted) is displayed at the top left with the channel name. The step identifiers on the right belong to the program which has the focus. The selected step is marked as identifier text and icon.

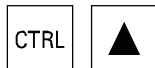
Display limit

A **maximum of 10 channels** can be displayed simultaneously.

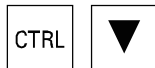
Navigation in multi-channel
displays



Return to previous/go to next **step**.



Back to previous step with the same **execution** (icon)



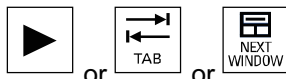
To next step with the same execution (icon)



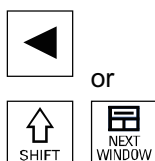
Return to the previous **synchronization mark**



Go to the next synchronization mark



A step is searched for in the channel to the right of the current channel - which is executed at approximately **the same time** as the one just viewed. If the current channel is the rightmost channel, then the search is performed in the leftmost channel.



A step is searched for in the channel to the left of the current channel - which is executed at approximately **the same time** as the one just viewed. If the current channel is the leftmost channel, then the search is performed in the rightmost channel.

6.8 Multi-channel step display (option from SW 6.4)

Settings

Settings

Other operations are identical to those for simple step display.

If step display is active (single-channel and multi-channel) you can use the softkey to select whether e.g. the times are to be specified in the step displays next to the step identifiers. The settings become active immediately.

Template

Similarly to the JOB list, the internal file *workpiece name*.DAT for coordinating the multi-channel step display and simulation can also be created via template. See Section Start-up operating area, HMI "System settings".

It is updated upon selection of a multi-channel program and softkey "Edit workpiece" and with the time computation. See above.

A *workpiece name*.DAT file is only created/entries entered in this file if the files were selected via *workpiece name*.WPD, *workpiece name*.JOB or *workpiece name*.MPF.

Multi-channel simulation

The multi-channel step display also supports multi-channel simulations. Here the NC language labels such as **WAIT2:** WAITM are inserted in the NC part program. The labels are removed again when you go back from simulation to multi-channel step display.

If you are in simulation or in the compensation editor and

- switch the control off or
 - remove write access from the opened files,
- the labels are retained.

6.9 Program simulation



Function

Simulation Drilling/milling and turning complete machining

Simulation selection

The graphical simulation is implemented as a self-contained process. The simulation can be selected directly from the program overview or from the ASCII editor after selection of a part program in the "Program" operating area.

Startup

You can perform simulation without any special start-up. The simulation can be started with default data in accordance with the set technology turning or milling. Using the "match data" function, selected data can be loaded from NC to the simulation environment and are then available for the simulated programs just like during program execution in the NC.

Providing the correct password is used, additional functions and optimizations (e.g. accelerate simulation run-up) can be performed. This is described in /IM4/ Start-up HMI Advanced /Section simulation data alignment. In addition, the default values listed in the simulation interface can be adapted to suit the machine manufacturer's needs.

Technology versions

- Drilling/milling
- Complete turning operations
- Workpiece-specific technology assignment through local "dpwp.ini" file

Superimposing principle

The simulation results of several part 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 part programs. **The direct simulation of several part programs (simultaneously) is not possible.**

The range of effectiveness (channel, spindle, tool, sequence) of the currently selected part program is visible in the status line on the lower edge of the simulation window.

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 cycles the program pointer is positioned at the line with the corresponding subroutine call. If changes are made in the editor, simulation is returned to the last active section (intermediate model if available).

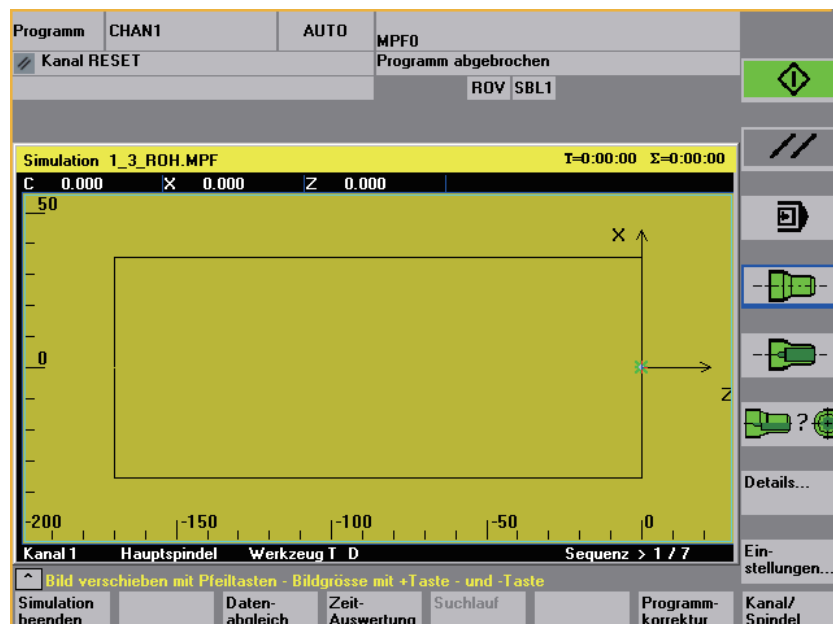
Other information

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. **WAIT: 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. The model used is based on a 3D Cartesian model.

6.9.1 Simulation operation

Main menu

After a program or workpiece is selected, the Simulation softkey becomes available.



Turning technology main screen

When you select a simulation via a **workpiece**, first the machining list from the Channel/spindle menu (see below) is displayed. Press OK to exit the menus. The main menu is now displayed.

Basic horizontal menu

End
simulation

Terminate simulation. Return to the program overview or ASCII editor. You need the correct password for this.

Loaded
data or Match
data

With the correct password you can match the simulation data with the corresponding "NC active data" (setup data, tool data, machine data, cycles).

- See menu "Match data"

Time
computation

Tabular evaluation of the execution times for the current simulation session (see Section "Settings for secondary times")

Block

The function "Search run" can be used to start the simulation selectively for specific sections.

Correct
program

Activate the ASCII editor from the current simulation interruption status (cursor position synchronized with graphic, user interface)

- Return to simulation with "Close editor"

Channel/
spindle

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)



Simulation RESET
The previous execution result is discarded for the program at hand and a polymarker is displayed.



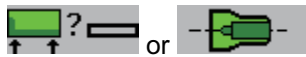
Simulations single block ON/OFF - SINGLE BLOCK
(status display in header SBL1, SBL2 or SBL3)

Milling  or Turning 

Technology-specific workpiece views

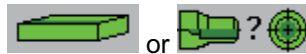
Milling: Top view

Turning: External view, frontal



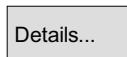
Milling: Default top view
and front view
(free selection under "Details...")

Turning: Full section, frontal

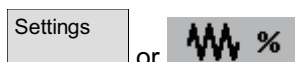


Milling: 3D view
or wire model

Turning: Default half-section, frontal
Front and wire model
(free selection under
"Details...")



Select status-dependent detail menu (active view, alarm status)



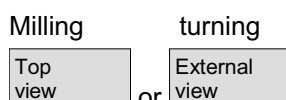
Select user and vendor-specific setting menus
in RESET or STOP state
or
in RUN state display current simulation override



Note

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. **With the technology "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



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 the different views in window activated by means of



Front view	or	Half section	ditto
Side view	or	Full section	ditto
Wire model	or	Wire model	ditto
		Face end	ditto
		Periph. surface	ditto

Menu "Details...." standard vertical

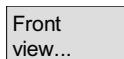
- "Top view" or "top view and front view" for milling
- "Wire-frame model" (3D without tool data) for milling
- All views for turning

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).

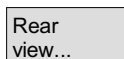


With 2 window views only: Select active window
Possible in the same manner with "TAB"- or "NEXT WINDOW (from V06.02.13).



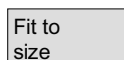
Conditionally with 2 window views:

"from front..." depending on type of view active (horizontal selection) same meaning as "from above..." or "from left..."




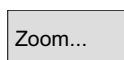
Conditionally with 2 window views:

"from behind..." depending on type of view active (horizontal selection) same meaning as "from below..." or "from right..."



Automatic screen resizing

relative to the window activated with . Also possible with "ENTER" or "INPUT" key.



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

"3D view" (3D with tool data) for milling, not effective for turning simulation



Standard 3D view, orientation top/front



3D view, orientation top/left, rotated clockwise by 90°



3D view, orientation top/right, rotated clockwise by 90°



3D view, orientation bottom/front, inclined towards top

Fit to
sizeAutomatic screen size adaptation
Also possible with "ENTER" or "INPUT" key.

Zoom...

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

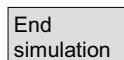
Reset POWER ON alarms of simulation. The simulation is terminated and loaded again.



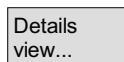
Reset RESET alarms of simulation. The simulation interpreter is reset. The simulation can be started again.



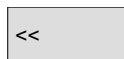
Reset CANCEL alarms of simulation. The simulation can be continued.



The simulation process is terminated. A load operation is necessary before the simulation is selected again.



Open the menu bar corresponding to the active workpiece view "Details..." (standard or 3D) from the alarm state. The simulation alarms remain active.



Return to main simulation menu

**Additional note**

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 "Match data"
verticalLoaded
data

Correct password and NC components are prerequisite for using Match data to load data selected in the NC to the simulation environment. The data is then available for the simulated programs just like during program execution in the NC.

(Matched data see data management \DP.DIR\SIM.DIR)

The loaded files are displayed.

User programs, user cycles, standard cycles, manufacturer cycles and base data.

Compare
setup data

The selected data is loaded into the simulation environment from the NC.

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.

Compare
tools

The tool data for all channels is loaded to the simulation environment and copied to the file TO_INI.INI.

Compare
mach. data

The machine data and active definition files are loaded to the simulation environment and copied to the file INITIAL.INI.

Compare
cycles

The cycles which have already been loaded into the simulation are replaced by cycles with more recent time stamps.

<<

Return to main simulation menu.



Additional note

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 part 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.
4. It is recommended to set MD11210=0 "Adjust all machine data".

Menu "Channel/spindle" vertical

Channel/
spindle

Can be accessed from RESET or STOP status only.

The screen shows the current **execution list** for the selected part programs.

Programm	CHAN1	AUTO	MPF0	
Kanal RESET		Programm abgebrochen		Einstellung Kanäle
		ROV	SBL1	
Kanal/Spindel: INDEX_2_G400_A2032620				
Sequenz	Programm-Name	Start-Label	Stopp-Label	Kanal Spindel aus
1	1_3_ROH.MPF			1 HS <input type="checkbox"/>
2	3_0.MPF		WM15:	3 HS <input type="checkbox"/>
3	1_0.MPF		WM15:	1 HS <input type="checkbox"/>
4	3_0.MPF	WM15:	WM20:	3 HS <input type="checkbox"/>
5	1_0.MPF	WM15:	WM20:	1 HS <input type="checkbox"/>
6	3_0.MPF	WM20:		3 HS <input type="checkbox"/>
7	1_0.MPF	WM20:		1 HS <input type="checkbox"/>
Hinweis : Mit 'Ok' wird das erste Programm angewählt und ein Reset ausgelöst.				
Mit Pfeiltaste Element auswählen . Kanäle : (+/-) 1, 2, 3, 6				
Programm- auswahl				

Kopieren
Einfügen
Löschen
Fertigteil
Abbruch
OK

Sequence

Simulation sequence of programs

Program name

List of programs

Start label

Path label from which a program is simulated. If this is not specified, simulation takes place from program start.

Stop label

Path label up to which a program is simulated. If this is not specified, simulation takes place to program end.
(See also "Multi-channel, section by section program simulation".)

Channel

Processing channel in the SIMNCK interpreter

Spindle

Turning: MS main spindle, CS counterspindle, var variable controlled via NC vocabulary word.
(Milling: currently unused)

Skip

Skip. The designated section/program is not simulated in the current session.

Channel
settings

Leads to the manufacturer-specific channel settings (subject to correct password).

Milling

Constant machine arrangement

Turning

Machine arrangement in front or behind turning center.

Spindle
settings

Leads to the manufacturer-specific spindle settings (subject to correct password).
currently unused

Milling**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 to clipboard.

Paste

Insert the machining list line that was copied/deleted from the clipboard at the position of the selected line (the selected line is moved down one row).

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



Channel/
spindle

Additional note

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 so that it 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 part programs in succession is to be viewed at a single blank). In these cases please pay particular attention to the notes in the lower edge of the screen. They contain e.g. lines for which an intermediate model was already created, special status display, etc. 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**.

6.9.2 Simulation settings

"Settings..." menu

Only accessible from the RESET or STOP state of the simulation (e.g. single block mode).

Other settings can only be displayed/hidden if you enter a valid password via the Options softkey.

Load
standard

Load manufacturer-specific default settings

(See data management: \DP.DIR\SIM.DIR\SIMINI_M.COM for milling or
SIMINI_T.COM for turning and differential files in the \USER directory.

Options
on/off

or

Options
on/off

Display/hide optional setting parameters in the current window,
(Initial setting is options off)

Change
default

The default values can be modified according to the manufacturer's
needs if a valid password is entered.

The changes to the manufacturer-specific default set values in files
"SIMINI_M.COM" and "SIMINI_T.COM" are used as differential
parameters in the files "DPMWP.INI" and "DPTWP.INI" in the \USER
directory and taken into account for all **new workpieces**.

Downtime
settings

With the correct password, you can set the desired computation mode
for secondary times and selectively define estimated secondary times
for the NC functions T, S, M and H.

- Settable parameters: See Section "Settings secondary times"

Display and
colors...

The specified display and color properties for the simulation can be modified according to the manufacturer's needs if a valid password is entered.

- 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.

Basic "Settings..."

Blank

- **Milling:** without model (display area), cuboid (blank dimension cuboid), cylinder (blank dimension cylinder).
- **Turning:** without model (display area), cylinder (blank dimension cylinder) and active processing diameter for peripheral surface.

Active view

- **Milling:** X-Y, Z-X, Y-Z (for "Cuboid" and "Without model" only)
- **Turning:** Z-X always defined, cannot be changed

Blank dimension cuboid

- Minimum and maximum dimensions for each axis

Program control

- SKP: skip blocks (SW 6.4 and higher 10 skip levels are supported)
- M01: Programmed Stop 1 or
- M101: Programmed Stop 2 (SW 6.3 and higher) need to be taken into account
- Single block mode:
 - SBL1: Stop after each machine function
 - SBL2: Stop after each block
 - SBL3: Stop in the cycle (V06.02.12 and higher)
- Display all blocks or only traversing blocks

Tool data (source) NC active data

The tool offsets are used in accordance with the active NC (tool data adjustment \TO_INI.INI) and the tool path is simulated with graphic tools. With technology turning the cutter radius is not modeled. The tool radius compensation is evaluated. Tools that are not interpreted are displayed as polymarkers.

With tools

MMC data, local TOA data, global SPF file, tool adjustment data from data management \DP.DIR\SIM.DIR\TO_INI.INI

Default tools

Evaluation only with milling/drilling if no TO_INI.INI (tool data adjustment) is present and tool management option not active. The tool diameter can be parameterized (providing access is enabled via a valid password) if ON is activated under the Options settings. The tool radius compensation is evaluated.

Without tool data

The tool path is simulated with a polymarker. No evaluation of tool radius compensation i.e. broken-line graphics with compensation value D0.

Additional note

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..."

Options
on/off

Other settings can only be displayed/hidden if you enter a valid password via the Options softkey.

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).
 - IPO mm or inch: Approximation precision of simulation interpolator in mm or inches depends on actual measuring system.
 - F override %. Setting the simulation feed
- % setting options: %1 to %500 (from V06.12.13 < 10 %)
- 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

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 (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 display (zoom, etc.).
 - * In position "off", the resulting tool paths are visualized once and not stored temporarily in the model. When the display is subsequently manipulated (e.g. Zoom +), 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 note

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.

Recommended settings

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"
 - .. In training operation
 - 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" (no memory used for graphic!)

- Or "without tool data", if representation of the tool center-point path (broken-line graphics) is sufficient.

If the blank model is active ("cuboid", "cylinder"), reduced graphic memory capacity might result, which, depending on the complexity of the part and model resolution set, could exhaust the graphic memory.

6.9.3 Setting downtimes

Time recording mode

Settings

Downtime
settings

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".

Per program

Central time recording includes the elements listed in "Inclusion on" in downtime consideration. The tabular "Time computation" is processed once per program in each case with M30, for example.

Per section with labels

Effect as for "per program", only that processing of the tabular "time computation" is also performed when freely definable program labels occur or in combination with the path labels displayed in the simulation graphic (see ..\ Display and colors...\ Manage path labels).

Inclusion

Enabling and setting of fixed downtimes in seconds

Tool change

ON/OFF

- Definition of a fixed downtime for tool changes

Spindles

ON/OFF

- Definition of a fixed downtime for main spindle and secondary spindle instructions

M functions

ON/OFF

- Definition of a fixed downtime for M commands

H functions

ON/OFF

- Definition of a fixed downtime for H commands

Settings

Load
standard

Change
default

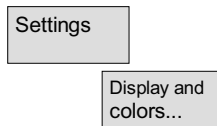
Additional note

The default values can be modified according to the manufacturer's needs if a valid password is entered. The new values are then globally valid for all new workpieces.

6.9.4 Display and colors

General attributes

Setting the general properties of the simulation graphics:



Rapid traverse dashed line Alternatively, rapid traverse as a full line, as used for feedrate

Scale at edge of window Alternatively, as scale on the coordinate axes

Path labels

Under "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.

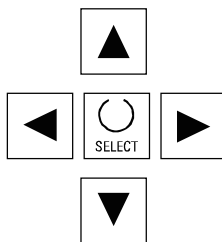
Program labels can also be used to mark program sections which can then be included by the central time recording facility (see ..\

Downtime settings \ Time calculation mode \ In sections with labels).

Path labels are specified in "Channel/spindle" as delimiter for section by section simulation in the same or different channels.

You can save section models and use a **search run** to **start again at that point** without needing to repeat the preceding sections.

Use of color palettes



- Select a color with the cursor keys and press the Select key. The color of the focus frame changes.

- Use the cursor keys to select the object and press the Select key again. The object displays the selected color.

General colors

In addition to the standard VGA colors, the color elements black and transparent to hide graphic elements are available in the color palette. Color selection option for **background**, **blank**, **axis intersection**, **tool holder** and **tool edge**.

6.9 Program simulation

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.

Planes for color classification

In the case of milling/drilling, the available colors are assigned to the defined **cutting depth range**. The default value is the thickness of the blank.


 Settings


 Load
standard


 Change
default

Additional note

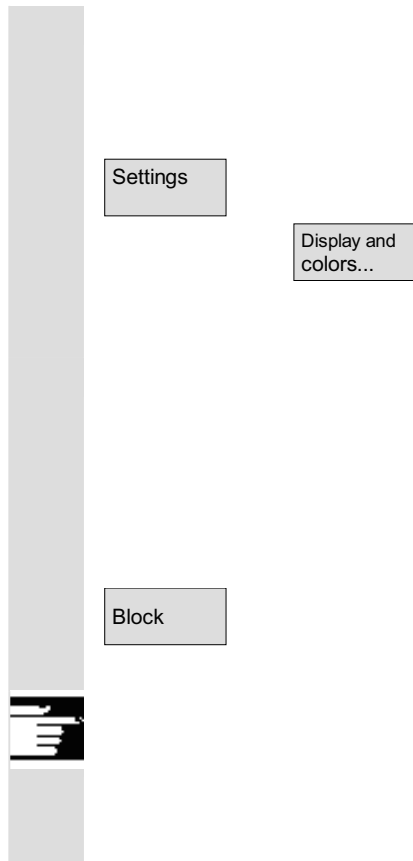
The default values can be modified according to the manufacturer's needs if a valid password is entered. The new values are then globally valid for all new workpieces.

6.9.5 Section by section simulation

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 part 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 always defined by path markers (program labels).



Requirements:

- 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 Note

- You can also load the machining list in menu "Channel/spindle" directly from the current workpiece via the softkey "Program selection" or from a JOB list.

6.9.6 Multi-channel, section by section program simulation

Introduction

The above mentioned operator options allow a multi-channel, section by section program simulation with the following characteristics: There are specific NC instructions for program coordination for synchronizing program execution as required for multi-channel manufacturing of workpieces: (*INIT(..)*, *START(..)*, *WAITM(..)* etc.). In principle, they can be used to start independent program sections in parallel channels almost *simultaneously* and synchronize them via e.g. *WAITM(Marker No., ...)* once the associated machining operations have been executed and before starting the next sections.

Simultaneous aspects can only be sequentially emulated with simulation. **NC program labels serve to mark program sections.** Emulating the multi-channel program coordination in the simulation assumes that NC instructions for program coordination were marked with NC labels (*manually or automatically during program generation*). Simultaneous operational sequences which are independent of one another are executed one after the other in the respective channels (*channel change per section required*), before sequences are created for the next sections in the same manner.

