

**SIEMENS**

# **SINUMERIK 840C**

## **Software Version 6**

**Operator's Guide**

**09.2001 Edition**



# SIEMENS

## SINUMERIK 840C

### Software Version 6

#### Operator's Guide

**Valid for**

<i>Control</i>	<i>Software Version</i>
SINUMERIK 840C/CE	
(Standard/Export version)	6

**09.2001 Edition**

**Introduction**

**1**

**Operator Interface**

**2**

**Machine**

**3**

**Parameter**

**4**

**Programming**

**5**

**Services**

**6**

**Diagnosis**

**7**

**Maintenance**

**8**

**List of Abbreviations**

**9**

# 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
11.92	6FC5198-0AA00-1BP0	<b>A</b>
06.93	6FC5198-2AA00-0BP0	<b>C</b>
12.93	6FC5198-3AA00-0BP0	<b>C</b>
10.94	6FC5198-4AA00-0BP0	<b>C</b>
03.95	6FC5198-4AA00-0BP1	<b>C</b>
09.95	6FC5198-5AA00-0BP0	<b>C</b>
04.96	6FC5198-5AA00-0BP1	<b>C</b>
08.96	6FC5198-5AA00-0BP2	<b>C</b>
07.97	6FC5198-6AA00-0BP0	<b>C</b>
01.99	6FC5198-6AA00-0BP1	<b>C</b>
09.01	6FC5198-6AA00-0BP2	<b>C</b>

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

Edition	Order No.	Remarks
10.01	6FC5 198-6CA00-0BG2	<b>C</b>

## Trademarks

SIMATIC®, SIMATIC HMI®, SIMATIC NET®, SIROTEC®, SINUMERIK® and SIMODRIVE® are trademarks of Siemens AG. All other product and system names are registered trademarks of their respective companies and must be treated accordingly.

For more information refer to the Internet under:  
<http://www.ad.siemens.de/sinumerik>

This publication was produced with Interleaf V 7

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

© Siemens AG 1992-2001.  
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.

# Preliminary Remarks

*Notes for the reader* The SINUMERIK documentation is organized in four parts:

- General documentation
- User documentation
- Manufacturer documentation and
- Service documentation



This documentation has been written for machine tool users. The publication provides detailed information required by the user for operating the SINUMERIK 840C control.



This Operator's Guide only describes functions which belong to the standard version. Options or modifications carried out by the machine manufacturer, for example a special additional operator panel from the machine manufacturer cannot be described in this Operator's Guide.



More information on other SINUMERIK 840C publications and on publications which are available for all SINUMERIK controls (e.g. "Universal Interface", "Measuring Cycles" ...) can be obtained from your Siemens local branch office.

Other functions not described in this documentation might also be executable in the control.

This does not, however, represent an obligation to supply such functions with a new control or when servicing.

	The symbol shown here appears in this documentation when facts of importance must be observed.	
---	--	---

	The symbol shown here appears in this documentation whenever the machine tool manufacturer can influence or modify the described function by changing a machine data (MD). Please observe the machine manufacturer's specifications.	
---	--	---

	This Operator's Guide applies to: SINUMERIK 840 C/CE control Software Version 6 (6.3)	
---	---	---

## Safety



### **Danger**

This warning notice means that loss of life, severe personal injury or substantial material damage **will** result if the appropriate precautions are not taken.



### **Warning**

This warning notice means that loss of life, severe personal injury or substantial material damage **can** result if the appropriate precautions are not taken.



### **Caution**

This warning notice (with warning triangle) means that a minor personal injury **can** result if the appropriate precautions are not taken.

### **Caution**

This warning notice (without warning triangle) means that a material damage **can** result if the appropriate precautions are not taken.

### **Notice**

This warning notice means that an undesired event or an undesired state **can** result if the appropriate notices are not observed.

# Contents

	Page
<b>1</b>	<b>Introduction – The SINUMERIK 840C and its Operation . . . . . 1–1</b>
1.1	Switching on the control . . . . . 1–2
1.2	Structure of the operator interface . . . . . 1–3
1.3	Menu switchover . . . . . 1–5
1.4	Directory structure . . . . . 1–7
1.5	The meaning of the SIEMENS and user directories . . . . . 1–11
1.6	Screen darkening . . . . . 1–12
1.7	Data backup options . . . . . 1–13
1.8	Help displays . . . . . 1–14
<b>2</b>	<b>Operator Interface . . . . . 2–1</b>
2.1	Operator panel . . . . . 2–1
2.2	Operating elements . . . . . 2–2
2.2.1	Operator panel . . . . . 2–2
2.2.2	PC standard keyboard (MF–2) . . . . . 2–5
2.2.3	Operating elements of the operator panel keyboard/full keyboard . . . . . 2–7
2.2.4	The operating elements of the machine control panel . . . . . 2–12
2.2.5	Operating elements on the hand-held unit . . . . . 2–21
2.2.6	Operating elements on the A–MPC hand-held unit . . . . . 2–22
2.2.7	Screen layout . . . . . 2–26
<b>3</b>	<b>Machine . . . . . 3–1</b>
3.1	Operating modes . . . . . 3–3
3.1.1	Operating states – Operating state changes on change of operating mode . . . . . 3–5
3.1.2	Machine functions – overview . . . . . 3–7
3.1.3	Status displays . . . . . 3–8
3.1.3.1	Channel-independent status display by means of icons . . . . . 3–8
3.1.3.2	Spindle utilization display . . . . . 3–9
3.1.4	JOG: Set-up mode . . . . . 3–10
3.1.4.1	Traversing in JOG mode . . . . . 3–11
3.1.4.2	Approach reference point (REFPOINT) . . . . . 3–12
3.1.4.3	User agreement (“Safety Integrated” option) . . . . . 3–13
3.1.4.4	Increment mode – selection . . . . . 3–14
3.1.4.4.1	Indexed traversing . . . . . 3–15
3.1.4.4.2	Indexed traversing with incremental dimension (INC) . . . . . 3–15
3.1.4.5	Repos (Repositioning) . . . . . 3–16
3.1.4.6	Scratching . . . . . 3–17
3.1.4.7	Finish thread . . . . . 3–18
3.1.5	TEACH IN mode . . . . . 3–19
3.1.5.1	Set breakpoints . . . . . 3–22
3.1.5.2	Edit mode . . . . . 3–24
3.1.5.3	Block structure settings . . . . . 3–25
3.1.5.4	Creating a TEACH IN program in the edit mode . . . . . 3–27
3.1.5.5	Accepting axis positions . . . . . 3–28
3.1.5.6	MDA in edit mode . . . . . 3–29