Restrictions

The result in the simulation is a *similar* sequence of machining steps to that on the machine.

- Only **one** out of 10 possible channel instances can be assigned to the single-channel graphic module (simulation display) **at any one time** (*no simultaneous mode for channels!*). Commands for multi-channel program coordination (*INIT(..), START(..), WAITM(..) etc.*) are not executed in the program interpreter for the simulation.
- Partial processing of the "channels" on the same stationary blank are **superimposed**.
- **Path labels** Labels are universal markings/section structure aids in NC programs which can trigger specific actions during the simulation run:
 - Path labels can be optionally displayed in the graphic
 - The intermediate time can be computed for sections delimited by path labels.
 - The graphic state can be saved as an intermediate step for sections that are delimited by path labels.
 Path labels have a defined label syntax:
 Identifiers:
 For simulation they should be by themselves in one NC block in so far as possible.

- The channel changes section by section with defined path labels (NC program labels).

Section delimitation using path labels

If the "Stop label" column is clearly defined, the "Start label" results implicitly:

- An empty field "Start label" has the same meaning as a start from the beginning of the program
- An empty field "Stop label" has the same meaning as a stop at the end of the program
- With the same "program name", the "start label" of a section is equal to the "stop label" of the preceding section
- If start and stop labels are not specified, the entire program is executed.

JOB list

If the machining list does not yet exist, it is created at simulation selection and populated with default values from the job list or from the corresponding list from the STEP Editor in the current workpiece directory.

Simulation sequence

Upon confirmation of the machining list with OK, the relevant [JOB] section is generated in the workpiece-specific DPWP.ini file.

The following entries are managed:

- Program name.type
- Channel number
- Spindle unit: Main spindle, counterspindle, variable
- Start label (path label)
- Stop label (path label)
- Length of the machining list (number of sections)

The machining list can be updated at any time via the "Channel/spindle" menu.

The simulation sequence is controlled by the contents of the machining list menu "Channel/spindle" and supports simulation of multi-channel program coordination:

- Selecting the list validates the specified operational sequence.
- In each sequence step the designated program can be fully executed - as before - (*i.e. from program start to M2/M30 or M17*) in the specified channel by leaving the fields "Start label" and "Stop label" empty
- With each sequence step you can change to a different channel at the section of another program (*section by section channel change*) up to the specified "Stop label".
- If a label which was not previously defined as a "Stop label" for the sequence is specified in the "Start label" field, an implicit search run will be performed (*without output of graphics*) up to the "Start label" in the selected program before the simulation run is started.
- If the label assignments are inconsistent, the simulation is reset to the last valid state and a message is issued to this effect, e.g. "Label xxxxxx not found. Adapt label definition in machining list or adapt part program!".
- Each sequence step should be started separately and can also be reset separately.
- Section by section change of the spindle unit within the sequence is permissible and controlled by means of vocabulary words from the program.
- The section by section channel change during program run is only supported in conjunction with **specification of the corresponding program labels** (path labels) which are either programmed manually or generated internally via integrated functions.

Simulation result

By superimposing the partial executions from the machining list **on a blank** the section by section simulation mode only changes for the sequence of machining sections in the graphic. There is no change in the overall result.

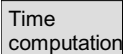

 Block

The list of machining steps (for which an intermediate model is available in the graphic module in the current simulation state) allows direct selection of a section in the simulation. See also "Display and colors"/Manage path labels.

The corresponding section is selected and confirmed with OK.

The simulation will start at the beginning of this section and is based on the state which the preceding section was in the last time its execution was completed.

Time computation


 Time computation

The table shows the analysis of **time computation from the simulation** and encompasses the specified sequence of execution up to the current state of the simulated machining sections.

The computation table is in cumulated form and has taken the **defined downtime values** into account (*for T, S, M and H functions if present*); data is shown for the respective section limits.

The section limits are marked with the associated ASCII label derived from the "Stop label" (*if present*) or with M30 at the end of the program. Times are computed at each section end.

6.9.7 Simulation with orientable toolholder



Function

You can process part 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.
- Inactive files are grayed out.

6.9.8 Quick display in the simulation for mold making

Target

The function offers quick display of machining paths, e.g. like those used by CAD systems. Generally these part programs are very large.

Any existing work offsets, Go, G2, G3 are not taken into account, only those **axis paths resulting from G1** are displayed.

Characteristics of quick display

- Simulation for mold making is a standard function
- Switching between 2D and 3D is supported
- Rotation of workpiece in 3D view
- Representation in the various different planes
- Size scaling, zoom for the displayed workpiece
- Search via row number/string
- Search via machining block which traverses marked positions
- Progress display
- Execution of part program blocks
- Distance measurement
- Can be interrupted by other operating areas/can be aborted
- The programs to be simulated can be run from external drives
- The NC interpreter does not take part in the simulation.

Activation

Operation for mold making simulation is available in the Program operating area if display MD 9480: MA_SIMULATION_MODE is set to a value between 0 and 2.

- | | |
|----|---|
| -1 | Standard simulation |
| 0 | Selection between standard simulation/mold making G1 blocks via operation |
| 1 | Mold making only, G1 blocks |
| 2 | Automatic mode selection via program size |
- The limit value for the program size is specified in the display machine data 9481: MA_STAND_LIMIT.

Size scaling

The following scaling settings are available: **Zoom in**, **Zoom out**, **Fit to size** and **Fit in window**.

The automatic scaling function "Fit to size" uses the part program to take the largest expansion of the workpiece in the individual axes into account.

Search

In the submenu **Details**, the search function displays the part program block whose machining path traverses the **position** (which was previously marked using the cursor keys or softkeys) in the workpiece representation.

First activate **Block search** then use the cursor keys or softkeys to move the cross-hair to the desired location. Press the input key to search for the required block and display it. The cross-hair must be positioned in the area of the displayed workpiece section.

In the submenu **Edit** you can choose whether to search for a block designated by a **block number** or a block which contains a specified **string**.

The search function will return the blocks that match the criteria and display them in the 2-line program section above the graphic display and highlight them.

Progress display

The message line shows the percentage value of the entire part program which is displayed in the graphic.

Loading the workpiece graphic can be cancelled at any time by pressing **End**. Changing to a different operating area interrupts loading the graphic. When you return to the Program operating area, the loading process is continued.

Rotation

In 3D representation the displayed workpiece can be rotated around any of the axes. The rotation instructions become active after you press **Accept** to confirm.

Distance measurement

In the graphic display you can use the softkey **Mark point A** and **Mark point B** to enter two markers at the positions you have set via the cursor keys. The direct path (space diagonal) between the points is output in the message line.

6.9.9 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 part 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.

6.10 Manage programs

6.10.1 Overview

Program management

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 part programs for immediate execution and
- Hard disk drive

Programs can be exchanged with the program memory and hard disk. Part 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.

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:

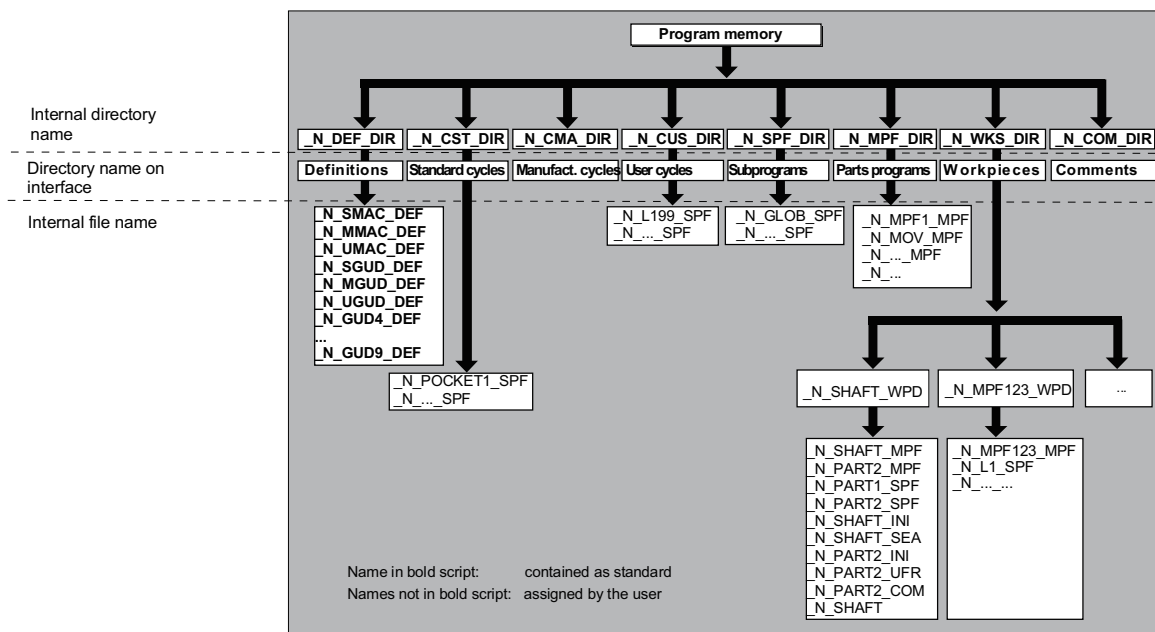
Name of directories:

- Subroutines
- Part programs
- Workpieces
- Definitions
- Comments
- Standard cycles
- Manufacturer cycles
- User cycles

Operating area:

Programs and Services
Programs and Services
Programs and Services
Services
Services
Programs and Services
Programs and Services
Programs and Services

The following diagram shows an example of directory contents:



6.10.2 NC data types and directories

File types

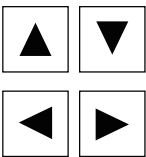
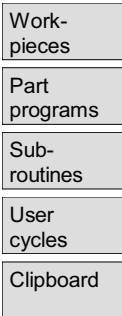
File types can be identified by the file name extension (e.g. .MPF).

name .MPF	Main program
name .SPF	Subroutine
name .TEA	Machine data
name .SEA	Setting data
name .TOA	Tool offsets
name .UFR	Work 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 via "Manage data".



6.10.3 New workpiece/part program



Select workpiece/part program

The following subsection describes how you can select workpieces and part programs in a directory. A selected file can then be called and edited in the text editor.

Operating sequence

Select workpiece/part program:

- Workpieces
- Part programs
- Subroutines
- 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 part 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 part program.

Open a workpiece:

The workpiece directory is opened and the programs it contains displayed on the screen.

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.



Workpieces

New

Operating sequence

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.
Specify the created type in the data type field: WPF

If there are any templates for the specified data type in the Templates directory they are offered for selection. Press OK to confirm your selection.

A new directory is set up in the workpiece overview.

You are asked straight away for the name of the first part program and the Editor is opened.

When creating a workpiece with **New**, all **templates** corresponding to your selection are only copied from the Templates\Siemens ... Manufacturer ... User directory to Startup\Settings\Templates if "**No template**" is selected in the Templates selection field.

If a template is proposed for the workpiece, then all elements assigned to it such as job list, part program, subroutines are transferred to the new workpiece in compliance with the template and its language-specific elements.

For more information, see Subsection 6.1.5 Templates



New



Set up programs/data in a workpiece directory

This section explains how you can set up a new file for a part program or workpiece.

Operating sequence

The current overview of the workpiece directories stored on the NC appears.

Position the cursor on the required workpiece and open it.

An overview will be displayed listing the data and programs that have already been created in the workpiece directory. If no data is available, the program overview is blank.

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
.WPD	Workpiece
.MPF	Part program (main program file)
.SPF	Subroutine (subprogram file)
.JOB	Job list
.GUD	Channel user data
.IKA	Compensation data
.INI	Initialization program
.UFR	Work offset/frame
.TOP	Tool plan
.TOA	Tool offsets (tool offset active)
.TMA	Magazine data
.TEA	Machine data (testing data active)
.TCM	Tool plan reformatted (for SINTDI)
.STP	MCSE data
.SEA	Setting data active
.RPA	Arithmetic parameters
.PRO	Protection zones
.DAT	Time calculation

.COM	Comment
.CEC	Sag/angularity
.041	AutoTurn program

Part programs or Sub-routines

New



Number of workpieces

Creating part programs in the part program/subroutine directory:

You can set up main programs and subprograms by opening directories "Part programs" and "Subroutines".

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.

6.10.4 Saving setup data



Workpieces



Save setup data

Function

With the softkey "Save 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.

Other information

"Save setup data" can be disabled by the manufacturer using the protection levels.

Operating sequence

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 parameters (RPA)
- Work offset (UFR)


 Save

- Setting data (SEA)
- ...

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 in the relevant workpiece directory.

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.

 Load
standard

With the softkey "Load standard" you can load the default settings for input window "Save workpiece data".

Note:

Default settings are always supplied by SIEMENS.

 Set
standard

Press this softkey if you want to define your own defaults in the input window.

6.10.5 Select program for execution



Function

Workpieces and part programs must be selected for machining/execution before you press the NC Start key.



Operating sequence

Select a program:

Use the cursor keys to select a program in the

program overview, e.g. part programs, and

then press softkey "Selection".

The program name is displayed in the "Program name" window at the top right.

You can start the part program by pressing "NC Start".

 Part
programs


Selection



Select a workpiece:

A workpiece directory can be selected for machining in the currently active channel.

Work-
pieces

Selection



Cycle Start

Selection

Find path for program call

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 part program you have selected by pressing the "NC Start" key.

- If an INI file of the same name exists (e.g. `SHAFT.INI`), it will be executed once at first part program start after select the part program. Machine data 11280 `$MN_WPD_INI_MODE` controls which additional INI files are executed.

When you select workpiece directory `SHAFT.WPD` you implicitly select program `SHAFT`.

If a `.JOB` file with the same name is stored in the workpiece directory, 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 part 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 data type** ("identifier" or "extension"), e.g. `SHAFT1.MPF`,

the system searches through the directories in the following sequence:

- | | |
|---|---|
| 1. Current directory / <code>name.type</code> | Workpiece/standard directory <code>MPF.DIR</code> |
| 2. <code>/SPF.DIR / name.type</code> | Global subroutines |
| 3. <code>/CUS.DIR / name.type</code> | User cycles |
| 4. <code>/CMA.DIR / name.type</code> | Manufacturer cycles |
| 5. <code>/CST.DIR / name.type</code> | Standard cycles |



/PGA/, Programming Guide, Advanced

/IA/ Installation & Start-Up Guide HMI/MMC, IM4

6.10.6 Load/unload program



Download
HD->NC



Alter
enable

Upload
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.

Operating sequence

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 uploaded from the NC memory to the hard disk.

The selected program is deleted in the NC memory.

Other information

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!).

6.10.7 Load/unload program



Function

You can reorganize programs and files in the "program overview" using the "Manage programs" function:

New ...	Select new workpiece/part program
Copy/insert	Copy directories and files
Delete	Delete workpieces and files
Rename	Rename file and data type
Modify enable	Enable each workpiece/part program in the program overview or not.

Operating sequence

In the "Program overview" dialog box press the horizontal softkey "Manage programs...". Active functions are indicated by softkeys in which the text appears on a black background. Inactive softkeys are grayed out.

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.

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.

Confirm your input with the "Input" key.

Manage
programs...

New ...




 Abort

OK

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.

6.10.8 Copy/paste



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
Intermediate memory		or
Diskette (is specified as network drive)		or
Network drive (Network1 to Network4 with 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 too.

All files with the workpiece name are automatic renamed to the new workpiece name.

Files from the Workpieces directory can only be overwritten in another directory.

All workpiece files of type .WPD can be copied from one drive to any drive.

Specifying the file name and file type

Copying network drives (SW 6.3 and higher):

All data types known to the HMI are set according to their extension and the data management scheme.

A dialog box is opened for files with an unknown extension.

You can enter or change the name and file type - up to 3 characters are permissible.

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



Operating sequence

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.

Paste

OK



Press the softkey "Paste", 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 part program and in the global subprogram directory.



Query dialogs "Save as"

All without
query

Skip file

Abort

OK

Other information

- 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!

6.10.9 Delete



Function

This section explains how you can delete workpieces or files.

Operating sequence

The softkey "Manage programs" must be pressed.

Position the cursor on the workpiece or the file you want to delete.

Delete several files:

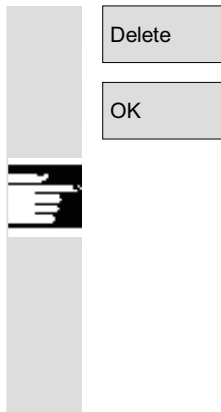
If you wish to select several files, position the cursor on the first file, press the "Select" key and then position the cursor on the last file. The files you have selected are highlighted.



Manage
programs



SELECT



The prompt "Do you really want to delete the file?" appears.

Confirm your input.

Other information

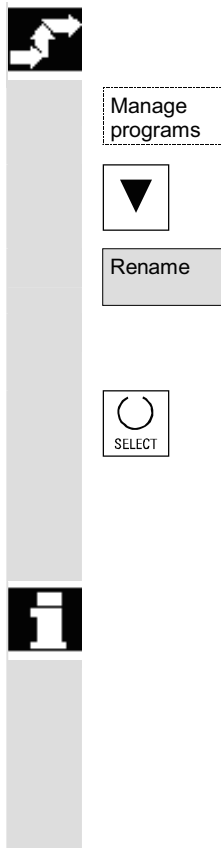
- You can only delete programs that are not currently running.
- If you want to delete a workpiece directory, make sure that none of the programs it contains is currently selected.
- If a workpiece directory is deleted, all the files that it contains are deleted at the same time.

6.10.10 Rename



Function

As regards files, you can alter their name as well as the associated file type.



Operating sequence

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 part 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` is renamed `B.WPD`:

All files with the name `A.XXX` are renamed to `B.XXX`, i.e. the extension is retained.

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 `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.

6.10.11 Enabling



Change
enable



Function

The program overview indicates whether a workpiece or part program is enabled.

This means that 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.

Operating sequence

To set the enable for a program or abort it, position the cursor on the desired workpiece or part program in the program overview.

Select softkey "Change enable".

A cross indicating "Enable issued" appears behind the workpiece or part program.

(X) Enable issued (program can be executed)

() No enable issued (program must not be executed)

Other information

- The system checks whether a program may be executed when the program is called (after selection via operator input or from part program). If an enable is required it must have been set previously. (See also Chapter 6 "Changing properties of file/directory/archive")

6.10.12 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.

**Operating sequence**

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.



Log



Yes



All



No



Stop

6.11 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



Operating sequence

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.



Other information

- 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	7-360
7.2	Directory structure	7-360
7.2.1	NC active data	7-360
7.2.2	Hard disk	7-361
7.2.3	Directories	7-363
7.2.4	Data selection	7-364
7.3	Formats for saving and importing data	7-367
7.3.1	Punched tape format	7-368
7.3.2	PC format	7-371
7.4	RS-232 interface parameters	7-372
7.4.1	Interface parameters	7-375
7.5	Operation	7-376
7.5.1	Services main screen	7-376
7.5.2	Setting the RS-232 interface	7-379
7.5.3	Reading in data	7-383
7.5.4	Reading out data	7-384
7.5.5	Log	7-386
7.6	Manage data	7-388
7.6.1	Standardized program and data management (from SW 6.3)	7-388
7.6.2	Creating a new file/directory	7-389
7.6.3	Loading/unloading	7-389
7.6.4	Copy/paste	7-390
7.6.5	Deleting	7-392
7.6.6	Changing the properties of a file/drive/archive	7-393
7.6.7	Defining and activating user data (GUD)	7-395
7.7	Start-up functions	7-397
7.7.1	Series start-up	7-397
7.7.2	Restoring the original state via NC card	7-399
7.7.3	Upgrade	7-400

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

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
- Downloaded 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.

Copy to diskette

When you copy to diskette the full file names are stored on the diskette.



Other information

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.

Query dialogs "Save as"

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

Limit of data management (DM):

A maximum of 100,000 files can be stored in the following data management directories:

- Workpieces

- Part 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:

- Users
- Service

Data selection for display: users
User cycles
Display machine data
Archives
Machining sequence
Data management
Definitions
Diagnostics
Interactive programming
Manufacturer cycles
MSD data
Start-up
Comments
MBDDE alarm texts
NC active data
NC data backup
OEM data
Standard cycles
System
Part programs
Templates
Subroutines
RS-232 (V.24) 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 parameters
- Start-up data
 - NCK data
 - PLC data
- Compensation data
 - Leadscrew pitch/encoder error
 - Quadrant errors
 - Sag/angularity
- Display machine data
- Workpieces
- Part programs
- Subroutines
- User cycles
- Standard cycles
- Comment data
- Definitions
- Feed drives
- AC Main Spindle Drives
- OEM data
- System data (NC)
- Log book
- Communication error log

If your control includes additional directories, these can be found in the file tree.



Data selection



Operating sequence

Services

The "Services" operating area is selected.

Data
selection

Press softkey "Data selection".

The "Data selection for display" window is opened.

The vertical softkey bar changes.

PAGE
DOWN

PAGE
UP

You can page through the window using the "Page" keys.



You can toggle between the settings

Users

- Users

Service

- Service

Default
values

- Default values

The softkey "Default 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.



OK

Mark the directory of your choice and press softkey "OK".

Abort

A reset command by means of softkey "Abort" for the selected access level resets the data selection.



Other information

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_WKS_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` (subroutines), files with extension `.INI` in the working memory and all other files in `MPF.DIR` (part programs).

Example of file with path name:

```
%_N_SHAFT_MPF
; $PATH=/_N_WKS_DIR/_N_SHAFT_WPD
N10 G0 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").

/IAD/, Installation and Start-Up Guide

Formats

Files can be stored in two different formats in archive files:

- Punched tape/ASCII format
- 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 is 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 name 1>
;$PATH=<Path name 1>                    ;can be included
1st block    LF      ;contents of file 1
2nd block    LF
...          LF
last block   LF

%<File name 2>
;$PATH=<Path name 2>                    ;can be included
1st block    LF      ;contents of file 2
...          LF
last block   LF

...                                     ;contents of file n
last block   LF

<Trailer>                                ;can be included
1.
```


<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.

LF
CR LF

Character for end of block/new line; ANSI value 10 (0x0A)
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 contain the characters 0...9, A...Z, a...z or _ and must not exceed 24 characters. 2. File names must have a 3-character extension (_xxx). 3. Data in punch tape format can be generated externally or processed with an editor. The file name of a file stored internally in the NC memory starts with "_N_". A file in punch tape format begins with %<name>, "%" must appear in the first column of the first line.
Examples:	<pre>%_N_SHAFT123_MPF = Part program SHAFT123 or %Flange3_MPF = Part 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 name can 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 punch tape format begins with ;\$PATH=<path name> in the first column of the program. Path names in punched tape format start with _N_ and end in _DIR (any directory) or _WPD (workpiece directory).
Example:	<pre>;\$PATH=/_N_WKS_DIR/_N_SPIGOT_WPD Workpiece directory SPIGOT in Workpiece directory</pre>
	<p>The data listed after the file name/path name belong to the file with the name specified after "%" in the directory specified after ";\$PATH=".</p>
<Trailer>	<p>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.</p>

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 **only** be saved 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-C:

- 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 XON as soon as it can receive data again (= default).

RTS/CTS (handshake)

The signal RTS (Request To Send) controls the transmission 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.

Transfer

From HMI software version 6.2, 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 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 with end of transmission

- ☒ Active: Text mode: The end of transmission character is active.
- ☐ Inactive: Binary mode: The end of transmission character is not evaluated.
Default value for the end of transmission character is hexadecimal 03 (ETX).

With leader and trailer

- ☒ Active: Leader ignored during input during output, 120x0(Hex) output (feed before and after data).
- ☐ Inactive: Both lead and trailer are read in.
No leader 0(hex) on output.
Read-in is automatically recognized.

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).



Saving RS-232 interface settings

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: RS-232-C PG/PC

Interface:	COM2
Protocol:	RTS/CTS
Parity:	
Stop bits:	1
Data bits:	8
Baud rate:	>= 9600
Archive format:	Binary format (PC format)

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.

Other settings are possible with ASCII data. They must correspond to those on the programmer. Cable 6FX 2002-1AA01- is used for this purpose.

Parameters for DIN programs

Default: RS-232-C user

Interface:	COM1	
Protocol:	RTS/CTS	
Parity:	None	
Stop bits:	1	<input type="checkbox"/> With leader and trailer
Data bits:	8	<input checked="" type="checkbox"/> 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 Operation

7.5.1 Services main screen

All programs/data stored on the hard disk or in the NC memory are listed in the "Services" basic display.

Services	CHAN1	Auto	MPF0			
Channel reset			Program aborted			
			ROV SBL1			
Programs/data: TARGET \ARC.DIR						
	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						
RS232C, disk/card, archive -> Control						
Data in	Data out	Manage data		Data selection		Interface

Explanation of main screen

Display of the current file tree

Name

The following file attributes can be displayed for each file (depending on the default setting):

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.



Length

File length in bytes (directory length is not displayed)

Date

Date of creation or date of last modification to file

Enabling

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 authorization

There are also 5 access rights for each file:

- Read Corresponds to level 5
- Write Corresponds to level 3
- Execute Corresponds to level 7
- Display 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.



Data in

Horizontal softkeys

Read in archives/files

- V.24
- PG
- Diskette (if available)
- Archive from (directory "archive" on the hard disk)

Data out

Read out archives/files

- V.24
- PG
- Diskette (if available)
- Archive to (directory "archive" on the hard disk)



You can archive data for a series start-up. The softkey is password-protected.

Series
start-upManage
data

Files/directories can be created, loaded, saved, deleted or copied and their attributes can be displayed/changed.

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.

Data
selection

Under softkey "Data selection", you can choose the directories that you wish to have displayed in the "Services" basic display.

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.

RS-232

- V.24 (RS-232)

PG

- PG

Disk

- Disk

Archives

- "Archives" directory on hard disk

NC card

- "Archives" directory on the 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. V.24 (RS-232)
2. PG/PC

Services	CHAN1	Auto	MPF0	
Channel reset			Program aborted	
			ROV	SBL1
Interface: V24 (RS232C)				
Interface:	COM1			
Protocol:	Xon/Xoff	XON (hex):	11	
		XOFF (hex):	13	
Transmission:	Normal			
Parity:	None			
Stop bits:	1			
Data bits:	8 bit			
Baud rate:	2400			
Arch. format:	Punch tape only with LF			
		Send XON at Start Data on		
		Wait for XON at Start Data off		
		With leader + trailer		
		Stop with EOT character		
		End of transmis. (hex):	1A	
		Timeout (sec.):	6	
Data in	Data out	Manage data	Data selection	Interface



Interface

Operating sequence

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); As of SW 6.2: If a parameter file is assigned to the current interface, the name of this file is displayed instead of RS-232. "Printer" in the display above.

PG

- PG/PC; As of SW 6.2: If a parameter file is assigned to the current interface, the name of this file is displayed instead of PG.

Manage-
ment

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

Services	CHAN1	Auto	MPF0
Channel reset		Program aborted	
		ROV	SBL1

Manage V24 interfaces: \V24\LASERPRINT.V24					
Name	Type	Length	Date	Time	
LASERPRINT	V24	36	15/10/2001	10:27:51	
R2570	V24	35	15/10/2001	10:32:17	

COM2, RTS/CTS, Baud rate:19200, Data bits:8, Stop bits:1

Display

New ...

Delete...

Assign...

Copy...

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 (maximum of 10 characters) the current RS-232 parameters are written to a new file in the RS-232.DIR directory.

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.

As of SW 6.2: 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.

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 line

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

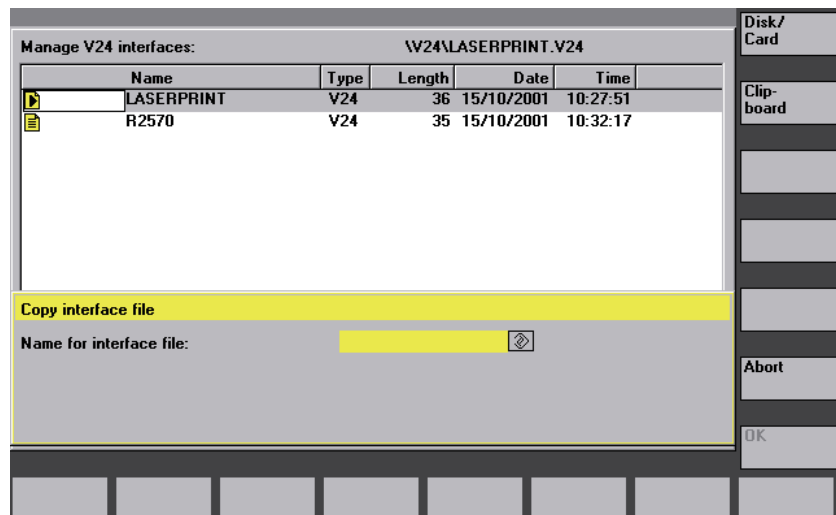
Return to "Interface: RS-232" screen form

<<

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.

Power-up

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

When 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

Operating sequence

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

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 connected to the RS-232 interface (e.g. PC)
- A diskette drive
- "Archives" directory on hard disk
- Free memory areas on the NC card

Operating sequence

The "Programs/Data" file tree is displayed.
The vertical softkey bar changes.

RS-232

PG

Diskette

Archives

NC card

Start



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.

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.

Other information

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 "Abort", otherwise all joblists are executed and archiving is then started.

7.5.5 Log



Services

Log

Manage
data

RS-232

PG

Diskette



No

Yes all

Yes

Name/type

Function

You can use the log function to view a job log, e.g. for "Manage data".

Operating sequence

The "Services" operating area is selected.

The "Log" window is opened. The source or target of the job is displayed in the header.

The vertical softkey bar changes.

Jobs are assigned to the vertical softkeys, i.e. jobs for

- "Manage data"
- RS-232 interface
- Programming device
- Floppy disk 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



Stop



Delete
log

- Abort complete job

The currently displayed log is deleted.
The log window is closed on the last log.

7.6 Manage data

7.6.1 Standardized program and data management (from SW 6.3)



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

Programs

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

Operating sequence

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

Operating sequence

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. If a job list does not exist, all the files in that directory are loaded/unloaded.

Loadable compile cycles, SW 6.3 and higher

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 Copy/paste**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



SELECT

Copy

Programs/
data

Clipboard

NC card

Diskette

Paste



Operating sequence

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

**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).

Manage
data

Delete

OK

**Operating sequence**

Select softkey "Manage data".

The horizontal and vertical softkey bars change.

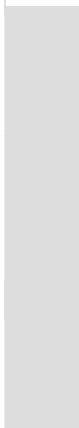
Position the cursor on the desired file.

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/directory/archive



Manage
data

Properties



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.

Operating sequence

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 is renamed B.WPD:

All files with the name A.XXX are renamed to B.XXX, i.e. the extension is retained.

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 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.



OK



Other information

- 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 a definition file is edited in the NC, when exiting the Editor you are prompted whether the definitions are to be set active.

Example:

"Do you want to activate the definitions from file GUD7.DEF?"

"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.

Unload

If a definition file is unloaded, the associated data block is deleted after a query is displayed.

OK

Abort

Activation

If a definition file is loaded, a prompt is displayed asking whether to activate the file or retain the data. If you do not activate, the file is not loaded.

If the cursor is positioned on a loaded definition file, the softkey labeling changes from "Load" to "Activate" to activate the definitions. If you select "Activate", another prompt is displayed asking whether you want to retain the data.

Data is only saved for variable definition files, not for macros.

Other information

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.

Creating a definition directory

The definition files available to the user

- SACCESS.DEF (_N_SACCESS_DEF)
- MACCESS.DEF (_N_MACCESS_DEF)
- UACCESS.DEF (_N_UACCESS_DEF)

are created and edited under the definition directory _N_DEF_DIR.

They do not need to be activated but are loaded to the passive file system of the NCK as follows:

Select softkey "Manage data".

The horizontal and vertical softkey bars change.

Loading the definition directory in the HMI without modifying the existing access authorization.

Changing write access

The following files which were installed during a series startup

- SACCESS.DEF (_N_SACCESS_DEF)
- MACCESS.DEF (_N_MACCESS_DEF)
- UACCESS.DEF (_N_UACCESS_DEF)

are created and edited under the definition directory _N_DEF_DIR.

They do not need to be activated but are loaded to the passive file system of the NCK as follows:

Definition and creation of user data

See /PGA/, Programming Guide, Advanced.

Manage
data

New

7.7 Start-up functions

7.7.1 Series start-up



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.

Operating sequence

Requirement: The password, e.g. with access level 3 (user) is set.

Press softkey "Series start-up".
The vertical softkey bar changes.

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.

MMC data
selection

RS-232

PG

Diskette...

Archives

NC card

Read start-
up archive

RS-232

PG

Diskette...

Archives

NC card

Start

Make start-
up archive

Example:

Based on the original archive name MMCNCPLC, the new archive name MMCNCPLC is suggested when you select PLC.

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
- Floppy disk drive
- "Archives" directory on hard disk
- "Archives" directory on the NC card
- The softkey label changes to "Stop".
The series start-up archive is created.

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:

- A device connected to the RS-232 interface
- A programming device
- The diskette drive
- The "Archives" directory on the hard disk
- The "Archives" directory on the NC card
- Select an archive file under softkeys "Diskette", "Archives" and "NC card".
- Start reading in the archive. The softkey label changes to "Stop".

Switchover from "Read" to "Make".



In software version 6.3
and higher

Other information

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 CC is performed by copying to data medium, e.g. a diskette. Only archives with access authorization for series machine start-up in access levels 0 to 4 can be 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 loaded onto the NC card with SINUCOPY-FFS (on an external PG/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).



Operating sequence

Requirement:

The start-up archive with the name _N_ORIGINAL_ARC is located on the NC card (in the 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 protocol window appears with the query: "Series startup archive: 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 Upgrade



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.



Upgrade

Operating sequence

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:

RS-232

PG

Diskette...

Archive...

NC card

- The device connected to the RS-232 interface
- Programming device
- Floppy disk drive
- "Archives" directory on hard disk
- The contents of the archives directory on the NC card are displayed.
Enter the name of the new archive file.

The softkey label 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 are read in via:

- A device connected to the RS-232 interface
- A programming device
- The diskette drive
- The "Archives" directory on the hard disk.
- The "Archives" directory on the NC card

Select an archive file under softkeys "Diskette", "Archives" and "NC card".

Start reading in the archive. The softkey label changes to "Stop".

RS-232

PG

Diskette...

Archives ...

NC card

Start

Notes

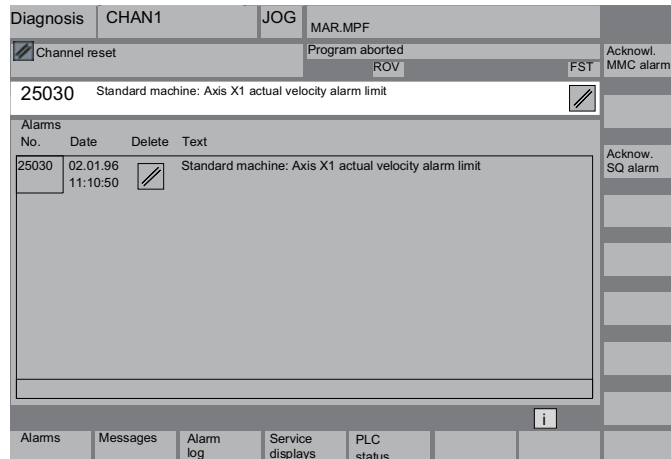
"Diagnosis" Operating Area

8.1	Diagnostics main screen	8-404
8.2	Alarms/messages/alarm log.....	8-406
8.3	Service display	8-408
8.3.1	Service overview (from SW 6)	8-408
8.3.2	Service axis.....	8-410
8.3.3	Service drive	8-411
8.3.4	Service Safety Integrated.....	8-412
8.3.5	Configuration data.....	8-416
8.3.6	Communication error log.....	8-416
8.3.7	Action log	8-417
8.3.8	Version	8-417
8.3.9	Sorting and saving version information.....	8-418
8.3.10	Displaying the version screen for cycles (from SW 6.3)	8-419
8.3.11	Display cycle versions (from SW 6.3)	8-421
8.3.12	Displaying loadable compile cycles (from SW 6.3).....	8-422
8.4	PLC status.....	8-424
8.4.1	General	8-424
8.4.2	Changing/deleting values.....	8-425
8.4.3	Symbolic PLC addresses.....	8-427
8.5	Selecting/creating operand screens for PLC status.....	8-432
8.5.1	File functions	8-433
8.6	Displaying NC system resources	8-434

8.1 Diagnostics main screen

Basic screen

A display headed "Alarms" appears when you select the operating area "Diagnosis".



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.

Delete 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 "Alarms" display.

Messages


An overview of active messages is displayed.

Alarm log

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

Service displays

You can view updated information about axes and drives installed in your system under softkey "Service displays".

	PLC status	Information on the current status of the PLC memory locations.
	Remote diagnostics	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 Diagnosis.
	Acknowl. MMC alarm	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.
	Acknowl. SQ alarm	

8.2 Alarms/messages/alarm log



Function

You can display a list of alarms and messages and acknowledge them.



Operating sequence

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 "Alarm Cancel" 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).

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.

Alarms



Messages

Alarm
log



Display
new

Save log

Sorting
old -> new

Acknowledge symbols

The following acknowledge 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:



or



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 (from SW 6)



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:

Software version 6.3 and higher

Columns for machine axes 1 to 31

Traffic light indicator states per axis

Green dot:		The axis is behaving normally
Amber dot:		The axis is not ready
Gray dot:		Does not concern this axis
Red dot:		An alarm is pending for this axis
Dash:		No drive assigned to axis
Hash:		Error reading the data if e.g. data is missing

In software version 6.3 and higher

Rows with the drive diagnostic functions

Drive enable (terminal 64/63)
Pulse enable (terminal 64/48)
Pulse enable (term. 663 / SI: drive relay)
Setup mode (terminal 112)
PLC pulse enable
NC speed controller enable

In software version 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

Operating sequence

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 axis" 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
displaysService
axisPAGE
DOWNPAGE
UPAxis
+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 ½, 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

Operating sequence

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



Service
displays

Service
drive



PAGE
UP

PAGE
DOWN

Drive
+

Drive
-

Direct
selection...

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 ½, speed setpoint, speed actual value)
- check the drive status
- display the current ramp-up phase
- display the 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

Operating sequence

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



Status SI

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

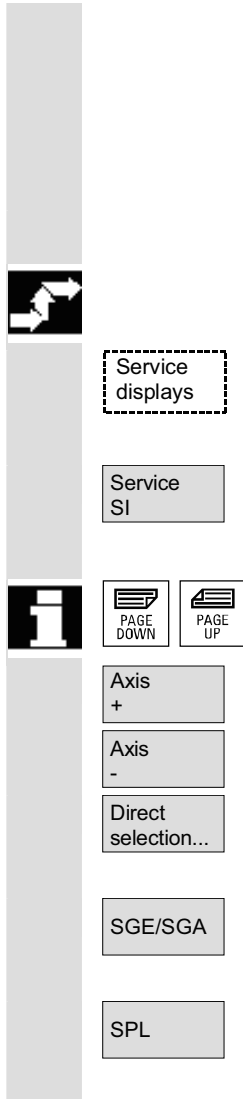
References: FBSI/, Description of Functions Safety Integrated.

Diagnosis	CHAN1_MILL	Jog	\MPF.DIR AA_JOL.MPF	
Channel reset		Program aborted		Axis +
		ROV		
Status SI				X1 1
AX1:X1 (DR1:SRM)				Direct selection
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	-	%	Status SI
Safe actual velocity limit	Inactive	-	mm/min	
Set velocity limitation	Inactive	-	mm/min	
Current velocity difference	0.000	-	mm/min	SGE/SGA
Max. velocity difference	0.000	-	mm/min	
Active safe software limit switch	1	1		
Active gear ration (stage)	1	1		SPL
Active stop	None	None		
Currently requested external stop	None	None		
Stop-F code value (alarm 300911)	-	0		
Service axis	Service drive	Service SI	System resources	Config. data
				Commun. log
				Action log
				Version

Available signals/values

The vertical softkeys axis +, axis - or direct selection are used to set the desired axis. The active axis is displayed in the top right half 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

Operating sequence

Select the menu headed "Service displays".
 The horizontal softkey bars change.

The "Service SI" window displays information about Safety Integrated data together with the associated 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.

Use this softkey to display the safety-relevant input and output signals SGE and SGA.

Use this softkey to display the safe programmable logic signals SPL.

SGE/SGA

Diagnosis	CHAN1_MILL	Jog	\MPF.DIR AA_JOL.MPF
Channel reset		Program aborted	
		ROV	
SGE/SGA X1 1			
Ax1:Ax1 (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	
<div> Service axis Service drive Service SI System resources Config. data Communic. log Action log Version </div>			

The available signals can be seen in the above screen.

The vertical softkey Status SI take you to the Status SI screen,
SPL opens the Safe Programmable Logic screen.