3.1.5.7	Block by block teach-in .....	3-31
3.1.5.8	Modifying an existing part program with block by block teach-in .....	3-32
3.1.5.9	Examples .....	3-37
3.1.6	MDA mode .....	3-40
3.1.6.1	Copying MDA programs .....	3-42
3.1.7	AUTOMATIC mode .....	3-44
3.1.7.1	Workpiece and program selection with the data selector .....	3-45
3.1.7.2	SELECT PROGRAM/JOB function .....	3-50
3.1.7.3	Starting and interrupting a part program .....	3-51
3.1.7.4	Search for first letter function (from SW 6) .....	3-53
3.1.7.5	Program view (from SW 6) .....	3-54
3.1.7.6	Expanded program view (from SW 6.3) .....	3-55
3.2	Additional machine functions .....	3-56
3.2.1	Overstore .....	3-56
3.2.2	Extended overstore .....	3-57
3.2.3	Altering F and S values on-line .....	3-60
3.2.4	Program modification .....	3-61
3.2.4.1	Description of the functions DRY, M01, ROV, ACR, DRF, DSB, PST, BRK, CLR, EXT, NCY, SAV, SKP and predec. blocks .....	3-63
3.2.4.2	Single block/Decoding single block .....	3-66
3.2.5	Block search .....	3-68
3.2.6	Program correction .....	3-72
3.2.7	Saving programs .....	3-74
3.2.8	PRESET (Set actual value)/DRF .....	3-77
3.2.8.1	PRESET – Offset .....	3-77
3.2.8.2	DRF offset .....	3-80
3.2.9	Axis-specific G functions .....	3-81
3.2.10	Extended stop and retract .....	3-81
3.3	Multichannel display .....	3-83
<b>4</b>	<b>Parameter .....</b>	<b>4-1</b>
4.1	Editing data in the PARAMETER area .....	4-3
4.1.1	Selecting data .....	4-3
4.1.2	Entering and correcting data .....	4-5
4.1.3	Entering PLC data in ASCII format .....	4-6
4.2	Program parameters .....	4-7
4.2.1	Tool offsets .....	4-7
4.2.2	Zero offset .....	4-10
4.2.3	Angle of rotation (coordinate rotation) .....	4-12
4.2.4	R parameters .....	4-13
4.2.5	Plane .....	4-15
4.2.6	Setting data .....	4-15
4.2.6.1	Working area limitation .....	4-16
4.2.6.2	General setting data .....	4-16
4.2.6.3	Spindle setting data .....	4-17
4.2.6.4	Scale .....	4-17
4.2.6.5	General setting data bits .....	4-18
4.2.6.6	Axial setting data bits .....	4-18
4.2.6.7	Position measuring signals .....	4-19
4.2.6.8	Cycle setting data .....	4-20
4.2.6.9	Axis and spindle converter .....	4-21
4.2.6.10	Gearbox interpolation .....	4-22
4.2.6.11	Travel to fixed stop .....	4-27
4.2.7	Additional protection zone adjustment via setting data (from SW 6.3) ...	4-28
4.2.8	Inch/metric conversion (from SW 6.3) .....	4-29



<b>5</b>	<b>Programming</b>	<b>5-1</b>
5.1	Programming in the NCK memory area (directory NCK)	5-3
5.1.1	Select program	5-4
5.1.2	Editing an existing program	5-5
5.1.3	Editing a new NC program	5-8
5.1.4	Graphic tool path simulation	5-10
5.1.5	Program input with operator support	5-15
5.1.6	Machining cycles	5-17
5.1.7	Plane	5-17
5.1.8	Program management	5-18
5.1.8.1	Changing the access rights	5-18
5.1.8.2	Copy program	5-20
5.1.8.3	Rename program	5-21
5.1.8.4	Delete program	5-22
5.1.8.5	Move cycles	5-24
5.1.9	FANUC program converter (from SW 4.9)	5-25
5.2	Programming with the ASCII editor in the hard disk data management	5-40
5.2.1	Creating a workpiece	5-41
5.2.2	Creating a part program file	5-43
5.2.3	Creating job lists	5-47
5.2.3.1	General	5-47
5.2.3.2	Creating the JOB LIST file	5-47
5.2.3.3	Editing job lists	5-50
5.2.3.4	Creating job lists	5-51
5.2.3.5	Syntax description for the job lists	5-52
5.2.4	Shopfloor sheet	5-56
5.2.5	Copy, paste and delete files in the data management area of the hard disk	5-57
5.2.6	Description of the ASCII editor	5-58
5.2.6.1	Editing with the ASCII editor	5-58
5.2.6.2	Input fields	5-63
5.2.6.3	Selection fields (toggle fields)	5-64
5.2.6.4	Double ASCII editor	5-64
5.2.7	Viewing IKA data	5-65
<b>6</b>	<b>Services</b>	<b>6-1</b>
6.1	LOAD NC	6-3
6.1.1	Loading workpieces and programs	6-4
6.1.2	Loading data	6-7
6.2	SAVE NC	6-9
6.2.1	Saving NC data	6-10
6.2.2	Saving part programs or NCK data in a workpiece in the local or global directory	6-13
6.3	Data input from an external device	6-16
6.3.1	Insert data from the clipboard	6-19
6.3.2	Continuous mode for data input via RS232C (V24) (from SW 5.6)	6-21
6.4	Data output to an external device	6-22
6.4.1	Output formats	6-24
6.4.1.1	Punch tape format	6-24
6.4.1.2	PC format	6-24
6.4.2	Outputting workpieces and archives	6-24
6.4.3	Data output	6-26
6.5	Data input and output with the FD-E2 floppy drive (option)	6-27
6.5.1	Data output with the FD-E2 floppy drive	6-28
6.5.2	Data input from the FD-E2 floppy drive	6-30
6.5.3	Formatting diskettes	6-31
6.6	Data input and output with the SINUMERIK FD-E1 floppy drive (option)	6-32
6.7	Printing out files	6-33

6.8	Archive lists .....	6-34
6.8.1	Creating new archive lists .....	6-35
6.8.2	Editing existing archive lists .....	6-38
6.8.3	Edit, copy and delete archive lists .....	6-39
6.9	Device settings (up to SW 6.2) .....	6-40
6.9.1	Selecting an interface .....	6-41
6.9.2	Editing interfaces .....	6-41
6.9.3	Device settings (from SW 6.3) .....	6-44
6.10	Data management .....	6-47
6.10.1	Copy and paste in the SERVICES/MANAGEMENT area .....	6-48
6.10.2	Delete data in the SERVICES/MANAGEMENT area .....	6-49
<b>7</b>	<b>Diagnosis .....</b>	<b>7-1</b>
7.1	Alarm and message displays .....	7-2
7.2	Service displays .....	7-3
7.2.1	Alarm logs .....	7-3
7.2.2	NC information .....	7-4
7.2.3	Drift compensation .....	7-4
7.3	Softkeys for the start-up engineer in the DIAGNOSIS basic display .....	7-6
<b>8</b>	<b>Maintenance .....</b>	<b>8-1</b>
8.1	Operating data .....	8-1
8.2	Replacing the battery .....	8-2
8.3	Handling modules .....	8-4
8.4	Practical tips on remedying electromagnetic compatibility problems .....	8-6
8.5	Cleaning .....	8-8
<b>9</b>	<b>List of Abbreviations .....</b>	<b>9-1</b>

# 1 Introduction – The SINUMERIK 840C and its Operation

The SINUMERIK 840C is a CNC (**C**omputer **N**umerical **C**ontrol), intended mainly for automation applications.



Generally, the CNC implements the following basic functions (in conjunction with a machine tool or such like):

- Automatic workpiece machining
- Free programming of axes
- Programming of technological functions such as feed, spindle speed, etc.
- Controlling of the axes and spindles in conjunction with the drives and the measuring systems
- Scanning and controlling input and output signals via the PLC program
- Machine operation via the machine control panel
- Storing user data in the CNC memory
- Organizing data exchange with I/O devices.