SPL

Diagnosis	CHAN1_MILL	Jog	\MPF.DIR AA_JOL.MPF
Channel reset		Program aborted	
		ROV	
SPL			
Variable	Bit	Current values	Format
\$A_INSE(P)	01...08	NCK 0000 0111 PLC 0000 0111	B
\$A_OUTSE(P)	01...08	NCK 0000 0001 PLC 0000 0001	B
\$A_INSI(P)	01...08	NCK 0000 0000 PLC 0000 0000	B
\$A_OUTSI(P)	01...08	NCK 0011 1111 PLC 0011 1111	B
\$A_MARKERSI(P)	01...08	NCK 1100 0011 PLC 1100 0011	B
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	
<div> Service axis Service drive Service SI System resources Config. data Communic. log Action log Version </div>			

In the "Variable" selection box, you can choose:

\$A_INSE(P) corresponds to simultaneous selection of

\$A_INSE top row; NCK origin and

\$A_INSEP bottom row; PLC origin

The following should then be selected for the remaining variables:

\$A_OUTSE(P)

\$A_INSI(P)

\$A_OUTSI

\$A_MARKERSI(P)

Under Bit you can request an 8-bit range from the selected signal.

Saving

Your variable sections and assigned bit ranges are saved and are included when the screen is activated subsequently.

In addition to the current values, the origin of the displayed NCK/PLC signals is displayed.

Format**Pre-assigned format...**

The following formats can be selected in the variable rows using the

select key	B	Binary
	H	Hexadecimal
	D	Decimal.

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.

Selectable variables

\$A_INSE(P)
\$A_OUTSE(P)
\$A_INSI(P)
\$A_OUTSI(P)
\$A_MARKERSI(P)

Displayed signals

KDV level (KDV = cross-checking)
Cross-checking status
Cross-checking control word
SPL power up status
SPL powered 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 terminated
NCK cross-checking has been started
PLC cross-checking has been started
Cyclic SPL checksum test active
All SPL protection features active

8.3.5 Configuration 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.



Service
displays

Config.
data

Operating sequence

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.



Other information

Configuration data can be read out via the RS-232 interface in the "Services" operating area.

8.3.6 Communication error log



Function

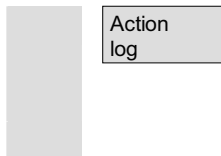
Errors that occur during communication between the HMI and NCK/PLC are registered in a communication error log.

You can display this log by pressing the softkey .

The error log file is principally used by the control manufacturer (Siemens) as a diagnostic tool for communication errors.



8.3.7 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).

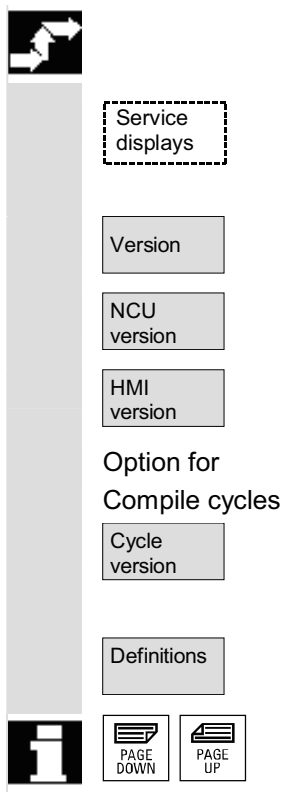
8.3.8 Version



Function

The version data of the installed system software are output in the Version display.

You can sort and save the version displays. Version displays saved as ASCII files can be edited as required or sent to the hotline in the event of an error.



Operating sequence

Press softkey "Service displays".
The horizontal softkey bars change.

Open the "Version" window in the "Service displays" menu.
The following softkeys are displayed:
"NCU version" for the version data of the NCU and

"HMI version" for the version data of the HMI.

If "compile cycles" are available on the NCK, the version is displayed.

"Cycle version" version data for each of the cycle packages (SW 6.3 and higher)

"Definitions" the version data for definitions

Use the "Page" keys to scroll up and down.

8.3.9 Sorting and saving version information

Sort

Save
versions

Specific features
of HMI versions

There are various sorting functions for the following version lists:

- HMI version
- Cycle version
- Definitions

In the left-hand part of the pane for sorting, you can select a **sort criterion** from the column designations in the list by pressing the Select key. In the right-hand pane you can choose from **descending** and **ascending** sorting sequence for the selected sort criterion.

The sorted and unsorted version displays of the display area are saved as follows for HMI versions:

Range	Target	File
HMI version	Services →Diagnostics\LogFiles	HMI.COM

How to save the other areas is described in the area descriptions.

Software objects are displayed in **red** if they are not unambiguous (e.g. several executable programs with the same name in different directories) or, if the entry in the registry file does not match the actual storage location of an object.

A language-independent message is displayed in the message line:

1: <actual registry entry> or

1: - , if there is no registry entry or

2 for all other files under a different path

The dialog messages are also written to the log file.

The checking procedure takes place when the respective version display is first accessed.

8.3.10 Displaying the version screen for cycles (from SW 6.3)



Service
displays

Versions

Package overview

Cycle
version

Package
overview



Predefined package lists

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.

Operating sequence

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 "Cycle version data" overview by pressing the horizontal softkey "Version data cycles". 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
<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 for cycles
<code>cyc_cma.cyp</code>	Manufacturer cycles (predefined name)
<code>cyc_cus.cyp</code>	User cycles (predefined name)

Details of packagesCycle
version

Details

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)
- Date
- Version entry Version

Overview ofCycle
version**All cycles**All
cycles

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.

Cycle
version**User**User
cycles**Manufacturer**Manufac-
turer cycles**Standard
Cycles**Standard
cycles

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.

DefinitionsDefi-
nitions

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.

Sort

The version information can be sorted according to the proposed criteria in either ascending or descending order.



Save
versions

The sorted and unsorted version displays of the displayed definitions are saved as follows:

Range	Target	File
Definitions	Services →Diagnostics\LogFiles	DEF.COM

8.3.11 Display cycle versions (from SW 6.3)



Function

You can save the contents of the version displays for cycles in a separate log file.



Operating sequence

To create a separate log file for the following version contents, press the "Save versions" softkey in the "Version data *type* cycles" version display:

Service
displays

Versions

Cycle
version

Save
versions

File:	Type:
CYP.COM	Package overview
CYP_DET.COM	Details
ALLCYCLE.COM	All cycles
CUS.COM	User cycles
CMA.COM	Manufacturer cycles
CST.COM	Standard cycles

When the data has been saved, the following message appears:

"File stored!"

⇒ Services:\Diagnostics\LogFiles\xxx.com.

xxx stands for the respective file name preceding the extension .COM for each type.

8.3.12 Displaying loadable compile cycles (from SW 6.3)

**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

**Operating sequence****Loadable compile cycles**

Service
displays

Compile cycles are loaded when you initiate an NCU reset and can then be displayed under Service displays/Version/Compile cycles.

Version

Open the "Version" window in the "Service displays" menu. The softkey "Compile cycles" appears.

Compile
cycles

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**.

Properties

Press the "Properties" softkey
The horizontal and vertical softkey menu changes and a "Properties" window, such as the one below for NC card, appears.

Version display

Properties			
Path	:NCcard\LoadableCompileCycles		
Name	:CCMCSC	Date:	Time
Extension	:ELF	Length:	Loaded: <input type="checkbox"/>
Type	:LoadableCompileCycle		
Access authorization			
Read:	Write:	Enable:	List: Delete:
Contents: Loadable compile cycle			
Version:	MCSC Coupling axes Machine	Time	Date
Advance version of compile cycle (Preliminary)			
Interface: 001.001@Interfaces=002.000 @TChain=001.000			
Current access authorization access level:			

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 (Ex), input byte (EBx) Input word (Ewx), input double word (Edx)
Outputs:	Output bit (Ax), output byte (Abx) Output word (Awx), output double word (Adx)
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	Condition	Writing	Format	Value	Range
Inputs		Yes	Yes			0-127
	I 2.0			B	0	
	IB 2			B	0101 1010	
				H	5A	
				D	90	
Outputs		Yes	Yes			0-127
	Q20.1			B	1	
	QB 20			B	1101 0110	
				H	D6	
				D	214	
Markers		Yes	Yes			0-255
	M 60.7			B	1	
	MB 60			B	1101 0110	
	MW 60			H	B8	
				D	180	
Timers	T20	Yes	no			0-31
				B		
				H		
				D		



Operand	Example	Condition	Writing	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.



Operating sequence

The "Diagnosis" operating area is selected.

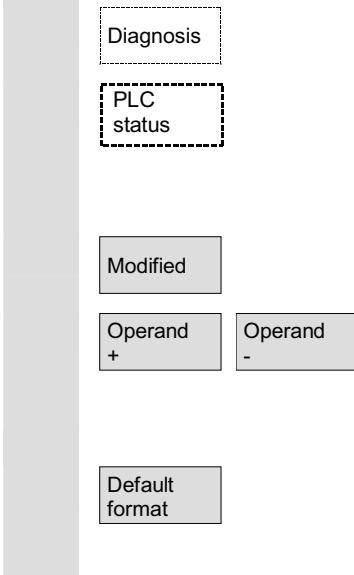
Press softkey "PLC status".
The first operand screen form appears.
The vertical softkey bar changes.

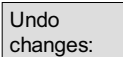
Cyclic updating of the values is interrupted.

You can increase or decrease the address of the operand by 1 place at a time.

Softkey assignment

A selection window appears.
You can preset the format fields to "B" (binary), "H" (hexadecimal), "D" (decimal) or "G" (floating point).



DeleteUndo
changes:Accept

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.

Other information

Press the "information key".

A description of the permissible input syntax for the PLC status display is overlaid.

8.4.3 Symbolic PLC addresses



Preconditions

Language

Function

With HMI Advanced 6.3 you can also edit the PLC information using symbols. Prerequisite is that the symbol tables and texts for the symbols in the PLC project have been suitably prepared and made available in HMI.

Other information

The program **PlcSymbolsGenerator.exe** and its description are located in the PLC tool box. It creates the files with symbols and language-dependent texts in the representation required for the HMI.

- Save the files created with PlcSymbolsGenerator.exe in the Services operating area under path: Diagnostics/PLC Data (F:\DH\DG.DIR\PLC.DIR\) with the following predefined names and extensions:
PlcSym.SNH (symbols)
PlcSym_<language extension>.SNT (texts e.g. PlcSym_GR.SNT)
- NC RESET or HMI reboot for file activation
- (For M:N configurations only): There must be an entry which makes the symbols available in netnames.ini for each addressable NCU. See IAM, IM4.

If the texts (symbol comments) are stored for multiple languages, you can also change language to work with texts in the other languages.



Operating sequence

The "Diagnosis" operating area is selected. If the above listed requirements are met, the softkey PLC symbols is displayed.

Diagnosis

PLC
symbols

Diagnose	Labor_Kanal1	JOG Ref	\MPF.DIR ABCKOPIE.MPF
Kanal RESET			Programm abgebrochen
PLC-Symbole: Übersicht			
Symbolische Adresse	Absol. Adresse	Format	Wert
TMLoadIF..IF[7].Free2	DB71.DBD192	H	#
TMLoadIF..IF[7].Free1	DB71.DBD188	H	#
TMLoadIF..IF[6].Free3	DB71.DBD166	H	#
TMLoadIF..IF[6].Free2	DB71.DBD162	H	#
TMLoadIF..IF[1].Free3	DB71.DBD16	H	#
TMLoadIF..IF[6].Free1	DB71.DBD158	D	#
TMLoadIF..IF[5].Free3	DB71.DBD136	H	#
TMLoadIF..IF[5].Free2	DB71.DBD132	H	#
TMLoadIF..IF[5].Free1	DB71.DBD128	H	#
TMLoadIF..IF[1].Free2	DB71.DBD12	H	#
TMLoadIF..IF[4].Free3	DB71.DBD106	H	#
TMLoadIF..IF[4].Free2	DB71.DBD102	H	#
TMLoadIF..IF[4].TMno	DB71.DBB97	H	#
TMLoadIF..IF[4].Channel	DB71.DBB96	H	#

The following are represented in the table from left to right:

- Symbolic address
- Absolute address
- Format
- Value

Vertical softkeys

Sort

Select a sort option from:

PLC-Symbole: Sortieren	
<input type="radio"/>	gemäß Symboltabelle
<input type="radio"/>	symbolische Adresse alphanumerisch aufsteigend [A -> Z]
<input checked="" type="radio"/>	symbolische Adresse alphanumerisch absteigend [Z -> A]
<input type="radio"/>	absolute Adresse alphanumerisch aufsteigend [A -> Z]
<input type="radio"/>	absolute Adresse alphanumerisch absteigend [Z -> A]

Depending on your selection, sorting takes place on column one or column two in ascending or descending alphabetical order to make it easy to find a symbol or address. "As in symbol table" means: In the same sequence as the symbol table in the PLC project.

Filter

Find/
go to...

Start

Select the categories of symbols/addresses that you want to display. Press OK to activate the filter. Filters are retained after you exit screens/operating areas and after reboot.

PLC-Symbole: Filter

- ☒  **Eingänge**
- ☐ **Ausgänge**
- ☒ **Datenbausteine**
- ☐ **Merker**
- ☒ **Timer**
- ☒ **Zähler**

After entering a search term and search direction, the section of the table is displayed with the focus on the symbol/absolute address if it was found.

The search is performed in the following sequence: absolute addresses, symbols.

When you enter the search term in the "Address" field, take care to enter the correct notation. Press OK to start the search.

Diagnose	Labor_Kanal1	JOG Ref	\MPF.DIR ABCKOPIE.MPF	
Kanal RESET			Programm abgebrochen	Anfang
				Ende
PLC-Symbole: Übersicht				aktiv
Symbolische Adresse	Absol. Adresse	Format	Wert	
TMLoadIF.IF[7].Free2	DB71.DBD192	H	##	
TMLoadIF.IF[7].Free1	DB71.DBD188	H	##	
TMLoadIF.IF[6].Free3	DB71.DBD166	H	##	
TMLoadIF.IF[6].Free2	DB71.DBD162	H	##	
TMLoadIF.IF[1].Free3	DB71.DBD16	H	##	
TMLoadIF.IF[6].Free1	DB71.DBD158	H	##	
TMLoadIF.IF[5].Free3	DB71.DBD136	H	##	Suchen
TMLoadIF.IF[5].Free2	DB71.DBD132	H	##	
TMLoadIF.IF[5].Free1	DB71.DBD128	H	##	
TMLoadIF.IF[1].Free2	DB71.DBD12	H	##	
TMLoadIF.IF[4].Free3	DB71.DBD106	H	##	
PLC-Variablen: Suchen ab Cursorposition				Abbruch
Adresse	if			
Suchrichtung	rückwärts			Ok

If the search term is not found, the dialog box remains open for a new search. After you change screens, the search term and search direction are retained in the dialog box.

Press this softkey to jump to the beginning of the symbol table.

End

Press this softkey to jump to the end of the symbol table.

Details

All information is displayed for the selected address or symbol in the following representation:

The screenshot shows the 'Diagnose' screen with the 'Labor_Kanal1' selected. The 'JOG Ref' is 'VMPF.DIR' and 'ABCKOPIE.MPF'. The 'Kanal RESET' button is visible. The '510104' address is highlighted in red, and the value '0000' is displayed. Below this, the 'PLC-Symbole: Übersicht' table is shown, listing various symbols and their addresses. The 'B_MAG_SKZ_Magazin' symbol is selected, and its details are shown in the 'PLC-Variablen: Details' section. The details include the symbolic address 'B_MAG_SKZ_Magazin', absolute address 'MB162', and description 'MAG: Schrittkettenzaehler Magazin'. The format is set to 'Hexadezimal' and the value is '00'. The 'Ändern' (Change) softkey is visible on the right side of the screen.

Symbolische Adresse	Absol. Adresse	Format	Wert
B_MA_SKZ_MaschineEin	MB131	H	00
B_MA_SKZ_Oelstand	MB130	H	00
B_MA_SKZ_Schmierung	MB133	H	00
B_MA_SKZ_Schmierzyklus	MB134	H	00
B_MAG_SKZ_Magazin	MB162	H	00

Symbolische Adresse	B_MAG_SKZ_Magazin
Absolute Adresse	MB162
Beschreibung	MAG: Schrittkettenzaehler Magazin
Zusatz-Information	MAG: Schrittkettenzaehler Magazin
Format	Hexadezimal
Wert	00

If relevant texts are available in the PLC project, they are displayed here as "description".

Modified

You can change the value of the symbol that is currently displayed in detail.

The input marker is positioned on Value. Value input is completed by pressing the Enter key; however, the values are only activated when the "Accept" softkey is pressed. You can also choose a more suitable display format.

Accept

The entered value is transferred to the PLC interface.

Modified

You can change the value for the selected symbol. Value input is completed by pressing the Enter key; however, the values are only activated when the "Accept" softkey is pressed. You can also choose a more suitable display format. Before pressing the Accept softkey to validate the data changes, you can modify any number of symbol values in sequence.

Accept

The previously entered values are transferred to the PLC interface.

Diagnosis

PLC status

Symbols

Symbols in the status screen

If the above mentioned requirements are met, the additional "Symbol" softkey is displayed in the PLC status screen.

Symbol / Operand	Format	Wert
DB19.DBB26	H	01
DB19.DBX26.0	H	1
DB19.DBW13	B	0000000000000001
	H	
B_MA_Entprell_Zaehler	D	0
B_MA_SKZ_HydraulikEin	H	00
	H	
	H	
	H	
	H	

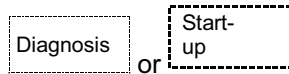
You can use the softkey to switch between absolute addresses and symbols for inputs and display in the status screen.

If a symbol is selected, the softkeys "Operand+" and "Operand-" defined for absolute addresses are not available.

Otherwise, the above description for PLC status also applies to the status screen.

8.5 Selecting/creating operand screens 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.

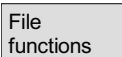
**Operating sequence**

The operating area "Start-up" is selected.



Select softkey "PLC".

The horizontal and vertical softkey bars change.

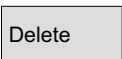


Select softkey "File functions".

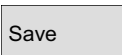
The vertical softkey bar changes.

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:



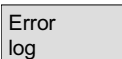
The selected operand back-up file is deleted.



The selected operands are saved to the specified file.

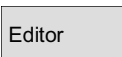


The selected operand file is loaded to the "PLC status" window for processing.



If errors occur during transfer of the machine data, these are entered in an error log.

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



Diagnosis

PLC status

File
functions

Delete

Save

Load



Function

You can use the file functions to manage operand screens.

Operating sequence

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.

Other information

The operand screen forms are ASCII files.

8.6 Displaying NC system resources**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.

**Operating sequence**

The "Diagnosis" operating area is selected.

Press softkey "Service displays" then "System resources:."

The "NC utilization" screen 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".

Diagnosis

Service
displays

System
resources

NC

Stop

Start



"Startup" Operating Area

9.1	Machine data.....	9-441
9.1.1	Display options: Suppression filter	9-443
9.2	User views.....	9-445
9.3	NC	9-447
9.4	PLC	9-447
9.4.1	PLC status.....	9-447
9.4.2	Set time/clock.....	9-448
9.4.3	File functions	9-448
9.5	Drives/servo	9-449
9.6	HMI.....	9-451
9.6.1	Changing the HMI interface	9-451
9.6.2	System settings.....	9-454
9.6.3	Transferring display MD from HMI Embedded	9-457
9.7	Tool management	9-458

Start-up main screen



Danger

Changes in the "Start-up" operating area have a major impact on the machine. Incorrect parameterization may 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 manufacturer
- Service personnel
- Machine users (machine setters)

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.

Start-up	CHAN1	JOG	MPF.DIR MAR.MPF																																					
<input checked="" type="checkbox"/> Channel reset		Program aborted ROV		LCD brighter																																				
				LCD darker																																				
Machine configuration																																								
<table border="1"> <thead> <tr> <th colspan="3">Machine axis</th> <th colspan="2">Drive</th> <th rowspan="2">Channel</th> </tr> <tr> <th>Index</th> <th>Name</th> <th>Type</th> <th>Number</th> <th>Type</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>X1</td> <td>Linear axis</td> <td>6</td> <td>FDD</td> <td>1</td> </tr> <tr> <td>2</td> <td>Y1</td> <td>Linear axis</td> <td>7</td> <td>FDD</td> <td>1</td> </tr> <tr> <td>3</td> <td>Z1</td> <td>Linear axis</td> <td>10</td> <td>FDD</td> <td>1</td> </tr> <tr> <td>4</td> <td>A1</td> <td>Spindle</td> <td>14</td> <td>MSD</td> <td>1</td> </tr> </tbody> </table>						Machine axis			Drive		Channel	Index	Name	Type	Number	Type	1	X1	Linear axis	6	FDD	1	2	Y1	Linear axis	7	FDD	1	3	Z1	Linear axis	10	FDD	1	4	A1	Spindle	14	MSD	1
Machine axis			Drive		Channel																																			
Index	Name	Type	Number	Type																																				
1	X1	Linear axis	6	FDD	1																																			
2	Y1	Linear axis	7	FDD	1																																			
3	Z1	Linear axis	10	FDD	1																																			
4	A1	Spindle	14	MSD	1																																			
Current access level manufacturer																																								
Machine data	User views	NC	PLC	Drives Servo	MMC																																			
					Tool managem.																																			

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 function PLC status is available.

You can update 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 and measuring functions. See also Section "Drive/servo".

For more information about start-up see /IAD/, Installation and Start-Up Guide, Drive Optimization with Start-up Tool

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.

Machine data

User views

NC

PLC

Drives servo



MMC

Tool management


 LCD
brighter

 LCD
darker

 Change
language

 NCK
Reset

Password...

Vertical softkeys

Only for OP 010 with STN display:

You can change the screen brightness.

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.

There are access levels 0 to 7,

- 0 is the highest and
- 7 is the lowest.

Protection levels

- 0 to 3 is controlled by means of **passwords** and
- 4 to 7 by means of keyswitch settings.

The operator only has access to information protected by one particular level and the levels below it. The machine data is assigned different protection levels as a standard measure.

Protection level	Locked by	Range
0	Password	Siemens
1	Password	Machine manufacturer
2	Password	Installation engineer, service
3	Password	End user
4	Keyswitch position 3	Programmer, machine setter
5	Keyswitch position 2	Qualified operator
6	Keyswitch position 1	Trained operator
7	Keyswitch position 0	Semi-skilled operator

Depending on the authorization level, it will be possible to edit data such as cycles and machine data.

Access levels 0 to 3 require the input of a password. The password for level 0 provides access to all data areas. At system boot in Startup mode (NCK startup switch in position 1), the default passwords are entered as standard.

Press softkey "Password" to set default passwords 1 to 3 which were entered at system boot after activation to the required standard for the software version.

If one of the above passwords is set, the keyswitch position is ignored.

For further information, please see

/IAD/, Installation and Start-Up Guide, "Parameterization of the control".

/IAM/, IM4 Start-up HMI Advanced, "Function/Parameters"

Operating sequence

The operating area The operating area "Start-up" is selected.

Press softkey "Password".

You can activate the password for the system via the horizontal softkeys Set password, Delete password and Change password.

Press softkey "Set password".

The input window for entering the password is displayed.

Enter one of the three possible default passwords and press the Input key or symbol to confirm. A permissible password is acknowledged as set and the currently applicable access level is displayed. Invalid passwords will be rejected. You must delete the old password before activating a password for a lower access level than the one activated.



Start-up

Password...

Set
password

Delete
password

Press softkey "Delete password".

After pressing the "Delete password" softkey the valid password is deleted and deletion is acknowledged. The currently valid access level: keyswitch position 0 is set.

Change
password.

If one of the three default passwords is set, proceed as follows to change them. Press the "Change password" softkey. The input box appears.

"Change password".

Access level

- System ☐
- **Manu-
facturer** ☒
- Service ☐
- User ☐

Please enter new password:

Please confirm password:

The modified password must be confirmed in both input fields then confirmed by pressing softkey "OK". Both entered passwords must match for the modified one to become valid.

9.1 Machine data



Areas

Danger

Changes in the machine data have a considerable influence on the machine. Incorrect parameterization may endanger human life and cause damage to the machine.

Access to the Machine data operating area can be controlled by keyswitch or password.

Function

The machine data are divided into 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 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) , with field index if applicable.
- Value of the machine data.
- Unit of the machine data.
- 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 (meter/second squared): Acceleration
rev/s**3	rev/s ³ (revolution/second to the power of 3): Change in rate of acceleration for rotating axis
kg/m**2	kgm ² (kilogram/meters squared): Moment of inertia
mH	mH (millihenry): Inductance
Nm	Nm (Newton meters): Torque
us	μs (microseconds): Time
μA	μA (microamperes): Amperage
μVs	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 = Immediately active
- cf = When confirmed via softkey "Activate MD immediately"
- re = Reset
- po = POWER ON (NCK Power ON reset)

/IAD/, IAC/ Installation and Start-Up Guide 840D, 810D

Operating sequence

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: suppression 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 applies:

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).

From SW 6.4 group allocation is implemented for display machine data. For more information, see display machine data filter.

Filter criteria

The following table shows the criteria for displaying machine data in the order in which they are evaluated:

Criterion	Test
1. Access rights	If the level of access authorization is not sufficient, the MD is not displayed. Otherwise criterion 2 is checked.
2. Suppression 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 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 suppression 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>
------------------	--

SW 6.4 and higher

Display machine data filter

The display machine data filter allows you to hide machine data and only show selective machine data as "display options" according to their specific field of use. For example, machine data can be grouped according to their affiliation:

Display groups

H01	<input type="checkbox"/>
H02	<input type="checkbox"/>
H03	<input type="checkbox"/>
H04	<input type="checkbox"/>
H05	<input type="checkbox"/>
	<input type="checkbox"/>

Area of application**ShopMill****ShopTurn****ManualTurn****Access levels**

All display MD which determine the access *

Standard machine

HMI Advanced or HMI Embedded

All others

If no group bit is set.

* protection for specific functions.

☒ **MD display filter active**

If the display filter is active, only the set display groups are displayed. With "All others" only the machine data are displayed for which no filter bit is set. The softkeys behave in the same manner as with the display options for general machine data. The settings are saved.

Initialize

When you open a machine data window, the filter setting that matches the area is automatically updated.

Operating sequence

The operating area The operating area "Start-up" is selected.

Start-up

Machine data

Display MD

Display options

Press softkey "Machine data".

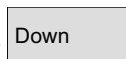
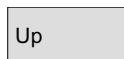
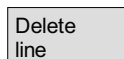
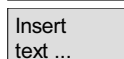
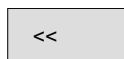
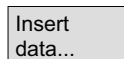
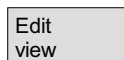
The horizontal and vertical softkey bars change.

Press "Display MD" softkey.

The horizontal and vertical softkey bars change.

Select the softkey "Display options", a list of all the ranges that can be displayed/hidden appears.

9.2 User views



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.

Manage views

Assign softkey ...

Delete

Save

Load

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.

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



Setting a key position for the startup switch with appropriate access rights.

View and edit the NCU/CCU address.

It is normally only necessary to edit the address for M : N operation. You cannot undo an address change with general reset. The address change is not saved in an NC series startup archive.

9.4 PLC



- Changes can only be made to the PLC operands with the appropriate 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 may endanger human life and cause damage to the machine.

9.4.1 PLC status



See Chapter 8, "PLC Status".

9.4.2 Set time/clock



Function

You can change the date and time of the PLC and synchronize the date and time of the PLC and HMI.



Operating sequence

The operating area The operating area "Start-up" is selected.

Start-up

PLC

Set
date/clock

Accept

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.

The performed synchronization can be verified in the "Current:" input field.

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

The values set are retained when the software is rebooted.




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

		<p>This menu contains functions for</p> <ul style="list-style-type: none"> • Current control loop • Speed control loop • Position control loop • Function Generator • Circularity test • Servo trace • DAC configuration
	<p>SW 6.2 and higher</p>	<p>Servo trace involves the recording of up to 10 bit signals during the integration period of bit-encoded Safety Integrated signals.</p> <p>See: /FBA/ Description of Functions, Drive Functions and /FBSI/ Safety Integrated</p>
<div data-bbox="284 987 408 1048" data-label="Text">Start-up</div> <div data-bbox="284 1375 408 1435" data-label="Text">Meas. parameters</div>		<p>Measurement of coupled axes</p> <p>For startup of SIMODRIVE 611 digital drives, coupled axes are supported for measuring in the current control loop, speed control loop or position control loop, e.g.:</p> <ul style="list-style-type: none"> • Pure gantry axis grouping from SW 5.1 <p>And from SW 6.4:</p> <ul style="list-style-type: none"> • Pure master/slave relationships • Hybrid coupling of master/slave axes with gantry axes <p>You can enter specific measuring parameters and select a measurement for each available axis in these axis groupings.</p> <p>The following measuring parameters are available:</p> <ul style="list-style-type: none"> • Amplitude leading axis or master axis • Amplitude synchronized axis/axes or slave axis/axes • Bandwidth of the frequency range to be analyzed • Information on measuring accuracy and measurement duration increase • Delay settling time • Offset to accelerate via an acceleration ramp <p>For further information about the meaning of this measurement parameter, please see:</p> <p>/IAD/ Installation and Start-Up Guide, Section "Measuring functions for coupled axes"</p>

SW 6.4 and higher

Display all coupled axes

You can record the results simultaneously for up to two axes. There is always **only one** leading axis. All other axes are synchronized axes; their amplitude can be assigned separately.

All active axes of the grouping are displayed. With pure gantry axis groupings,

- A max. of one leading axis and two following axes are visible.
- Master/slave couplings, one master and two slave axes.

With hybrid master/slave with gantry axis coupling

- the leading axis is always one from the gantry axis grouping.
All other axes are synchronized axes.

The texts displayed in the operator interface change and are updated according to the active type of pair.

You can navigate through all active axes in the "Gantry measurement" or "Master / slave measurement" selection screens where all active couplings are displayed. You can measure up to 2 axes simultaneously. After each selection of the axis you need to repeat the measuring process.

Other information

The axis couplings show the axis identifiers e.g. X1, Z1 or A1 as well as the axis number of the respective axis grouping. The values have the following meanings:

SRM Synchronous motors (synchronous rotation motor)

ARM Asynchronous motors (asynchronous rotation motor)

The HMI does not support initiation of a measurement job for a master/slave coupling in the position-control loop.

If you are currently performing a position control measurement on an axis which is coupled but not initiated by the HMI, start is inhibited for this slave axis and a message is issued.

9.6 HMI

9.6.1 Changing the HMI interface



Function

You can make individual settings to your HMI and save them.



Operating sequence

The operating area "Start-up" is selected.

Start-up

MMC

Select softkey "MMC".

The horizontal and vertical softkey bars change.

Colors

The following **submenus** are available via the horizontal softkeys:

The "Color Settings" menu is opened.

You can either define the color scheme of your user interface yourself

- User

or change it according to certain specifications

- VGA
- VGA positive
- Monochrome
- Monochrome positive.

change it.

Save

The current color settings are saved.

Language

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

Depending on the HMI variant in use (HMI and MCI2 board in the PCU [internal HMI] / HMI operating an 840Di via OPI or MPI [external HMI] / standard configurations 840D/810D), you only have write access to the parameters you can actually make changes to. You can make the following settings in the "Operator panel front interface parameters" menu:

- Connection
 - 1:1 (1NC and 1 HMI) or
 - m:n (1/multiple NC(s) and 1/multiple HMI(s)), not for 840Di
- Baud rate ("Bus")
 - OPI (1.5 Mbit/s)
 - MPI (187.5 Mbit/s)
 - Softbus MC
- Highest bus address (15 - 31 available)
- Network address
 - HMI address (local address which is sent to the bus)
 - NCK address (address with which the communication is set up)
 - 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.

Internal HMI for 840Di:

With internal HMI which is only available for the 840Di, the bus is always a SOFTBUS and cannot be changed. M:N is not possible. The default address for NCK is 3 and 2 for the PLC. It can be changed in the range from 1 to the highest bus address.

External HMI for 840Di:

The same bus settings as before are available for the external HMI: MPI (187.5 kBaud) , OPI (1.5 Mbaud) and the entry MCI2 (840DI-187.5 kBaud), which can be selected via a toggle box.

M:N is not possible with MCI2. There are no setting options for the NCK address as the NCK is addressed via the PLC and therefore always has the same address as the PLC.

The default address for the PLC is 2. It can be changed in the range from 1 to the highest bus address.

Bus
node

List of addresses of active nodes that can be activated with "Update".

System
settings

See "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 prot.
- Date
- Clock
- Enabling of
- Max. display levels (branch to directory trees, max. 7)
- Max. name length (max. 25 characters)

Your settings are automatically displayed in the "Forecast" window.



Inquiry



Request confirmation before

- Deleting 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 symbol () as text .

Templates

Action
log

Trace

When creating a new workpiece, specify here whether templates for:

- Job lists
- Part programs or
- Initialization programs

are to be transferred to the new workpiece (directory).

For more information, see Subsection 6.1.5 Templates

When you press the softkey, a screen is opened for setting parameters and activating the action log. You can make and save the following settings:

- Logging on
- Alarm status change;
- Keys (ShopMill only)
- Channel status/override;
- Data writing
- File access
- Instructions (PI services program invocation)

Input fields are available for:

Write interval
Log file size
Logged alarms

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.



Start-up

MMC

System
settings

File
display

Inquiry

Operating sequence

The operating area "Start-up" is selected.

Select softkey "MMC".

The horizontal and vertical softkey bars change.

Next select one of the desired functions:

Horizontal softkeys:

The "Settings for file display" window is opened.

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

For more information, see Subsection 6.1.5 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:

- Descending or
- Ascending

Confirm with OK.

In the operating areas

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, as in (1.)

The selection of another sorting criterion in the operating area changes the sorting criterion for all operating areas (machine, program, services).

Other information

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.



Save

Position the cursor on the desired point and perform the settings.

Transfers your settings to system.

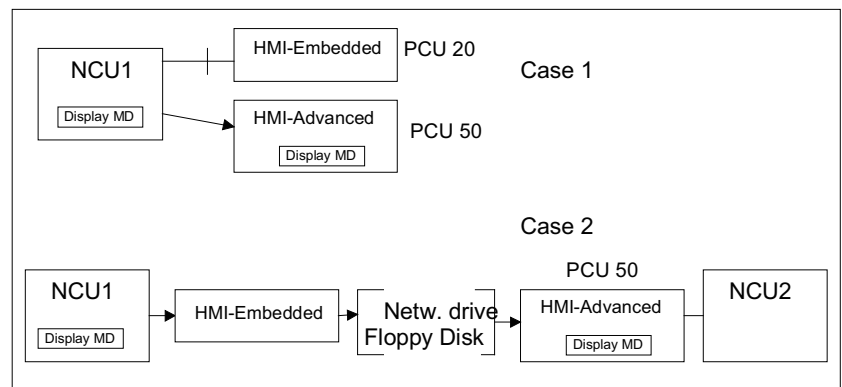
9.6.3 Transferring display MD from HMI Embedded

Use

When migrating from the HMI component HMI Embedded (PCU20) to HMI Advanced (PCU50/70) you may like to transfer tried and proven display machine data from the HMI Embedded configuration to the HMI Advanced configuration, e.g. for use with ShopMill/ShopTurn. The following section describes how to proceed for to transfer the machine data.

Initial situation

Display machine data reside on the NCU for HMI Embedded, and on the hard disk of the PCU50/70 for HMI Advanced.



Handling

The transfer consists of two steps:

- Copy the display machine data with HMI Embedded
- Read into the PCU50/70 under HMI Advanced

Case 1 acc. to diagram

On the HMI Advanced system: **Services** operating area

Transfer display machine data to the display if not yet activated:

Services → Select data

Select "Display machine data"

OK

Copy display machine data:

Services → Manage data

Display machine data\BDxxyy.tea

Copy

Target directory

Select the target directory
Diagnostics\MachDat\OperatorPanel
Insert

Activate inserted display machine data via **Start-up**:

Start-up → Machine data →
Display MD → File functions
Set up directory and file BDyy.tea
→Load

Other information

HMI Embedded manages more display machine data than HMI Advanced. An entry is made in the error log for each proposed display machine data that is not defined in HMI Advanced or contains an error.

Case 2 acc. to diagram

Transfer display MD to floppy disk or network drive:

On the **HMI Embedded** system:

Program → ETC →
Set up operator data BDx.tea
→Copy
→ ETC

Select destination medium: Network drive or floppy disk

Insert

On the **HMI Advanced** system:

Services → Manage data

Select transfer medium: Network drive or floppy disk

Select BDxxyy.tea

→Copy

Continue as for the 1st case, destination directory, see above.

9.7 Tool management**Function**

/FBA/ Description of Functions, Tool Management



Maintenance

10.1	Operating data	10-460
10.2	Cleaning	10-461

10.1 Operating data**Operating data**

	Value
Air humidity, humidity class to DIN 40040	F
Atmospheric 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	IP 54
• Back of operator panel	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.

It is resistant to:

1. Greases, oils, mineral oils
2. Bases and lyes
3. Detergent solutions and
4. Alcohol

Solvents such as chlorinate hydrocarbons, benzene, esters and ethers should be avoided!



Notes

Appendix



A	Abbreviations	A-464
B	Terms	A-474
C	References	A-497

A Abbreviations

AS	Automation System
ASCII	American Standard Code for Information Interchange
ASIC	Application Specific Integrated Circuit
ASUB	Asynchronous subprogram
BCD	Binary Coded Decimals
BCS	Basic Coordinate System
BIN	B inary files
BIOS	Basic Input Output System
BOT	Boot Files: Boot files for SIMODRIVE 611D
BP	Basic Program
C Bus	Communications bus
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

CRT	Cathode Ray Tube: Picture tube
CSB	Central Service Board: PLC module
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.
DCD	Data Carrier Detect
DDE	Dynamic Data Exchange
DIN	Deutsche Industrie Norm (German Industry Standard)
DIO	Data Input/Output: Data transfer display
DIR	Directory
DLL	Dynamic Link Library
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
DTE	Data Terminal Equipment
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
ERROR	Error from printer
FB	Function Block
FBS	Slimline screen
FC	Function Call: Function block in PLC
FDD	Floppy Disk Drive
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
FM	Function Module
FM-NC	Function Module Numerical Control
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)
IF	Pulse enable of drive module
IK (GD)	Implicit communication (global data)
IKA	Interpolative Compensation
IM	Interface Module
IMR	Interface Module Receive: interface module for receiving data
IMS	Interface Module Send: interface module for transmitting data
INC	Increment

INI	Initializing Data
IPO	Interpolator
IS	Interface signal
ISA	Industry Standard Architecture
ISO	International Standardization Organization
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)
LCD	Liquid Crystal Display
LEC	Leadscrew Error Compensation
LED	Light Emitting Diode
LF	Line Feed
LUD	Local User Data
MB	Megabyte
MC	Measuring Circuit
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
MS	MicroSoft (software manufacturer)
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
NRK	Numeric Robotic Kernel: name of NCK operating System
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
OSI	Open Systems Interconnection: standardization for computer communication
P bus	I/O (peripherals) bus
PC	Personal Computer

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
POS	POSitioning
RAM	Random Access Memory (read-write memory)
REF	Reference point approach function
REPOS	Reposition function
RISC	Reduced Instruction Set Computer: type of processor with small instruction set and ability to process instructions at high speed
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
SDB	System Data Block
SEA	Setting Data Active: File identifier for setting data

SFB	System Function Block
SFC	System Function Call
SK	Soft key
SKP	Skip block
SM	Stepper motor
SPF	Subprogram File
SR	Subroutine
SRAM	Static RAM (battery-backed)
SSI	Serial Synchronous Interface
STL	Statement List
SW	Software
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
μC	Micro controller

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

A spline

The A spline runs tangentially through the programmed interpolation points (3rd degree polynomial).

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

In order to obtain the optimum acceleration gradient for the machine while providing effective protection for the mechanical components, the machining program offers a choice between instantaneous acceleration and continuous (smooth) acceleration.

Access rights

The CNC program blocks and data are protected by a 7-level system of access restrictions:

- Three password levels for system manufacturers, machine manufacturers and users and
- Four keyswitch settings which can be evaluated via the PLC.

Activate/deactivate

Working area limitation is a means of restricting the axis movement over and above the restrictions imposed by the limit switches. A pair of values delimiting the protected zone area can be specified for each axis.

Address

An address is the designation for a specific operand or operand area, e.g. input, output, etc.

Alarms

All -> messages and alarms are displayed in plain text on the operator panel. Alarm text also includes the date, time and corresponding symbol for the reset criterion.

Alarms and messages are displayed separately.

1. Alarms and messages in the parts program
Alarms and messages can be displayed directly from the parts program in plaintext.
2. Alarms and messages from PLC
Alarms and messages relating to the machine can be displayed from the PLC program in plaintext. No additional function block packages are required for this purpose.

Analog input/output module

Analog input/output modules are signal transducers for analog process signals.
Analog input modules convert analog measured values into digital values that can be processed in the CPU.
Analog output modules convert digital values into manipulated variables.

Approach fixed machine point

Approach motion towards one of the predefined -> fixed machine points.

Archiving

Exporting files and/or directories to an **external** storage device.

Asynchronous subroutine

- A parts program that can be started asynchronously (or independently) of the current program status by means of an interrupt signal (e.g. "High-speed NC input" signal).

Automatic

Control system operating mode (block-sequential to DIN): Mode in NC systems in which a -> parts program is selected and continuously executed.