The operator interface (monitor displays, keyboard) is the connecting element between operator action and the machine. This Operator's Guide describes only functions which are within the standard scope of supply of Siemens. The machine manufacturer can also configure functions and monitor displays and he can connect his own keyboard.

Please read the machine manufacturer's Operator's Guide in these cases.

## 1.1 Switching on the control

	Switching on of the control can be implemented in various ways by the machine manufacturer. Please refer to the machine manufacturer's specifications.	
---	--	---

Please note:

The control is switched on by switching on the input voltage at the power supply of the central controller.

After the control has been switched on, different start-up images appear for a few seconds. The control software is loaded, i.e. the NCK and the MMC software is booted from the hard disk.

When the control is installed, the PLC software is loaded from the hard disk into the battery-backed RAM memory, i.e. the PLC software is not booted when the control is powered up.

Power-up of the control is performed in the following sequence:

1. The operating system software is loaded
2. The user data are loaded from the NC/DATA directory on the hard disk. These might be IKA and GIA data. The machine data (TEA1, TEA2, etc.) are **not** loaded during power-up. These data are stored in battery-backed memory.
3. The system now loads all of the data assigned to the STANDARD workpiece. The Siemens and user cycles can only be loaded with the STANDARD workpiece (LOADCYC command in the job list).  
The machine manufacturer can configure whether the last selected workpieces are to be loaded from the hard disk at this stage. If, in the 3rd phase, the same data are loaded with a workpiece (e.g. RPA, TOA etc.) as in the 2nd phase, these data are overwritten by the workpiece-related data.

If not otherwise configured by the machine manufacturer, the main menu of the JOG mode appears after power-up of the control. Operator actions on the control can be performed after the screen has built up completely.

## 1.2 Structure of the operator interface

### 5 operating areas

The SINUMERIK 840C operator interface is divided into 5 areas: MACHINE, PARAMETER, PROGRAMMING, SERVICES and DIAGNOSIS. The 5 areas will be displayed as softkey functions any time you press the area switchover key.

Area switchover key

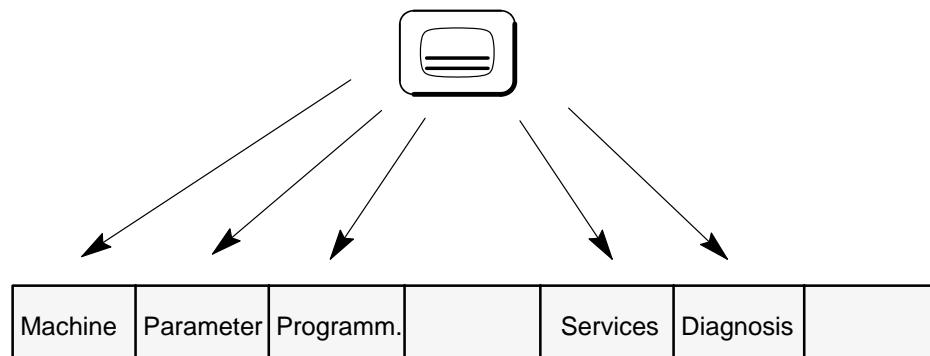


Fig. 1.1 shows you which operator actions are possible in the different operating areas. With the SINUMERIK 840C, you can edit the part programs in two different memories:

- in the NCK memory (volatile RAM memory)
- on the hard disk (MMC–CPU).

If you work on the volatile RAM memory, you must store the edited or newly created programs on the hard disk if they are to be retained after switching off the control.

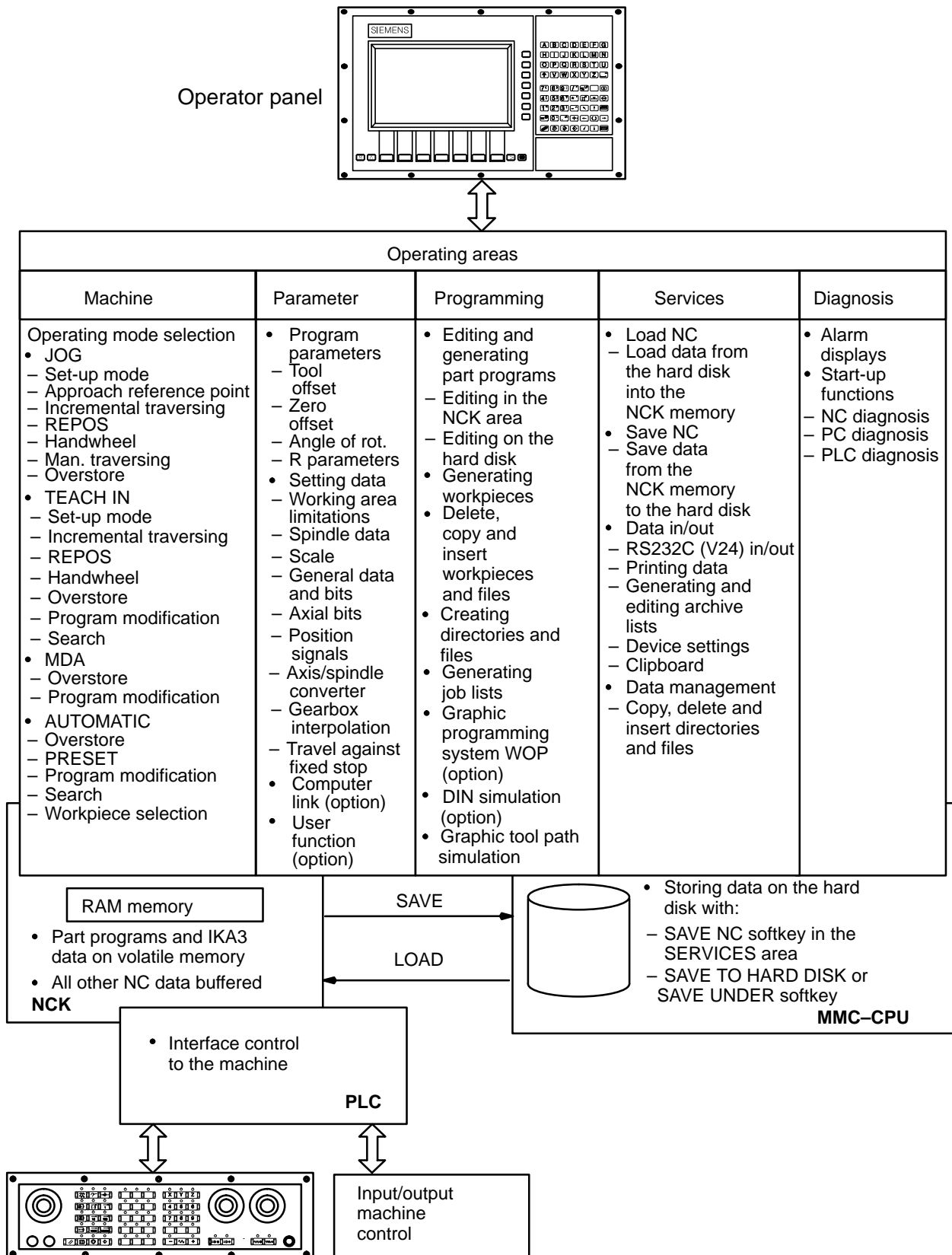
If you edit a program on the hard disk, this program must be loaded in the NCK memory in order to be executed.