Auxiliary functions

Auxiliary functions can be used to pass -> parameters to the -> PLC in -> parts programs, triggering reactions there which are defined by the machine manufacturer.

Axes

CNC axes are classified according to their functional scope as:

- Axes: Interpolative path axes
- Positioning axes: Non-interpolative infeed and positioning axes with axis-specific feedrates; axes can move across block limits. Positioning axes need not be involved 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 and Z for a right-handed rectangular -> coordinate system.
-> Rotary axes rotating around X, Y, Z are assigned the identifiers A, B, C. Additional axes, which are parallel to those specified, can be identified with other letters.

Axis name

See -> axis identifier

B**B spline**

The programmed positions for the B spline are not interpolation points, but merely "check points". The curve generated does not pass directly through these check points, but only in their vicinity (1st, 2nd or 3rd degree polynomial).

Back up

A copy of the memory contents stored on an external device.

Backlash compensation

Compensation of a mechanical machine backlash, e.g. backlash due to reversal of leadscrews. The backlash compensation can be entered separately for each axis.

Backup battery

The backup battery provides non-volatile storage for the -> user program in the -> CPU and ensures that defined data areas and flags, timers and counters are retentive.

Backup memory

The backup memory guarantees the backup of memory areas of the -> CPU without backup battery. A programmable number of timers, counters, flags and data bytes can be backed up.

Base axis

Axis whose setpoint or actual value is employed in calculating a compensatory value.

Basic coordinate system

Cartesian coordinate system, is mapped onto machine coordinate system by means of 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.

Baud rate

Rate at which data transmission takes place (bit/s).

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.
A distinction is made between -> main blocks and -> subblocks.

Block search

The block search function allows selection of any point in the parts program at which machining must start or be continued. The function is provided for the purpose of testing parts programs or continuing machining after an interruption.

Booting	Loading the system program after Power ON.
Bus connector	A bus connector is an S7-300 accessory that is supplied with the -> I/O modules. The bus connector extends the -> S7-300 bus from the -> CPU or an I/O module to the next adjacent I/O module.
C	
C axis	Axis about which the tool spindle describes a controlled rotational and positioning movement.
C spline	The C spline is the best known and the most widely used spline. The spline passes through each of the interpolation points at a tangent and along the axis of curvature. 3rd-degree polynomials are used.
Channel	A channel is characterized by the fact that it can process a -> parts program independent of other channels. A channel exclusively controls the axes and spindles assigned to it. Parts program sequences of different channels can be coordinated through -> synchronization.
Channel structure	The channel structure makes it possible to process the -> programs of individual channels simultaneously and asynchronously.
Circular interpolation	The -> tool is required to travel in a circle between defined points on the contour at a specified feed while machining the workpiece.
CNC	-> NC
CNC high-level language	The high-level language offers: -> user variables, -> predefined user variables, -> system variables, -> indirect programming, -> arithmetic and angular functions, -> relational and logic operations, -> program jumps and branches, -> program coordination (SINUMERIK 840D), -> macros.
CNC programming language	The CNC programming language is based on DIN 66025 with high-level language expansions. The -> CNC programming language and -> high-level language expansions support the definition of macros (sequenced statements).
COM	Numerical control component for the implementation and coordination of communication.
Compensation axis	Axis having a setpoint or actual value modified by the compensation value.

Compensation table	Table of interpolation points. It 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.
Connecting cables	Connecting cables are pre-assembled or user-assembled 2-wire cables with a connector at each end. They are used to connect the -> CPU via the -> multipoint interface (MPI) to a -> programming device or to other CPUs.
Continuous-path mode	The purpose of continuous-path mode is to avoid rapid deceleration -> of the path axes at parts program block boundaries and to make the transition to the next block at as constant a velocity as possible.
Contour	Outline of a -> workpiece.
Contour monitoring	The following error is monitored within a definable tolerance band as a measure of contour accuracy. Overloading of the drive, for example, may result in an unacceptably large following error. In such cases, an alarm is output and the axes stopped.
Coordinate system	See -> machine coordinate system, -> workpiece coordinate system
CPU	Central Processor Unit -> programmable controller
Cycle	Protected subroutine for executing a recurring machining operation on the -> workpiece.
Cycles support	The available cycles are listed in menu "Cycle support" in the "Program" operating area. Once the desired machining cycle has been selected, the parameters required for assigning values are displayed in plaintext.
D	
Data block	<ol style="list-style-type: none"> 1. Data unit of the -> PLC which can be accessed by -> HIGHSTEP programs. 2. Data unit of the -> NC: Data blocks contain data definitions for global user data. These data can be initialized directly when they are defined.
Data transfer program PCIN	PCIN is a routine for transmitting and receiving CNC user data, e.g. parts programs, tool offsets, etc. via the serial interface. The PCIN program can run under MS-DOS on standard industrial PCs.

Data word	A data unit, two bytes in size, within a -> data block.
Deletion of distance-to-go	Command in parts program which stops machining and clears the remaining path distance to go.
Design	<ul style="list-style-type: none">• The SINUMERIK FM-NC is installed in the CPU tier of the SIMATIC S7-300. The 200mm wide, fully encapsulated module has the same external design as the SIMATIC S7-300 modules.• The SINUMERIK 840D is installed as a compact module in the SIMODRIVE 611D converter system. It has the same dimensions as a 50mm wide SIMODRIVE 611D module. The SINUMERIK 840D comprises the NCU module and the NCU box.• The SINUMERIK 810D has the same design as the SIMODRIVE 611D with a width of 150mm. The following components are integrated: SIMATIC S7-CPU, 5 digital servo drive controls and 3 SIMODRIVE 611D power modules.
Diagnosis	<ol style="list-style-type: none">1. Control operating area2. The control incorporates a self-diagnosis program and test routines for servicing: Status, alarm and service displays.
Digital input/output module	Digital modules are signal transducers for binary process signals.
Dimensions in metric and inch systems	Position and lead/pitch values can be programmed in inches in the machining program. The control is set to a basic system regardless of the programmable unit of measure (G70/G71).
DRF	Differential Resolver Function NC function which generates an incremental zero 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.
Drive	<ul style="list-style-type: none">• SINUMERIK FM-NC has an analog $\pm 10V$ interface to the SIMODRIVE 611A converter system.• The SINUMERIK 840D control system is linked to the SIMODRIVE 611D converter system via a high-speed digital parallel bus.
E	
Editor	The editor makes it possible to create, modify, extend, join and insert programs/texts/program blocks.

Electronic handwheel	Electronic handwheels can be used to traverse the selected axes simultaneously in manual mode. 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 continues execution at the next block.
External zero offset	A zero offset specified by the -> PLC.
F	
Fast retraction from contour	When an interrupt is received, it is possible to initiate a motion via the CNC machining program which allows the tool to be retracted quickly from the workpiece contour currently being machined. The retraction angle and the distance retracted can also be parameterized. An interrupt routine can be executed after the rapid retraction. (SINUMERIK FM-NC, 840D).
Feedforward control, dynamic	<p>Contour inaccuracies resulting from following errors can be almost completely eliminated by the dynamic, acceleration-dependent feedforward control function. Feedforward control ensures an excellent degree of machining accuracy even at high tool path velocities.</p> <p>Feedforward control can only be selected or deselected for all axes together via the parts program.</p>
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.
Finished-part contour	Contour of the finished workpiece. See also -> blank.
Fixed machine point	A point defined uniquely by the machine tool, such as the reference point.
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 on the control. Where possible, the control moves these axes in -> rapid traverse.

Frame

A frame is a calculation rule that translates one Cartesian coordinate system into another Cartesian coordinate system. A frame contains the components -> zero offset, -> rotation, -> scaling and -> mirroring.

G**General reset**

The following memories of the -> CPU are erased by a general reset operation:

- -> Working memory
- Read/write area of the -> load memory
- -> System memory
- -> Backup memory

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.

**Global main
run/subroutine**

Each global main run/subroutine can be stored only once under its name in the directory; the same program name in different directories with different contents is not possible as a global program.

Ground

"Ground" is the term applied to all the electrically inactive, interconnected parts of a piece of equipment which cannot carry any hazardous contact voltage even in the event of a fault.

H**Helical interpolation**

The helical interpolation function is ideal for machining internal and external threads using form milling cutters and for milling lubrication grooves. The helix comprises two movements:

1. Circular movement in one plane
2. Linear movement perpendicular to this plane.

**High-speed digital
inputs/outputs**

As an example, high-speed CNC program routines (interrupt routines) can be started via the digital inputs. High-speed, program-driven switching functions can be initiated via the digital CNC outputs (SINUMERIK 840D).

HIGHSTEP

Combination of the programming features for the -> PLC in the S7-300/400 range.

I

I/O module	<p>I/O modules create the link between the CPU and the process. I/O modules are:</p> <ul style="list-style-type: none">• ->Digital input/output modules• ->Analog input/output modules• ->Simulator modules
Identifier	<p>In accordance with DIN 66025, identifiers (names) for variables (arithmetic variables, system variables, user variables), for subroutines, for vocabulary words and for words can contain several address letters. These letters have the same meaning as the words in the block syntax. Identifiers must be unique. Identical identifiers must not be used for different objects.</p>
Inch system of measurement	<p>System of measurement that defines distances in "inches" and fractions thereof.</p>
Increment	<p>Travel path length specification based on number of increments. The number of increments can be stored as a -> setting data or selected with keys labeled with 10, 100, 1000, 10 000.</p>
Incremental dimension	<p>A destination for axis traversal is defined by a distance to be covered and a direction referenced to a point already reached. See also -> absolute dimension.</p>
Initialization block	<p>Initialization blocks are special -> program blocks. They contain values which must be assigned before the program is executed. Initialization blocks are used primarily for initializing predefined data or global user data.</p>
Initialization file	<p>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.</p>
Intermediate blocks	<p>Movements with selected tool offset (G41/G42) can be interrupted by a limited number of intermediate blocks (blocks without axis motions in the offset plane). When such blocks are used, the tool offset can still be calculated correctly. The permissible number of intermediate blocks read in advance by the control can be set via system parameters.</p>
Interpolative compensation	<p>Interpolative compensation provides a means of compensating for leadscrew errors (LEC) and measuring-system errors (MSEC) resulting from the production process.</p>

Interpolator

Logical unit of the -> NCK which determines intermediate values for the movements to be traversed on the individual axes on the basis of destination positions specified in the parts program.

Interrupt routine

Interrupt routines are special -> subroutines which can be started by events (external signals) in the machining process. The parts program block being processed is aborted and the axis position at the instant of interruption is stored automatically.

Inverse-time feedrate

On SINUMERIK FM-NC and 840D controls, it is possible to program the time required to traverse the path of a block instead of the feedrate speed for the axis movement (G93).

J**Jog**

Control system operating mode (setup): The machine can be set up in Jog mode. Individual axes and spindles can be jogged by means of direction keys. Other functions in Jog mode are -> reference point approach, -> Repos and -> Preset -> (set actual value).

K**Keyswitch**

1. **S7-300**: The keyswitch is the mode selector switch on the -> CPU. The keyswitch is operated by means of a removable key.
2. **840D/FM-NC**: The keyswitch on the -> machine control panel has 4 positions which are assigned functions by the operating system of the control. There are also three keys of different colors belonging to the keyswitch that can be removed in the specified positions.

K_Ü

Transmission Ratio

K_v

Servo gain factor, control variable of a control loop

L**Languages**

The user interface texts, system messages and alarms are available in five system languages (floppy disk):

English, French, German, Italian and Spanish.

The user can select **two** of the listed languages at a time in the control.

Leadscrew error compensation

Compensation of mechanical inaccuracies in a leadscrew involved in the feed motion. Errors are compensated by the control based on stored deviation measurements.

Limit speed	Minimum/maximum (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 travels along a straight line to the destination point while machining the workpiece.
Load memory	On the CPU 314 of the -> PLC, the load memory is identical with the -> working memory.
Look Ahead	The Look Ahead function is a means of optimizing the machining velocity by looking ahead over a parameterizable number of traversing blocks.
Look Ahead for contour violations	The control detects and reports the following types of collision: 1. Path is shorter than tool radius. 2. Width of inside corner is less than the tool diameter.
M	
Machine	Control operating area
Machine axes	Axes which exist physically on the machine tool.
Machine control panel	An operator panel on a machine tool with operating elements such as keys, rotary switches, etc. and simple indicators such as LEDs. It is used for direct control of the machine tool via the PLC.
Machine coordinate system	System of coordinates based on the axes of the machine tool.
Machine zero	A fixed point on the machine tool which can be referenced by all (derived) measurement systems.
Machining channel	A channel structure makes it possible to reduce downtimes by allowing sequences of motions to be executed in parallel. For example, a loading gantry can execute its movements during a machining operation. In this case, a CNC channel ranks as an autonomous CNC control complete with decoding, block preparation and interpolation.

Macros	A set of instructions grouped under one identifier. In the program, the identifier represents the set of grouped instructions.
Main block	A block prefixed by ":" containing all the parameters required to start execution of a -> parts program.
Main program	-> Parts program identified by a number or name in which other main programs, subroutines or -> cycles may be called.
MDA	Control system operating mode: Manual Data Automatic. In the MDA mode, individual program blocks or block sequences with no reference to a main program or subroutine can be input and executed immediately afterwards through actuation of the NC Start key.
Measuring circuits	<ul style="list-style-type: none">• SINUMERIK FM-NC: The requisite control circuits for axes and spindles are integrated in the control module as standard. A maximum total of 4 axes and spindles can be implemented, with no more than 2 spindles.• SINUMERIK 840D: The signals from the sensors are analyzed in the SIMODRIVE 611D drive modules. The maximum configuration is a total of 8 axes and spindles, with up to 5 spindles being permissible.
Messages	All messages programmed in the parts program and -> alarms detected by the system are displayed in plain text with date and time and the corresponding symbol for the delete criterion on the operator panel. Alarms and messages are displayed separately.
Metric system	Standardized system of units for lengths in millimeters (mm), meters (m), etc.
Mirroring	Mirroring exchanges the leading signs of the coordinate values of a contour in relation to an axis. Analogously, several axes can be mirrored simultaneously.
Mode	An operating concept on a SINUMERIK control. The modes -> Jog, -> MDA, -> Automatic are defined.
Mode group	All axes/spindles are assigned to one and only one channel at any given time. Each channel is assigned to a mode group The same -> mode is always assigned to the channels of a mode group.

Multipoint interface

The multipoint interface (MPI) is a 9-pin sub-D port. A parameterizable number of devices can be connected to an MPI for the purpose of communicating with one another:

- Programming devices
- MMI (HMI) systems
- Other automation systems

The "Multipoint Interface MPI" parameter block of the CPU contains the -> parameters which define the properties of the multipoint interface.

N**NC**

Numerical Control It incorporates all the components of the machine tool control system: -> NCK, -> PLC, -> MMC, -> COM.

Note: CNC (computerized numerical control) would be a more appropriate description for the SINUMERIK 840D control or FM-NC : computerized numerical control.

NCK

Numerical Control Kernel: Component of the NC control which executes -> parts programs and essentially coordinates the movements on the machine tool.

Network

A network is the interconnection of several S7-300s and other terminal devices such as a programming device, for example, interlinked by means of -> connecting cables. The networked devices interchange data via the network.

Node number

The node number is the "contact address" of a -> CPU or the -> programming device or another intelligent I/O module if these devices are exchanging data with one another via a -> network. The node number is assigned to the CPU or the programming device by the S7 tool -> "S7 Configuration".

NRK

Numeric Robotic Kernel (operating system of the -> NCK)

NURBS

Motion control and path interpolation are implemented internally in the control on the basis of NURBS (Non-Uniform Rational B Splines). A standard procedure is thus available (SINUMERIK 840D) as an internal control function for all modes of interpolation.

O**Oblique-plane machining**

Drilling and milling operations on workpiece surfaces which are oblique to the coordinate planes of the machine are supported by the "Oblique surface machining" function.

OEM	The scope for implementing individual solutions (OEM applications) for the SINUMERIK 840D has been provided for machine manufacturers who wish to create their own operator interface or integrate process-oriented functions in the control.
Offset memory	Data area in the control in which tool offset data are stored.
Operator interface	The operator interface (OI) is the human-machine interface of a CNC. It takes the form of a screen and has eight horizontal and eight vertical softkeys.
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 retraction	RETTOOL: If machining is interrupted (e.g. when a tool breaks), a program command can be used to retract the tool in a user-specified orientation by a defined distance.
Override	Manual or programmable control feature which enables the user to override programmed feedrates or speeds in order to adapt them to a specific workpiece or material.
P	
Parameters	<ol style="list-style-type: none"> S7-300: The S7-300 uses two types of parameter: <ul style="list-style-type: none"> Parameter of a STEP7 statement A parameter of a STEP7 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"> Control operating area Computation parameter, can be set any number of times or queried 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 certain machining operations on a given -> blank.
Parts program management	The parts program management function can be organized according to -> workpieces. The size of the user memory determines the number of programs and data to be managed. Each file (programs and data) can be given a name consisting of a maximum of 16 alphanumeric characters.

Path axis	Path axes are all the machining axes in the -> channel which are controlled by the -> interpolator such that they start, accelerate, stop and reach their end positions simultaneously.
Path feed	The path feed acts on -> path axes. It represents the geometrical sum of the feeds on the participating -> geometry axes.
Path velocity	The maximum programmable path velocity depends on the input resolution. With a resolution of 0.1mm, for example, the maximum programmable path velocity is 1000 m/min.
PG	Programming Device
PLC	Programmable Logic Control Component of the -> NC: Programmable controller for processing the control logic on the machine tool.
PLC program memory	<ul style="list-style-type: none">• SINUMERIK Fm-NC: The PLC user program of CPU 314, the user data and the basic PLC program are stored together in the PLC user memory. A user memory of 24 KB is available on the S7-CPU314.• SINUMERIK 840D: The PLC user program, the user data and the basic PLC program are stored together in the PLC user memory. The PLC user memory can be expanded to up to 96 KB.
PLC programming	The PLC is programmed with the STEP7 software. The STEP 7 programming software is based on the standard WINDOWS operating system and incorporates the functionality of STEP5 programming with innovative expansions and developments.
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.
Polynomial interpolation	Polynomial interpolation provides a means of generating a very wide range of curves, including straight-line, parabolic and exponential functions (SINUMERIK 840D).
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.

Power ON	The action of switching the control off and then on again.
Preset	The control zero point can be redefined in the machine coordinate system by means of the Preset function. Preset does not cause the axes to move; instead, a new position value is entered for the current axis positions.
Program	<ol style="list-style-type: none"> 1. Control operating area 2. Sequence of instructions to the control system.
Program block	Program blocks contain the main programs and subroutines of the -> parts programs.
Programmable frames	Programmable -> frames can be used to define new coordinate system starting points dynamically while the parts program is running. A distinction is made between absolute definition using a new frame and additive definition with reference to an existing starting point.
Programmable logic controller	Programmable logic controllers (PLC) are electronic controllers whose functions are stored as a program in the control unit. The design and wiring of the unit are not, therefore, dependent on the control functions. Programmable logic controllers have the same structure as a computer, i.e. they consist of a CPU with memory, input/output modules and an internal bus system. The I/Os and programming language are selected according to the requirements of the control technology involved.
Programmable working area limitation	Limitation of the movement area of the tool to within defined, programmable limits.
Programming key	Characters and character sequences which have a defined meaning in the programming language for -> parts programs (see Programming Guide).
Protection zone	Three-dimensional area within a -> working area which the tool tip is not permitted to enter.
Q	
Quadrant error compensation	Contour errors on quadrant transitions caused by frictional fluctuations on guideways can be largely eliminated by means of quadrant error compensation. A circularity test is performed to parameterize the quadrant error compensation function.

R**R parameter**

Calculation parameter. The programmer can assign or request the values of the R parameter in the -> parts program as required (R variable).

Rail

This rail is used to mount the modules of the S7-300 system.

Rapid traverse

The highest traversing 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

Point on the machine tool with which the measuring system of the -> machine axes is referenced.

Reference point approach

If the position measuring system used is not an absolute-value encoder, then a reference point approach operation is required to ensure that the actual values supplied by the measuring system are in accordance with the machine coordinate values.

Remanence

Data areas in data blocks, such as timers, counters and flags are remanent if their contents are not lost in the case of a restart or Power Off.

REPOS

1. Reapproach contour, triggered by operator
REPOS allows the tool to be returned to the interrupt position by means of the direction keys.
2. Programmed contour reapproach
A selection of approach strategies are available in the form of program commands: Approach point of interruption, approach start of block, approach end of block, approach a point on the path between start of block and interruption.

Rigid tapping

This function is used to tap holes without the use of a compensating chuck. The spindle is controlled as an interpolative rotary axis and drill axis, with the result that threads are tapped precisely to the final drilling depth, for example, in blind tapped holes (precondition: Spindle 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 set to smaller 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, 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 on the -> programming device. These parameters are uploaded to the CPU.
S7-300 bus	The S7-300 bus is a serial data bus which supplies modules with the appropriate voltage and via which they exchange data with one another. The connection between the modules is made by means of -> bus connectors.
Safety functions	The control includes continuously active monitoring functions which detect faults in the -> CNC, the programmable controller (-> PLC) and the machine so early that damage to the workpiece, tool or machine rarely occurs. In the event of a fault, the machining operation is interrupted and the drives stopped. The cause of the malfunction is logged and an alarm issued. At the same time, the PLC is notified that a CNC alarm is pending.
Scaling	Component of a -> frame which causes axis-specific scale alterations.
Serial RS-232 interface	For data input / output, one serial RS-232 interface is available on the MMC module MMC100, and two serial RS-232 interfaces on the MMC modules MMC101 and MMC102. Machining programs and manufacturer and user data can be imported and exported via these interfaces.
Services	Control operating area
Setting data	Data which provide the control with information about properties of the machine tool in a way defined by the system software.
Simulator module	A simulator module is a module, <ul style="list-style-type: none">• on which, via control elements, digital input values can be simulated and• digital output values are displayed.

Softkey	A key whose name appears on an area of the screen. The choice of softkeys displayed is adapted dynamically to the operating situation. The freely assignable function keys (softkeys) are assigned to functions defined in the software.
Software limit switches	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	<p>The spindle functionality is a two-level construct:</p> <ol style="list-style-type: none">1. Spindles: Speed or position-controlled spindle drives analog $\pm 10V$ (SINUMERIK FM-NC) digital (SINUMERIK 840D)2. Auxiliary spindles: Speed-controlled spindle drives "auxiliary spindle" function package, e.g. for power tools.
Spline interpolation	Using the spline interpolation function, the control is able to generate a smooth curve from just a small number of specified interpolation points along a setpoint contour.
Standard cycles	<p>Standard cycles are used to program machining operations which repeat frequently:</p> <ul style="list-style-type: none">• For drilling/milling• For turning (SINUMERIK FM-NC) <p>The available cycles are listed in menu "Cycle support" in the "Program" operating area. Once the desired machining cycle has been selected, the parameters required for assigning values are displayed in plaintext.</p>
Subblock	Block prefixed by "N" containing information for a machining step such as a position parameter.
Subroutine	A sequence of instructions of a -> parts program which can be called repeatedly with different initial parameters. A subroutine is called from within a main program. Each subroutine can be blocked against unauthorized output. -> Cycles are a type of subroutine.
Synchronization	Instructions in -> parts programs for coordination of the operations in different -> channels at specific machining points.

Synchronized actions

1. Auxiliary function output
While a workpiece is being machined, technological functions (-> auxiliary functions) can be output from the CNC program to the PLC. These auxiliary functions control, for example, ancillary equipment on the machine tool such as the sleeve, gripper, chuck, etc.
2. High-speed auxiliary function output
The acknowledgement times for the -> auxiliary functions can be minimized and unnecessary halts in the machining process avoided for time-critical switching functions.

Synchronized axes

Synchronized axes require the same amount of time to traverse their path as -> geometry axes for their path.

System memory

The system memory is a memory in the CPU, where the following data is stored:

- Data required by the operating system
- The following operands: Timers, counters, flags

System variable

A variable which exists although it has not been programmed by the -> parts program programmer. 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 can be input via the keyboard and executed immediately.

Positions approached via the direction keys or handwheel can also be stored.

Additional information such as G functions, feedrates or M functions can be entered in the same block.

Text editor

-> Editor

Tool

Software tool for the entry and editing of -> parameters of a parameter block. Tools include:

- -> S7 Configuration
- S7 TOP
- S7 Info

Tool

A tool employed to shape the workpiece, for example, a turning tool, milling cutter, drill, laser beam, etc.

Tool nose radius compensation

A contour is programmed on the assumption that a pointed tool will be used. Since this is not always the case in practice, the curvature radius of the tool being used is specified so that the control can make allowance for it. The curvature center point is guided equidistantly 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 in the control is set in parameterization.

Tool radius compensation

In order to program a desired -> workpiece contour directly, the control must traverse a path equidistant to the programmed contour, taking into account the radius of the tool used (G41/G42).

Transformation

Programming in a Cartesian coordinate system, execution in a non-Cartesian coordinate system (e.g. with machine axes as rotary axes).

Traversing range

The maximum permissible travel range for linear axes is ± 9 decades. The absolute value depends on the selected input and position control resolution and the unit of measurement (inch or metric).

U**User memory**

All programs and data such as parts programs, subroutines, comments, tool offsets, zero offsets/frames and channel and program user data can be stored in the common CNC user memory.

User program

User programs for S7-300 automation systems are created using the STEP 7 programming language. The user program has a modular structure and is comprised of individual blocks.

The basic block types are:

Code blocks: Contain the STEP 7 commands.

Data blocks: Contain constants and variables for the STEP 7 program.

User-defined variable

Users can define variables in the -> parts program or data block (global user data) for their own use. A definition contains a data type specification and the variable name. See also -> system variable.

V**Variable definition**

A variable is defined through 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 -> look ahead.

Vocabulary words

Words with a specific notation which have a defined meaning in the programming language for -> parts programs.

W**Work offset**

-> Zero offset

Working memory

The working storage is a Random Access Memory in the -> CPU which the processor accesses as it executes the application program.

Working space

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.

Workpiece

Part to be produced/machined by the machine tool.

Workpiece contour

Setpoint contour of the -> workpiece to be produced/machined.

Workpiece coordinate system

The origin of the workpiece coordinate system is the -> workpiece zero. In machining operations programmed in the workpiece coordinate system, the dimensions and directions refer to this system.

Workpiece zero

The workpiece zero forms the starting point for the -> workpiece coordinate system. It is defined by distances to the machine zero.

X**Y**

Z

Zero offset

Specification of a new reference point for a coordinate system through reference to an existing zero and a -> frame.

-> Work offset

1. Settable

SINUMERIK 840D: A parameterizable number of settable zero offsets is available for each CNC axis. The offsets - which are selected by means of G functions - take effect alternately.

2. External

All offsets which define the position of the workpiece zero can be overlaid with an external zero offset

- defined by handwheel (DRF offset) or
- defined by the PLC.

3. Programmable

Zero offsets can be programmed for all path and positioning axes by means of the TRANS instruction.

C References

General Documentation

- /BU/** SINUMERIK & SIMODRIVE, Automation Systems for Machine Tools
Catalog NC 60
Order No.: E86060-K4460-A101-B1-7600
- /IKPI/** Industrial Communication and Field Devices
Catalog IK PI
Order No.: E86060-K6710-A101-B2-7600
- /ST7/** SIMATIC
Products for Totally Integrated Automation and Micro Automation
Catalog ST 70
Order No.: E86060-K4670-A111-A8-7600
- /Z/** MOTION-CONNECT
Cable, Connectors & System Components for SIMATIC, SINUMERIK,
MASTERDRIVES and SIMOTION
Catalog NC Z
Order No.: E86060-K4490-A001-B1-7600
- Safety Integrated
Application Manual
The safety program for industries of the world
Order No: 6ZB5000-0AA02-0BA0

Electronic Documentation

- /CD1/** The SINUMERIK System (03.04 Edition)
DOC ON CD
(includes all SINUMERIK 840D/840Di/810D/802 and
SIMODRIVE publications)
Order No.: 6FC5298-7CA00-0BG0

User Documentation

/AUK/	SINUMERIK 840D/810D Short Guide AutoTurn Operation Order No.: 6FC5298-4AA30-0BP2	(09.99 Edition)
/AUP/	SINUMERIK 840D/810D Operator's Guide AutoTurn Graphic Programming System Programming / Setup Order No.: 6FC5298-4AA40-0BP3	(02.02 Edition)
/BA/	SINUMERIK 840D/810D Operator's Guide MMC Order No.: 6FC5298-6AA00-0BP0	(10.00 Edition)
/BAD/	SINUMERIK 840D/840Di/810D Operator's Guide HMI Advanced Order No.: 6FC5298-6AF00-0BP3	(03.04 Edition)
/BAH/	SINUMERIK 840D/840Di/810D Operator's Guide HT 6 Order No.: 6FC5298-0AD60-0BP3	(03.04 Edition)
/BAK/	SINUMERIK 840D/840Di/810D Short Guide Operation Order No.: 6FC5298-6AA10-0BP0	(02.01 Edition)
/BAM/	SINUMERIK 810D/840D Operation/Programming ManualTurn Order No.: 6FC5298-6AD00-0BP0	(08.02 Edition)
/BAS/	SINUMERIK 840D/840Di/810D Operation/Programming ShopMill Order No.: 6FC5298-6AD10-0BP2	(11.03 Edition)
/BAT/	SINUMERIK 840D/810D Operation/Programming ShopTurn Order No.: 6FC5298-6AD50-0BP2	(06.03 Edition)
/BEM/	SINUMERIK 840D/810D Operator's Guide HMI Embedded Order No.: 6FC5298-6AC00-0BP3	(03.04 Edition)
/BNM/	SINUMERIK 840D/840Di/810D User's Guide Measuring Cycles Order No.: 6FC5298-6AA70-0BP3	(03.04 Edition)
/BTDI/	SINUMERIK 840D/840Di/810D Motion Control Information System (MCIS) User's Guide Tool Data Information Order No.: 6FC5297-6AE01-0BP0	(04.03 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.: 6FC5298-7AA20-0BP0	(03.04 Edition)

/KAM/	SINUMERIK 840D/810D Short Guide ManualTurn Order No.: 6FC5298-5AD40-0BP0	(04.01 Edition)
/KAS/	SINUMERIK 840D/810D Short Guide ShopMill Order No.: 6FC5298-5AD30-0BP0	(04.01 Edition)
/KAT/	SINUMERIK 840D/810D Short Guide ShopTurn Order No.: 6FC5298-6AF20-0BP0	(07.01 Edition)
/PG/	SINUMERIK 840D/840Di/810D Programming Guide Fundamentals Order No.: 6FC5298-7AB00-0BP0	(03.04 Edition)
/PGA/	SINUMERIK 840D/840Di/810D Programming Guide Advanced Order No.: 6FC5298-7AB10-0BP0	(03.04 Edition)
/PGK/	SINUMERIK 840D/840Di/810D Short Guide Programming Order No.: 6FC5298-7AB30-0BP0	(03.04 Edition)
/PGM/	SINUMERIK 840D/840Di/810D Programming Guide ISO Milling Order No.: 6FC5298-6AC20-0BP2	(11.02 Edition)
/PGT/	SINUMERIK 840D/840Di/810D Programming Guide ISO Turning Order No.: 6FC5298-6AC10-0BP2	(11.02 Edition)
/PGZ/	SINUMERIK 840D/840Di/810D Programming Guide Cycles Order No.: 6FC5298-7AB40-0BP0	(03.04 Edition)
/PI/	PCIN 4.4 Software for Data Transfer to/from MMC Module Order No.: 6FX2060-4AA00-4XB0 (English, French, German) Order from: WK Fürth	
/SYI/	SINUMERIK 840Di System Overview Order No.: 6FC5298-6AE40-0BP0	(02.01 Edition)

Manufacturer/Service Documentation**a) Lists****/LIS/**

SINUMERIK 840D/840Di/810D (03.04 Edition)
SIMODRIVE 611D

Lists

Order No.: 6FC5297-7AB70-0BP0

b) Hardware**/ASAL/**

SIMODRIVE 611, MASTERDRIVES VC/MC (10.03 Edition)
Planning Guide General Information for **Asynchronous Servomotors**
Order No.: 6SN1197-0AC62-0BP0

/APH2/

SIMODRIVE 611 (10.03 Edition)
Planning Guide **1PH2 Asynchronous Servomotors**
Order No.: 6SN1197-0AC63-0BP0

/APH4/

SIMODRIVE 611 (10.03 Edition)
Planning Guide **1PH4 Asynchronous Servomotors**
Order No.: 6SN1197-0AC64-0BP0

/APH7S/

SIMODRIVE 611 (03.04 Edition)
Planning Guide **1PH7 Asynchronous Servomotors**
Order No.: 6SN1197-0AC66-0BP0

/APH7M/

MASTERDRIVES MC (04.04 Edition)
Planning Guide **1PH7 Asynchronous Servomotors**
Order No.: 6SN1197-0AC65-0BP0

/APL6/

MASTERDRIVES VC/MC (03.04 Edition)
Planning Guide **1PL6 Asynchronous Servomotors**
Order No.: 6SN1197-0AC67-0BP0

/BH/

SINUMERIK 840D/840Di/810D (11.03 Edition)
Operator Components Manual
Order No.: 6FC5297-6AA50-0BP3

/BHA/

SIMODRIVE Sensor (03.03 Edition)
User's Guide (HW) **Absolute Position Sensor with Profibus-DP**
Order No.: 6SN1197-0AB10-0YP2

/EMV/

SINUMERIK, SIROTEC, SIMODRIVE (06.99 Edition)
Planning Guide **EMC Installation Guide**
Order No.: 6FC5297-0AD30-0BP1

The current Declaration of Conformity is available under the following Internet address:

<http://www4.ad.siemens.de>

Please enter the ID No.: 15257461 in the "Search" field (top right) and click on "go".

/GHA/

SINUMERIK/SIMOTION (02.03 Edition)
ADI4 – Analog Drive Interface for 4 Axes
Manual
Order No.: 6FC5297-0BA01-0BP1

/PFK6/

SIMODRIVE, MASTERDRIVES MC (05.03 Edition)
Planning Guide **1FK6 Synchronous Servomotors**
Order No.: 6SN1197-0AD05-0BP0

/PFK7/	SIMODRIVE 611, MASTERDRIVES MC Planning Guide 1FK7 Synchronous Servomotors Order No.: 6SN1197-0AD06-0BP0	(01.03 Edition)
/PFS6/	MASTERDRIVES MC Planning Guide 1FS6 Synchronous Servomotors Order No.: 6SN1197-0AD08-0BP0	(07.03 Edition)
/PFT5/	SIMODRIVE 611 Planning Guide 1FT5 Synchronous Servomotors Order No.: 6SN1197-0AD01-0BP0	(05.03 Edition)
/PFT6/	SIMODRIVE 611, MASTERDRIVES MC Planning Guide 1FT6 Synchronous Servomotors Order No.: 6SN1197-0AD02-0BP0	(01.04 Edition)
/PFU/	SINAMICS, MASTERDRIVES MICROMASTER SIEMOSYN Motors 1FU8 Order No.: 6SN1197-0AC80-0BP0	(09.03 Edition)
/PHC/	SINUMERIK 810D Configuring Manual CCU (HW) Order No.: 6FC5297-6AD10-0BP1	(11.02 Edition)
/PHD/	SINUMERIK 840D Configuring Manual NCU (HW) Order No.: 6FC5297-6AC10-0BP3	(11.03 Edition)
/PJAL/	SIMODRIVE 611, MASTERDRIVES MC Planning Guide Synchronous Servomotors General Part for 1FT/1FK Motors Order No.: 6SN1197-0AD07-0BP1	(01.04 Edition)
/PJAS/	SIMODRIVE 611, MASTERDRIVES VC/MC Planning Guide for Asynchronous Motors Content: General Part, 1PH2, 1PH4, 1PH7, 1PL6 Order No.: 6SN1197-0AC61-0BP0	(06.04 Edition)
/PJFE/	SIMODRIVE Planning Guide 1FE1 Built-In Synchronous Motors Three-Phase AC Motors for Main Spindle Drives Order No.: 6SN1197-0AC00-0BP5	(03.04 Edition)
/PJF1/	SIMODRIVE Installation Guide 1FE1 051.-1FE1 147. Built-In Synchronous Motors Three-Phase AC Motors for Main Spindle Drives Order No.: 610.43000.02	(12.02 Edition)
/PJLM/	SIMODRIVE Planning Guide 1FN1, 1FN3 Linear Motors ALL General Information about Linear Motors 1FN1 1FN1 Three-Phase AC Linear Motor 1FN3 1FN3 Three-Phase AC Linear Motor CON Connections Order No.: 6SN1197-0AB70-0BP4	(06.02 Edition)

/PJM2/	SIMODRIVE 611, MASTERDRIVES MC Planning Guide for Synchronous Servomotors Content: General Part, 1FT5, 1FT6, 1FK6, 1FK7, 1FS6 Order No.: 6SN1197-0AC20-0BP0	(03.04 Edition)
/PJTM/	SIMODRIVE Planning Guide 1FW6 Integrated Torque Motors Order No.: 6SN1197-0AD00-0BP1	(05.03 Edition)
/PJU/	SIMODRIVE 611 Planning Guide Inverters Order No.: 6SN1197-0AA00-0BP6	(02.03 Edition)
/PKTM/	MASTERDRIVES Planning Guide 1FW3 Complete Torque Motors Order No.: 6SN1197-0AC70-0BP0	(03.04 Edition)
/PMH/	SIMODRIVE Sensor Configuring/Installation Guide Hollow-Shaft Measuring System SIMAG H Order No.: 6SN1197-0AB30-0BP1	(07.02 Edition)
/PMH2/	SIMODRIVE Sensor Configuring/Installation Guide Hollow-Shaft Measuring System SIMAG H2 Order No.: 6SN1197-0AB31-0BP1	(03.04 Edition)
/PMHS/	SIMODRIVE Installation Guide Measuring System for Main Spindle Drives SIZAG2 Toothed-Wheel Encoder Order No.: 6SN1197-0AB00-0YP3	(12.00 Edition)
/PMS/	SIMODRIVE Planning Guide 2SP1 ECO Motor Spindle for Main Spindle Drives Order No.: 6SN1197-0AD04-0BP1	(03.04 Edition)
/PPH/	SIMODRIVE Planning Guide 1PH2, 1PH4, 1PH7 Motors AC Induction Motors for Main Spindle Drives Order No.: 6SN1197-0AC60-0BP0	(12.01 Edition)
/PPM/	SIMODRIVE Planning Guide Hollow-Shaft Motors for 1PM4 and 1PM6 Main Spindle Drives Order No.: 6SN1197-0AD03-0BP0	(11.01 Edition)

c) Software

/FB1/

SINUMERIK 840D/840Di/810D/FM-NC

(03.04 Edition)

Description of Functions **Basic Machine (Part 1)**

(the various sections are listed below)

Order No.: 6FC5297-7AC20-0BP0

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, Channel, 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 Offset

/FB2/

SINUMERIK 840D/840Di/810D

(03.04 Edition)

Description of Functions **Extended Functions (Part 2)**

including FM-NC: Turning, Stepper Motor

(the various sections are listed below)

Order No.: 6FC5297-7AC30-0BP0

A4	Digital and Analog NCK I/Os
B3	Several Operator Panels and NCUs
B4	Operation via PG/PC
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	Measurement
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/higher/FBSY/)
S6	Stepper Motor Control
S7	Memory Configuration
T1	Indexing Axes
W3	Tool Change
W4	Grinding

/FB3/	SINUMERIK 840D/840Di/810D	(03.04 Edition)
	Description of Functions Special Functions (Part 3)	
	(the various sections are listed below)	
	Order No.: 6FC5297-7AC80-0BP0	
	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	
	TE0 Installation and Activation of Compile Cycles	
	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	(03.04 Edition)
	Description of Functions Drive Functions	
	(the various sections are listed below)	
	Order No.: 6SN1197-0AA80-1BP1	
	DB1 Operation 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	(02.00 Edition)
	Description of Functions ANA MODULE	
	Order No.: 6SN1197-0AB80-0BP0	
/FBD/	SINUMERIK 840D	(07.99 Edition)
	Description of Functions Digitizing	
	Order No.: 6FC5297-4AC50-0BP0	
	DI1 Start-up	
	DI2 Scanning with Tactile Sensors (scancad scan)	
	DI3 Scanning with Lasers (scancad laser)	
	DI4 Milling Program Generation (scancad mill)	
/FBDM/	SINUMERIK 840D/840Di/810D	(09.03 Edition)
	Description of Functions NC Program Management	
	DNC Machines	
	Order No.: 6FC5297-1AE81-0BP0	