For this purpose, the SAVE softkey function is provided in the MACHINE and PROGRAM areas as well as the SAVE NC function in the SERVICES area. The PROGRAM area optionally offers a graphic programming system and a DIN code simulation. You will find a description of this system in separate publications of the User's Guide Graphic Programming System for

- Drilling/Boring and Milling
- Turning

and in User's Guide, Simulation: Milling and Turning.

## 1.2 Structure of the operator interface



## 1.3 Menu switchover

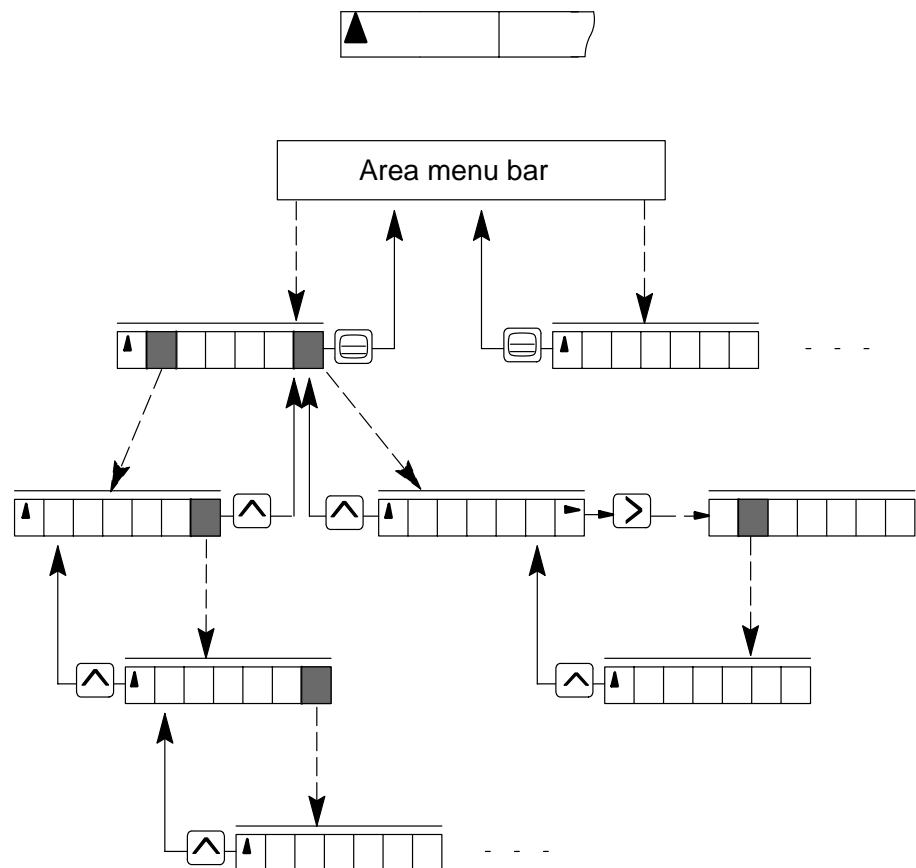
Softkeys are keys which are variably assigned with functions.

The 5 areas of the SINUMERIK 840C are selected via softkeys. Within these areas further branching via softkeys is possible.

Operating the RECALL key brings you back to the next higher menu level.



An arrow in the first softkey window indicates that you can return to a menu:



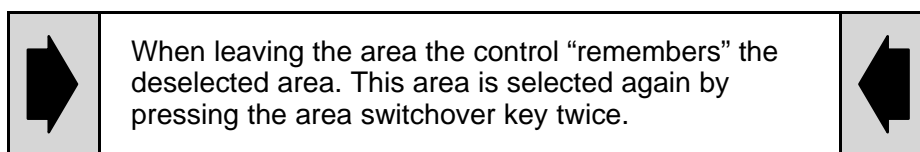
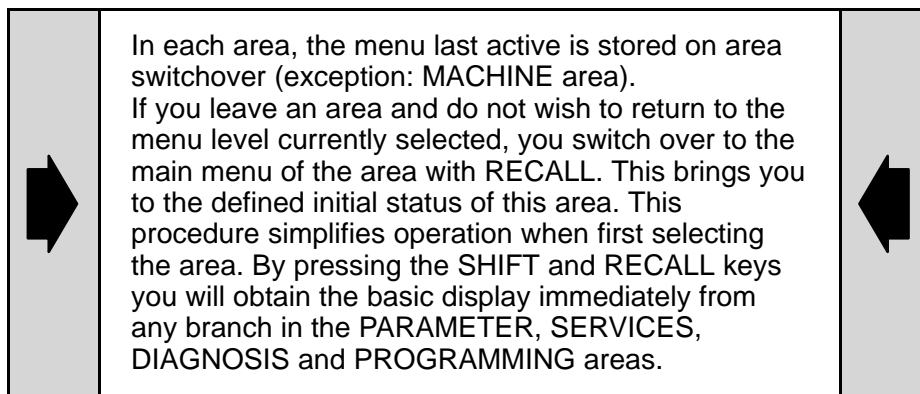
With the ETC key, you can expand the softkey menu if an extension is available.

If there is an arrow in the 7th softkey window



(on the right), this key is active.

## 1.3 Menu switchover





## 1.4 Directory structure


### Directories

Structures from the PC world have been integrated in the SINUMERIK 840C. The PROGRAMMING, SERVICES and DIAGNOSIS areas as well as program selection (DATA SELECTOR) contain directories. These directories are a file management aid for sorting and storing files. Directories can in turn contain subdirectories. Only **one** directory level is displayed on the screen at any one time.

### File

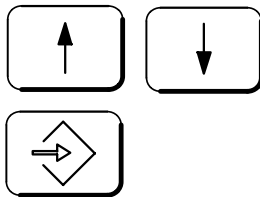
The softkey bar does **not** change when you “page” through the directories. Softkey function and directory are not directly related. A file is a logical data unit which you can address with a name (e.g. MPF1, TOA1, ...). Files can be assigned to directories according to specific criteria, i.e. MPF1 to the workpiece (directory) SHAFT1 or TEA1 to the directory NC. You can assign directories to other directories, which you also address with a name (e.g. LOCAL directory).

### Directory path

This results in “directory paths” which you can page through as follows. Using the vertical cursor keys, set the cursor on a directory and press the INPUT key. The next subdirectory in this directory is opened. Continue doing this until you have reached the directory of the files. You recognize files by the entry in the LENGTH column. You reach the next higher level from a directory level by using the vertical cursor keys to position the cursor on the line with the double dot  at the beginning of the directory and then acknowledge with INPUT. If you have reached the highest level, the double dot is replaced by a hyphen.

If you change the area by means of key or softkey functions (e.g. change from SERVICES to DIAGNOSIS) from any directory, the directory level last selected is stored. When reselecting the area, this brings you back to the directory used previously. Switchover between areas can be performed at any time, e.g. to look at a parameter or machine data.