/FBDN/	SINUMERIK 840D/840Di/810D	(03.03 Edition)
	Motion Control Information System (MCIS)	
	Description of Functions DNC NC Program Management	
	Order No.: 6FC5297-1AE80-0BP0	
	DN1 DNC Plant/DNC Cell DN2 DNC IFC SINUMERIK	
/FBFA/	SINUMERIK 840D/840Di/810D	(11.02 Edition)
	Description of Functions ISO Dialects for SINUMERIK	
	Order No.: 6FC5297-6AE10-0BP3	
/FBFE/	SINUMERIK 840D/810D	(03.04 Edition)
	Motion Control Information System (MCIS)	
	Description of Functions Remote Diagnosis	
	Order No.: 6FC5297-0AF00-0BP3	
	FE1 Remote Diagnosis (ReachOut) FE3 Remote Diagnosis (pcAnywhere)	
/FBH/	SINUMERIK 840D/840Di/810D	(11.02 Edition)
	HMI Configuration Package	
	Order No.: (supplied with the software)	
	Part 1 User's Guide Part 2 Description of Functions	
/FBH1/	SINUMERIK 840D/840Di/810D	(03.03 Edition)
	HMI Configuration Package	
	ProTool/Pro Option SINUMERIK	
	Order No.: (supplied with the software)	
/FBHL/	SINUMERIK 840D/SIMODRIVE 611 digital	(10.03 Edition)
	Description of Functions HLA Module	
	Order No.: 6SN1197-0AB60-0BP3	
/FBIC/	SINUMERIK 840D/840Di/810D	(06.03 Edition)
	Motion Control Information System (MCIS)	
	Description of Functions TDI Ident Connection	
	Order No.: 6FC5297-1AE60-0BP0	
/FBMA/	SINUMERIK 840D/810D	(08.02 Edition)
	Description of Functions ManualTurn	
	Order No.: 6FC5297-6AD50-0BP0	
/FBO/	SINUMERIK 840D/810D	(09.01 Edition)
	Description of Functions Configuring OP 030 Operator Interface	
	Order No.: 6FC5297-6AC40-0BP0	
	BA Operator's Guide	
	EU Development Environment (Configuring Package)	
	PSE Introduction to Configuring of Operator Interface	
	(IK Screen Kit: Software Update and Configuration)	
/FBP/	SINUMERIK 840D	(03.96 Edition)
	Description of Functions C-PLC-Programming	
	Order No.: 6FC5297-3AB60-0BP0	

/FBR/	SINUMERIK 840D/840Di/810D Description of Functions RPC SINUMERIK Computer Link Order No.: 6FC5297-6AD60-0BP0 NFL Host Computer Interface NPL PLC/NCK Interface	(01.04 Edition)
/FBSI/	SINUMERIK 840D/SIMODRIVE Description of Functions SINUMERIK Safety Integrated Order No.: 6FC5297-6AB80-0BP2	(11.03 Edition)
/FBSP/	SINUMERIK 840D/840Di/810D Description of Functions ShopMill Order No.: 6FC5297-6AD80-0BP2	(11.03 Edition)
/FBST/	SIMATIC Description of Functions FM STEPDRIVE/SIMOSTEP Order No.: 6SN1197-0AA70-0YP4	(01.01 Edition)
/FBSY/	SINUMERIK 840D/810D Description of Functions Synchronized Actions Order No.: 6FC5297-6AD40-0BP2	(03.04 Edition)
/FBT/	SINUMERIK 840D/810D Description of Functions ShopTurn Order No.: 6FC5297-6AD70-0BP2	(03.04 Edition)
/FBTC/	SINUMERIK 840D/810D IT Solutions Description of Functions Tool Data Communication SinTDC Order No.: 6FC5297-5AF30-0BP0	(01.02 Edition)
/FBTD/	SINUMERIK 840D/810D IT Solutions Description of Functions Tool Information System (SinTDI) with Online Help Order No.: 6FC5297-6AE00-0BP0	(02.01 Edition)
/FBTP/	SINUMERIK 840D/840Di/810D Motion Control Information System (MCIS) Description of Functions TPM Total Productive Maintenance Order No.: Document is supplied with the software	(01.03 Edition)
/FBU/	SIMODRIVE 611 universal/universal E Closed-Loop Control Component for Speed Control and Positioning Description of Functions Order No.: 6SN1197-0AB20-0BP8	(07.03 Edition)
/FBU2/	SIMODRIVE 611 universal Installation Guide (enclosed with SIMODRIVE 611 universal)	(04.02 Edition)
/FBW/	SINUMERIK 840D/810D Description of Functions Tool Management Order No.: 6FC5297-6AC60-0BP1	(11.02 Edition)
/HBA/	SINUMERIK 840D/840Di/810D Manual @Event Order No.: 6AU1900-0CL20-0BA0	(03.02 Edition)

/HBI/	SINUMERIK 840Di Manual SINUMERIK 840Di Order No.: 6FC5297-6AE60-0BP2	(09.03 Edition)
/INC/	SINUMERIK 840D/840Di/810D System Description Commissioning Tool SINUMERIK SinuCOM NC Order No.: (an integral part of the online help for the start-up tool)	(06.03 Edition)
/PGA1/	SINUMERIK 840D/840Di/810D List Manual System Variables Order No.: 6FC5297-7AE10-0BP0	(03.04 Edition)
/PJE/	SINUMERIK 840D/810D Description of Functions Configuring Package HMI Embedded Software Update, Configuration Installation Order No.: 6FC5297-6EA10-0BP0	(08.01 Edition)
/PS/	SINUMERIK 840D/810D Planning Guide Configuring Syntax This publication is included with the software and available as a pdf file	(09.03 Edition)
/POS1/	SIMODRIVE POSMO A User's Guide Distributed Positioning Motor on PROFIBUS DP Order No.: 6SN2197-0AA00-0BP6	(08.03 Edition)
/POS2/	SIMODRIVE POSMO A Installation Guide (enclosed with POSMO A)	(05.03 Edition)
/POS3/	SIMODRIVE POSMO SI/CD/CA User's Guide Distributed Servo Drive Systems Order No.: 6SN2197-0AA20-0BP5	(07.03 Edition)
/POS4/	SIMODRIVE POSMO SI Installation Guide (enclosed with POSMO SI)	(04.02 Edition)
/POS5/	SIMODRIVE POSMO CD/CA Installation Guide (enclosed with POSMO CD/CA)	(04.02 Edition)

/S7H/	SIMATIC S7-300 Installation Manual Technological Functions Order No.: 6ES7398-8AA03-8BA0 - Reference Manual: CPU Data (Hardware) - Reference Manual: Module Data	(2002 Edition)
/S7HT/	SIMATIC S7-300 Manual: STEP 7, Fundamentals, V. 3.1 Order No.: 6ES7810-4CA02-8BA0	(03.97 Edition)
/S7HR/	SIMATIC S7-300 Manual: STEP 7, Reference Manuals, V. 3.1 Order No.: 6ES7810-4CA02-8BR0	(03.97 Edition)
/S7S/	SIMATIC S7-300 FM 353 Positioning Module for Stepper Drive Order together with configuring package	(04.02 Edition)
/S7L/	SIMATIC S7-300 FM 354 Positioning Module for Servo Drive Order together with configuring package	(04.02 Edition)
/S7M/	SIMATIC S7-300 FM 357.2 Multimodule for Servo and Stepper Drives Order together with configuring package	(01.03 Edition)
/SP/	SIMODRIVE 611-A/611-D SimoPro 3.1 Program for Configuring Machine-Tool Drives Order No.: 6SC6111-6PC00-0BA□ Order from: WK Fürth	

d) Installation and Start-up

/BS/	SIMODRIVE 611 analog Description Start-Up Software for Main Spindle and Asynchronous Motor Modules Version 3.20 Order No.: 6SN1197-0AA30-0BP1	(10.00 Edition)
/IAA/	SIMODRIVE 611A Installation and Start-Up Guide Order No.: 6SN1197-0AA60-0BP6	(10.00 Edition)
/IAC/	SINUMERIK 810D Installation and Start-Up Guide (incl. description of SIMODRIVE 611D start-up software) Order No.: 6FC5297-6AD20-0BP1	(11.02 Edition)
/IAD/	SINUMERIK 840D/SIMODRIVE 611D Installation and Start-Up Guide (incl. description of SIMODRIVE 611D digital start-up software SIMODRIVE 611D) Order No.: 6FC5297-7AB10-0BP0	(03.04 Edition)
/IAM/	SINUMERIK 840D/840Di/810D Installation and Start-Up Guide HMI Order No.: 6FC5297-6AE20-0BP3 AE1 Updates/Supplements BE1 Expanding the Operator Interface HE1 Online Help IM2 Starting up HMI Embedded IM4 Starting up HMI Advanced TX1 Creating Foreign Language Texts	(03.04 Edition)



Notes

D Index**A**

Access authorization 2-45
Acknowledge alarm key 2-35
Acknowledge symbols 8-407
Activate skip levels 4-161
Activate tool offset immediately 5-190
Activation 1-25
Alarm help 2-71
Alarms 8-406
Alt key 2-37
Area switchover key 2-35
Arithmetic parameters 5-232
Automatic 2-41, 4-140
Automatic program overview 4-142
Axis feeds 4-106

B

Basic frame
 Activate 5-250
Baud rate 7-373
Block display during program execution 4-163
Block search 4-150
Block search in program testing mode - multi-channel 4-156
Block search without calculation, external, accelerated 4-153
Blueprint programming 6-275

C

Chain rule 5-181
Change language 9-438
Changing/finding user data 5-252
Channel
 Active 2-65
 Interrupted 2-65
 Reset 2-65
Channel status 2-65
Channel switching 2-65
Channel switchover 2-35
Channels 4-96
Cleaning 10-461
Close the contour 6-285
Communication error log 8-416
Configuration data 8-416
Continuous JOG 5-235

Contour

 Graphic representation 6-281
 Symbolic representation 6-280

Contour element

 Append 6-283
 Insert 6-284

Contour elements

 Input screens for parameterization 6-282

Copy 6-349**COPY 2-82****Create a new directory 7-389****Create a new file 7-389****Create tool 5-184****Ctrl key 2-37****Cursor 2-36****Cutting edge**

 Create 5-187

Cutting edge - NC memory 5-188**CYCLE800 4-107****Cycles 6-263****D****Data bits 7-373****Data selection 7-366****Data structure 4-93****Deactivation 1-25****Define and activate user data 7-395****Define user data 5-251****Delete 6-352****Delete key 2-37****Delete key (backspace) 2-36****Determine tool offsets 5-188****Diagnostics main screen 8-404****Directories 7-363****Display active axes 8-409****Display all axes 8-409****Display auxiliary functions 4-107****Display G functions 4-106****Display groups 9-444****Display loadable compile cycles 8-422****Display machine data filter 9-444****Display options 9-443****Display program level 4-103****Display selected axes 8-409****Display spindles 4-108****Display swivel data record 4-107****Display system frame 4-128**

- Display system frames 4-133
- Display tool data 5-222
- Display transformations 4-106
- Downtime settings 6-328
- DRF offset 4-164
- Drive diagnostics 8-408
- Drive service display 8-411
- Dry run feedrate 5-237
- Dual ASCII editor
 - Synchronized view 6-302
 - Views 6-302
- Dual editor 6-270
- Dual-channel display 4-98
- Dual-step editor
 - Synchronized display 6-303
- E**
- Edit key 2-36
- Edit tool data 5-222
- Editor gelp 2-71, 2-73
- Electronic gear 5-241
- EMERGENCY STOP key 2-40
- Enabling 6-355
- End of line key 2-37
- Executing job list 2-85
- Expansion key 2-34
- Exporting cycle versions 8-421
- Extended help for programming commands 2-77
- External network drive 6-357
- F**
- Fast external block search without calculation 4-153
- Feedrate override 2-42
- File
 - Change properties 7-393
 - Copy 6-349, 7-390
 - Download 7-389
 - Insert 6-349
 - NC memory 6-352, 7-392
 - Paste 7-390
 - Rename 6-353
 - Upload 7-389
- File functions 8-433
- File names 7-369
- File types 6-339, 7-364
- Find 2-61
- Find path for program call 6-345
- Find/Go to... 2-61
- Flat D numbers 5-167
- Following axis 5-241
- Free contour programming 2-62
- G**
- Geometry axes 4-104
- Go to 2-61
- Graphic model 6-329
- H**
- Handwheel 4-109
- Hard disk drive 4-145, 7-361
- Hardkey assignment 2-32
- Help function 2-73
- Help in the editor 2-77
 - Overview 2-73
- Help in the Editor 2-74
- Help with free contour programming 6-293
- Hierarchical file structure 7-365
- HMI A-467
- HMI interface
 - Changing 9-451
- I**
- Inc keys 2-41
- Inc. increment 4-123
- Inch/metric conversion 2-70
- Inch/metric switchover (from SW 5) 4-114
- Increment value 2-41
- Incremental mode 2-41
- Information key 2-35, 2-74, 2-77
- Initialization program
 - User data, define 5-251
- Input key 2-37
- Interface parameterization 7-375
- Interfaces 7-361
- J**
- Job list 2-78
- Job list syntax 2-80
- JOG 2-40, 4-119
- JOG data 5-235
- JOG feedrate 5-235
- JOG main screen 4-120
- JOG spindle speed 5-235
- K**
- Key functions
 - Step display 6-300
- L**
- Leading axis 5-241
- Limit of data management 7-362

- Load 5-210
 - From magazine list 5-211
- LOAD 2-80
- Load - from magazine list 5-214
- Load standard 6-344
- Loading 6-347
- Loading list 4-144
- Log 6-356
- M**
- MAC
 - Changing/finding user data 5-252
- Machine (MCS) 2-43
- Machine area key 2-34
- Machine axes 4-104
- Machine data
 - Display 9-441
- Machine function 2-41
- Machine functions 4-94
- Machine zero 5-242
- Machining list 6-332
- Mark block 2-60
- MCS 2-43, 4-104
- MD help 2-71
- MDI 2-40, 4-135
- MF-2 keyboard 2-32
- MMC A-469
- MMC 103
 - Reading in data 7-383
 - Reading out data 7-384
- MMC 103 main screen 7-376
- MMC 103 series start-up 7-397
- MMC 103 setting interface 7-379
- MMC 103 upgrade 7-400
- MMC103
 - Log 7-386
- Mode change 4-99
- Mode group 4-96
- Modes 4-94
- Multi-channel block search in program testing
 - mode 4-156
- N**
- Navigation
 - Step display 6-301
- NC
 - Restoring the original state 7-399
- NC active data 7-360
- NC card
 - Delete on (from SW5) 7-392
 - Read in data from... (SW 4.4 and higher) 7-384
 - Read out data to... (SW 5 and higher) 7-385
 - Series start-up of (from SW 5) 7-398
 - Upgrade from ... (SW 5 and higher) 7-401
- NC Start 2-46
- NC Stop 2-46
- Network drive 6-357
- New part program 6-340
- New tool 5-184
- New tool edge 5-187
- New workpiece 6-340
- O**
- Operand screens 8-432
- Operand value
 - Change 8-425
 - Delete 8-425
- Operating areas 1-23
- Operating data 10-460
- Operation
 - MMC 103 7-376
- Operator panels 2-29
- Orientable toolholder
 - Simulation 6-334
- Overstore 4-158
- Overwrite 2-60
- P**
- Page down 2-36
- Page up 2-37
- Parameter no. for radius calculation 5-181
- Parity 7-373
- Part programs 6-263
 - Continue 4-102
 - Load, unload 4-143
 - Select 6-340
 - Start 4-102
 - Stop 4-102
- Password 2-45
 - NC memory 9-440
- Path labels 6-332
 - Management 6-329
- Path name 7-367
- PC format 7-371
- PLC 9-447
- PLC status 8-424
- Pocket calculator 2-70

- Predefined package lists 8-419
- Preset 4-112
- Program
 - Edit 6-269
 - Enabling 6-355
 - Load 6-347
 - Main screen 6-267
 - Storing 6-267
 - Unload 6-347
- Program control 4-160
- Program editing 4-148
- Program management 6-338, 6-348
- Program protection
 - Selective 6-273
- Program simulation 6-311
- Program types 6-263
- Protection levels 2-45
- Protection zones 5-240
- Punched tape format 7-368
- R**
- R parameters 5-232
 - Edit 5-232
 - Find 5-233
 - NC memory 5-233
- Rapid traverse override 2-42, 2-43
- Recall key 2-34
- Recompile 2-63
- Recompile cycle 2-63
- Ref 2-41
- Reference point approach 4-116
- Rename 6-353
- Repos 2-41, 4-124
- Reset 2-46
- Restoring the original state 7-399
- RS-232 interface parameters 7-372
- S**
- Safety Integrated 4-125
- Save
 - Setup data 6-343
- Save file 2-62
- Save program (MMC 103) 4-137
- Save setup data 6-343
- Scratching 4-126, 4-130
- Search destination 4-150
- Select 6-344
- SELECT 2-81
- Select operating mode 4-99
- Service axis 8-410
- Service display 8-408
- Service overview 8-408
- Service SI 8-412
- Set standard 6-344
- Setting data 5-234
 - Other types 5-239
- Setting data - Edit 5-239
- Setting data - Find 5-239
- Setting data display 5-239
- Setting parameters for cycles 2-62
- Setting the actual value 4-113
- Setting the date 9-448
- Setting the time 9-448
- Settings (MMC 103 only) 2-63, 6-298
- Shift key 2-35
- Short help for program commands 2-74
- Simulation 2-63
 - Orientable toolholder 6-334
- Simulation drilling/milling 6-311
- Simulation of complete turning operations 6-311
- Simulation user interface 6-312
- Single block 2-46
- Softkey 2-34
- Softkey assignment 2-32
- Software version 8-417
- Space 2-36
- Special axes 4-104
- Special functions 7-374
- Spindle data 5-236
- Spindle number 5-181
- Spindle override 2-44
- Standardized program and data management 7-388
- Starting angle for thread cutting 5-238
- Startup
 - Main screen 9-436
- Step display
 - Selection 6-299
- Step editor 6-299
 - Collapsed display 6-300
 - Expanded display 6-300
 - Multi-channel simulation 6-310
 - Operation 6-300
- Step identifier 6-301
- Stop bits 7-373
- Subroutine 6-263

- Suppression filter 9-443
- Switch over MCS/WCS 4-104
- Synchronized actions
 - Status 4-110
- System resources
 - Display 8-434
- System settings 9-454
- System variable
 - Display 5-254
- System variables
 - Log 5-257
- T**
- Tab key 2-37
- TCARR 4-107
- Teach in 2-41, 4-135, 4-138
- Templates 6-263
- Toggle key 2-36
- Tool - relocate 5-216
- Tool cabinet 5-220
- Tool catalog 5-217
- Tool compensation 5-182
- Tool compensation structure 5-167
- Tool data 5-167
 - Display 5-218
 - Display/edit 5-201
 - Edit 5-218
- Tool data, creation 5-218
- Tool display 5-185
- Tool management 5-191
- Tool management, main screen 5-193
- Tool NC memory 5-186
- Tool offset data
 - Create 5-221
- Tool parameter calculation 5-176
- Tool parameters 5-167
- Tool type
 - Cutting/milling tools 5-168
 - Drills 5-169
 - Grinding tools 5-170
 - Slotting saw 5-174
 - Turning tools 5-173
- Tool types 5-167
- Tool zero 5-242
- Tool-specific parameters 5-173
- Traverse the axes 4-122
- Traversing velocity 4-122

U

- Undo key 2-36
- Unload 5-214
- User data 5-251
- Utilization display 8-434

V

- Variable increment 5-235
- Variable views
 - Create 5-255
 - Edit 5-255
 - Management 5-257
- Version screen for cycles 8-419

W

- WCS 2-43, 4-104
- Window selection key 2-35
- Work offset 5-242
 - Activate immediately 5-250
 - Active external 5-249
 - Active programmable 5-248
 - Active settable 5-247
 - Sum of active 5-249
- Work offset - settable 5-244
- Working area limitation 5-234
- Workpiece 6-263
 - Enabling 6-355
 - Execution 6-344
 - Load, unload 4-143
 - Select 6-340, 6-344
- Workpiece directory, create 6-340
- Workpieces
 - MMC 6-343
 - Quantity 6-343

E Commands, identifiers

G

GUD 5-251

P

Password 9-439

 Change 9-440

 Delete 9-440

 Set 9-439

Protection levels 9-438

S

SEFORM 6-299

G-BI A 9F 7', (\$8# (\$8]# %\$8

In the event that the end user of this product is to be the military and said product is to be employed in any weapons systems or the manufacture thereof, the export will fall under the relevant regulations as stipulated in the Foreign Exchange and Foreign Trade Regulations. Therefore, be sure to follow all procedures and submit all relevant documentation according to any and all rules, regulations and laws that may apply. Specifications are subject to change without notice for ongoing product modifications and improvements.

Machine Tool OEM Sales Div.

Gate City Osaki West Tower, 1-11-1, Osaki, Shinagawa-ku, Tokyo 141-8644, Japan
PHONE +81-3-3493-7411 FAX +81-3-3493-7422

Siemens Japan K.K.
<http://www.siemens.co.jp>

Published in Japan

02-WSC02, 02-WSC09

© Siemens Japan K.K. All rights reserved.