### Data selector



A program that interprets your keyboard entries (cursor, input) as a file or directory selection, displays it on the screen (bar) and passes it on to other programs (e.g. editor).

### Hotkeys



With 840C software version 5 and higher the data selector responds to hotkeys. The first object (directory/file) in the current directory that starts with the letter or number entered is found and selected.

## 1.4 Directory structure

*Workpiece*

With the SINUMERIK 840C you can store part programs and NCK data (TOA, SEA, RPA, ZOA) in a directory from now on referred to as workpiece. You can then load this workpiece and its data completely into the NCK area. Thus you can store all relevant data (part programs, tool offsets) required for a workpiece under one workpiece name and load it any time you wish.

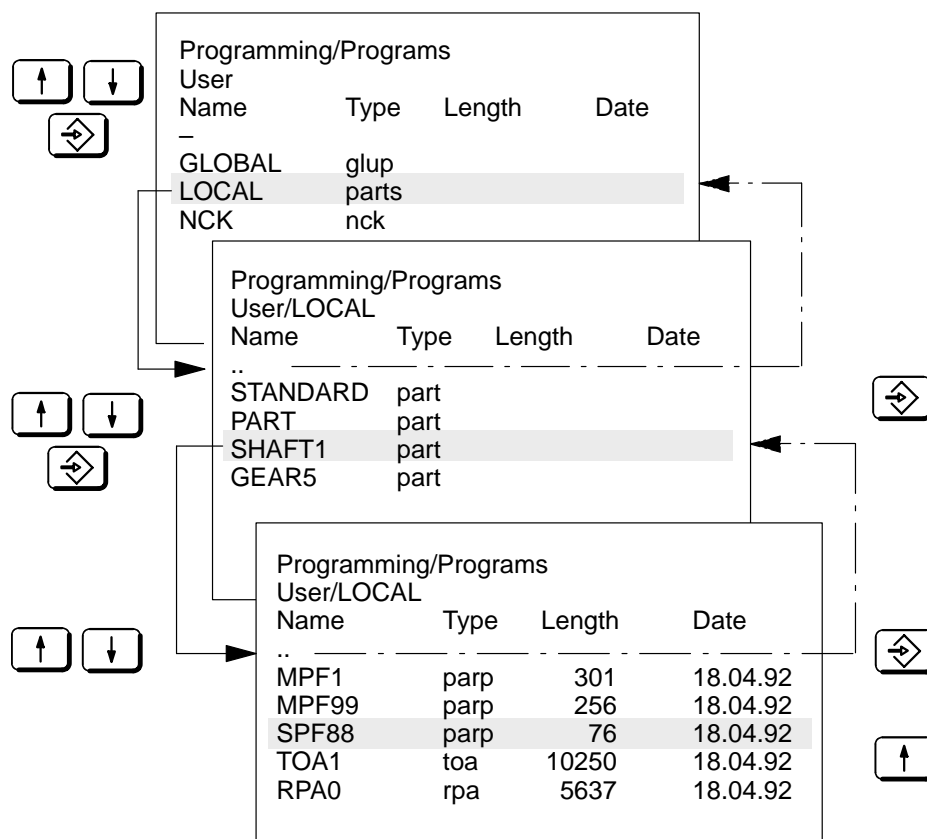


Fig. 1.1 Representation of the directory path PROGRAMMING/LOCAL/SHAFT1/SPF88

The directory path (e.g. User/LOCAL/SHAFT1) is displayed each time.



*Type*

In addition to being identified by name, workpieces and files are also assigned a type identifier. This provides a unique assignment between the variable name and the created file or directory type. The type identifiers are as follows:

Designation	Type	Designation	Type
Alarm help text	alrminf	Language	lang
Archive list	arc	LB data lists	lbds
Archive lists	arcs	LB semantics lists	lbss
Basic settings	bdgl	List module	lb
Boot strapper	nckcnf	Local programs	parts
Boot strapper	plccnf	Log	prot
Boot strapper	serbo	Log	prpr
Clipboard	clip	Logbook	rlog
Color assignment list	colli	Logs	prprs
Color table	clut	Master control	reg
Comment	comm	Measured value diagram 1	mdi1
Comment text	cmt	Measured value diagram 2	mdi2
Configuration	bdcnf	Memory dump	plcprg
Configuration	simcnf	Message descriptions	alrmda
Configuration	syncnf	Message descriptions (bin)	alrmd
Configuration texts	bdcnfl	Message texts	alrmta
Curve pointer	ika2	Message texts (bin)	alrmt
Curve points	ika3	Motor list	motl
Cycle	zpf	NC	nck
Cycle machine data (TEA4)	tea 4	NC machine data (TEA1)	tea1
Cycle setting data (SEA4)	sea 4	NC macro	nckmk
Data	mda	NC matching	nckmd
Data areas	dber	OEM	oem
Data list	lbd	Operator action	bedsy
Data management	dasy	Options	md
Data references	daref	Part program	parp
Device	dev	PC	mmc
Device assignment	secnf	PC area	mode
Drive machine data (TEA3)	tea3	PC system	sys
Drives	servo	PC user memory submodule (UMS)	mmcas
Empirical values	heur	PLC	plc
Error texts	simtxt	PLC machine data (TEA2)	tea 2
Form element	gpmk	Power section list	ltl
Function	func	Programs	progs
Functions	funcs	R parameter (RPA)	rpa
Function test parameter	fkt	Scheme	metab
Global geometries	glgeo	Scheme source	metac
Global programs	glup	Scheme texts	metat
Graphics	grfk	Screen dump	hcpy
Graphics macro	gramk	Screen dumps	hcpys
Graphics mask	grmsk	Screen form	msk
Help	wophelp	Screen forms	msks
Help text	bdht	Semantics list	lbs
Help text	texl	Services	serv
Help texts	htex	Setting data (SEA)	sea
Icons	ikon	Siemens	siem
IKA number	ika1	Simulation	simul
Interfaces	devs	Source program	sour
Intermediate save	bak	Startup	ibn
Job list	jobl	Startup info texts	ibninf
Keyboard table	kbtb	Status data (GIA)	gia
Kinematics	kine	System	ncks

## 1.4 Directory structure

Designation	Type	Designation	Type
System	plcs	UMS texts	text
System file	ncsysf	User	user
System file	plcsysf	User memory submodule (ASM)	asm
System files	sersy	Variable geometry	vgeo
Templates	exam	WOP	wop
Temporary	temp	WOP control	wopst
Text list	lbt	WOP setup	wopset
Tool catalog	toolb	Workpiece	part
Tool geometry macro	tomk	Work schedule	fainf
Tool list	toli	Zero offsets (ZOA)	zoa
Tool offsets (TOA)	toa		

	<p>250 directories or files can be created for each directory.</p> <p>e.g.:</p> <ul style="list-style-type: none"> <li>Directory LOCAL : 250 workpieces</li> <li>Directory GLOBAL : 250 workpieces</li> <li>Directory SHAFT1 : 250 files</li> <li>etc.</li> </ul>	
---	---	---

## 1.5 The meaning of the SIEMENS and user directories

The directories are divided into a Siemens directory and a user directory in the areas where it is sensible to do so:

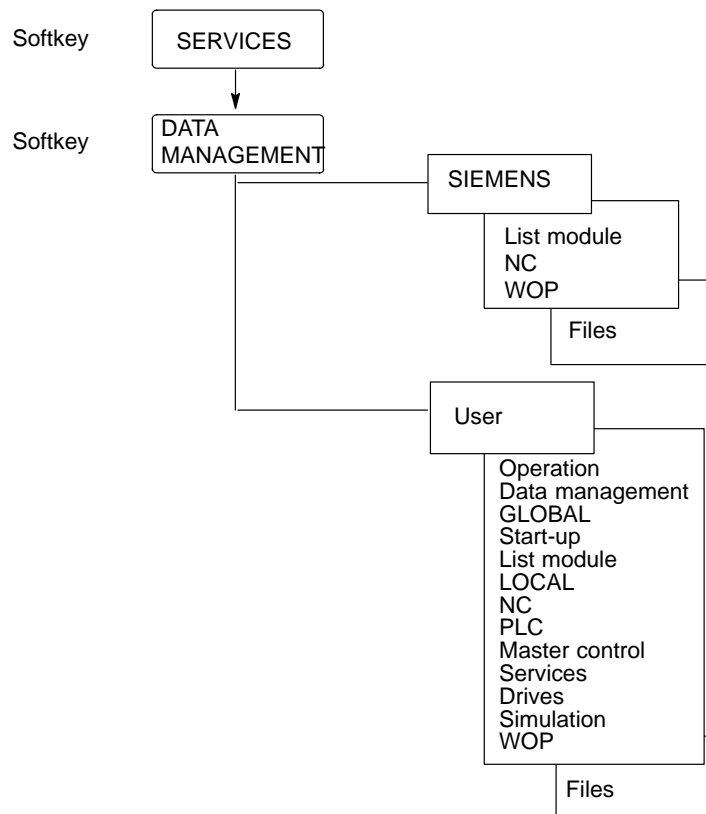


Fig. 1.2 SIEMENS directory and user directory in the SERVICES/DATA IN-OUT/DEVICES area

### Siemens directory

The Siemens directory includes all data which are relevant for the control's operating system. You cannot edit or read out data in the Siemens directory; you can merely copy this data into the user directory, where it can be handled in exactly the same way as user data.

### User directory

The user directory includes all user data. The fact that system data copied into the user directory can be edited there makes it possible to modify the configuration files of the control. The data of the user branch (if available) are always used before any other for the system configuration on powering up of the control. These possibilities are only relevant in the start-up phase and are password-protected against unauthorized operation.

You will find a possibility to use this structure in the SERVICES/DATA IN-OUT/DEVICE SETTINGS.

Here you can copy a standard interface configuration from the Siemens directory into the user directory and modify it in accordance with your data save device.



## 1.6 Screen darkening



The SINUMERIK 840C includes a function by which the screen is darkened after an adjustable period of time.

The default value of this period is 2 hours, i.e. as a rule the screen darkens 2 hours after a key has last been activated; this value can be changed in the PC data basic settings in the DIAGNOSIS area. The display is activated once any key on the alphanumeric keyboard or on the machine control panel, is pressed. The first keystroke simply reactivates the screen and does not initiate its normal function. With the screen darkening function, only the monitor is switched off and all other functions remain active.



By pressing the SHIFT and ACTUAL POSITION IN LARGE CHARACTERS keys, you can darken the monitor yourself and reactivate the display by pressing any key on the alphanumeric keyboard.

	Screen darkening can also be implemented by the machine manufacturer via the PLC program. In this case, the operator actions mentioned above do not apply. Refer to the machine manufacturer's Operator's Guide.	
---	--	---

	<b>Caution!</b> All internal processes which generate screen outputs do not cancel screen darkening. This also applies to alarms and operational messages.	
---	---	---

## 1.7 Data backup options

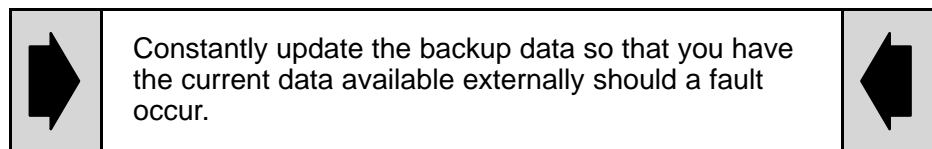
The memory capacity of the hard disk on the MMC–CPU module enables you to store large volumes of data in the SINUMERIK 840C. It is advisable, however, to perform a data dump of all relevant data on an external data device (programmer, PC, etc.) because the data will be lost irretrievably if the MMC–CPU (which incorporates the hard disk) develops a defect.

### *Archive list*

As a backup aid, the SINUMERIK 840C allows directories and files to be archived in archive lists, i.e. you can store a list of all important files and directories under a name that you specify and output this complete to an external device (see Services section, “Creating archive lists”). The SINUMERIK 840C has various hardware interfaces which can be addressed by the software.

### *Output via RS232C/V24*

The RS232C (V24)/20mA interface on the MMC–CPU enables you to output your user data from the hard disk in PC format or in punched tape format (see Services section, “Data input-output”). The interface automatically changes over between RS232C (V24) and 20 mA mode, depending on the connected device and corresponding cable.



Via the RS232C (V24) interface, you can also print out data directly with a printer (SERVICES/DATA IN–OUT/PRINT function).

The SINUMERIK FD–E2 diskette unit (option) can be used for backing up data.

### *Computer link*

Optionally you can implement a computer link with the SINUMERIK 840C (only in the NCK area) by using link modules.

## 1.8 Help displays

You can call up help displays in the different operating areas of the SINUMERIK 840C. The following is offered:



1. With this key you activate a help display for a special operation. The possibility is indicated by the “i” in the 1st (left) softkey.



Press RECALL to deselect this help screen.



2. By pressing SHIFT + i you select or deselect a help screen irrespective of an operator action, e.g. help screen for the data selector:

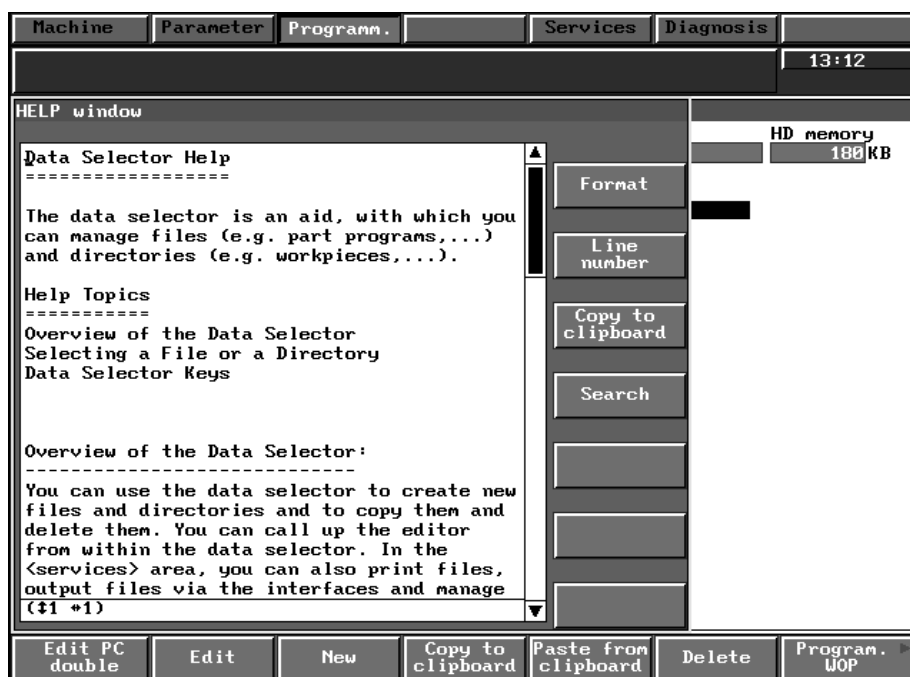


Fig. 1.3 Help display

END OF SECTION



## 2 Operator Interface

### 2.1 Operator panel

Different operator panels can be connected to the SINUMERIK 840C. The following components can be added: machine control panel, hand-held unit and PC standard keyboard (MF–2).

- 19" operating panel with 14" colour monitor  
(2x7 softkeys, switchover keys)
- 19" NC full keyboard  
(alphabetic group of keys, numeric group of keys, cursor group of keys)
- 19" slimline operator panel with 9.5" monochrome LC display including NC full keyboard  
(2x7 softkeys, switchover keys, alphabetic group of keys, numeric group of keys, cursor group of keys)
- 19" slimline operator panel with 10" or 9.5" TFT display including NC full keyboard  
(2x7 softkeys, switchover keys, alphabetic group of keys, numeric group of keys, cursor group of keys)
- PC standard keyboard (MF–2)
- 19" machine control panel, M version
- 19" machine control panel, T version
- Hand-held unit

## 2.2 Operating elements

### 2.2.1 Operator panel

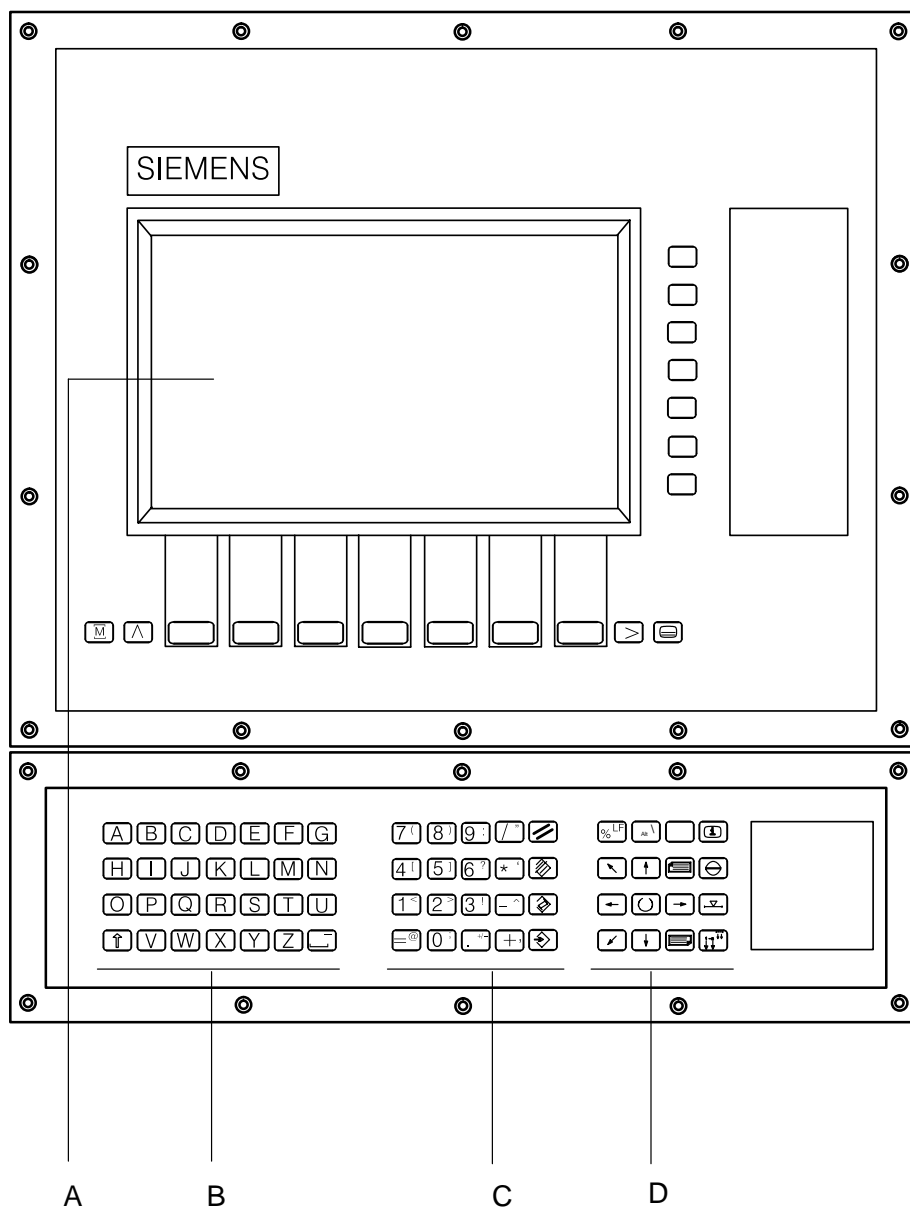


Fig. 2.1 19" operator panel with 14" colour monitor

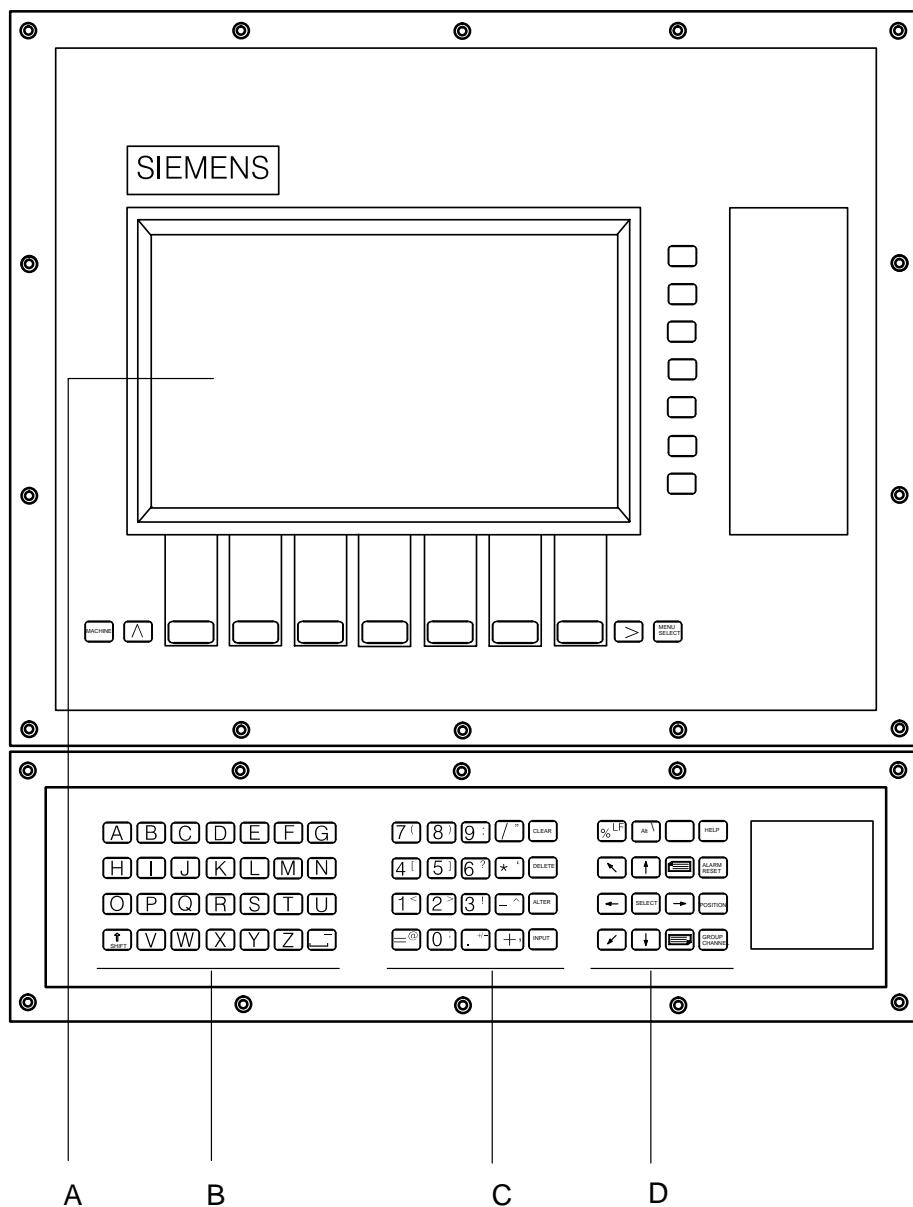


Fig. 2.2 19" operator panel with 14" colour monitor in US layout

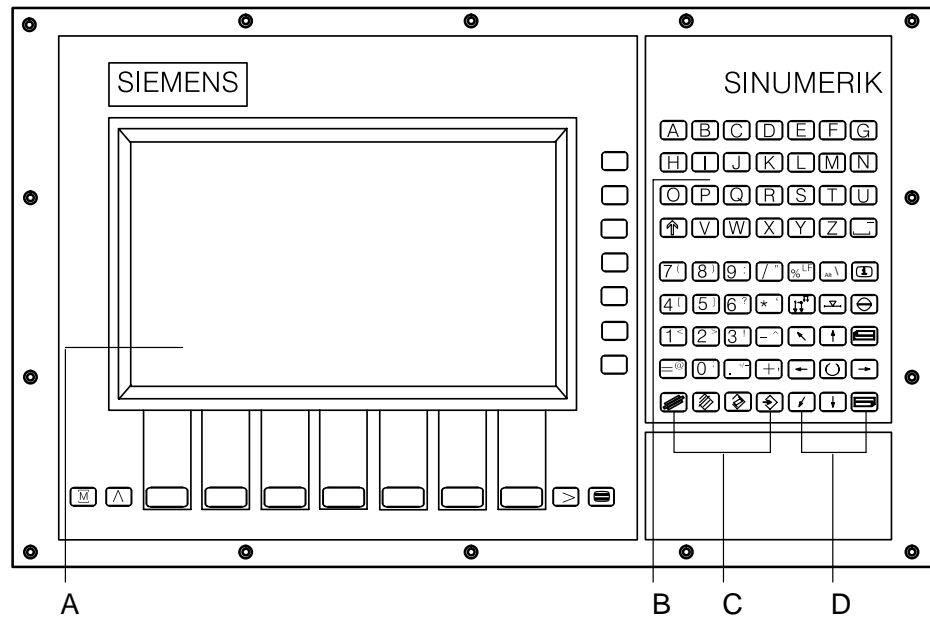


Fig. 2.3 SINUMERIK slimline operator panel

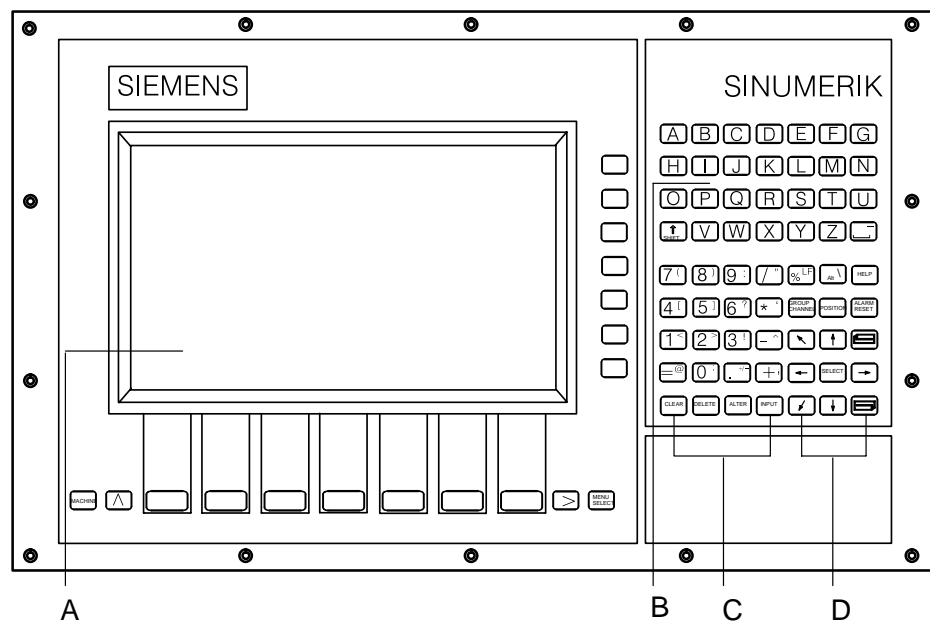


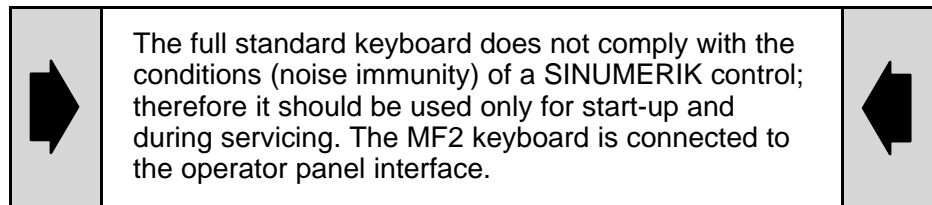
Fig. 2.4 SINUMERIK slimline operator panel in US layout

- A: 14"/10"/9.5" colour graphics monitor  
14 softkeys  
ETC key, Recall key  
Machine area key  
Area switchover key
- B: Alphabetic group of keys
- C: Numeric group of keys with editing and input keys
- D: Cursor group of keys with control keyboard

### 2.2.2 PC standard keyboard (MF–2)

An MF2 full standard keyboard can be connected. The machine control panel functions, however, cannot be activated via this keyboard.

A set of key caps is supplied with the standard MF2 keyboard. These key caps correspond to the symbols on the operating panel and can be mounted on the standard keyboard as shown in the drawing.



## 2.2.2 PC standard keyboard (MF-2)

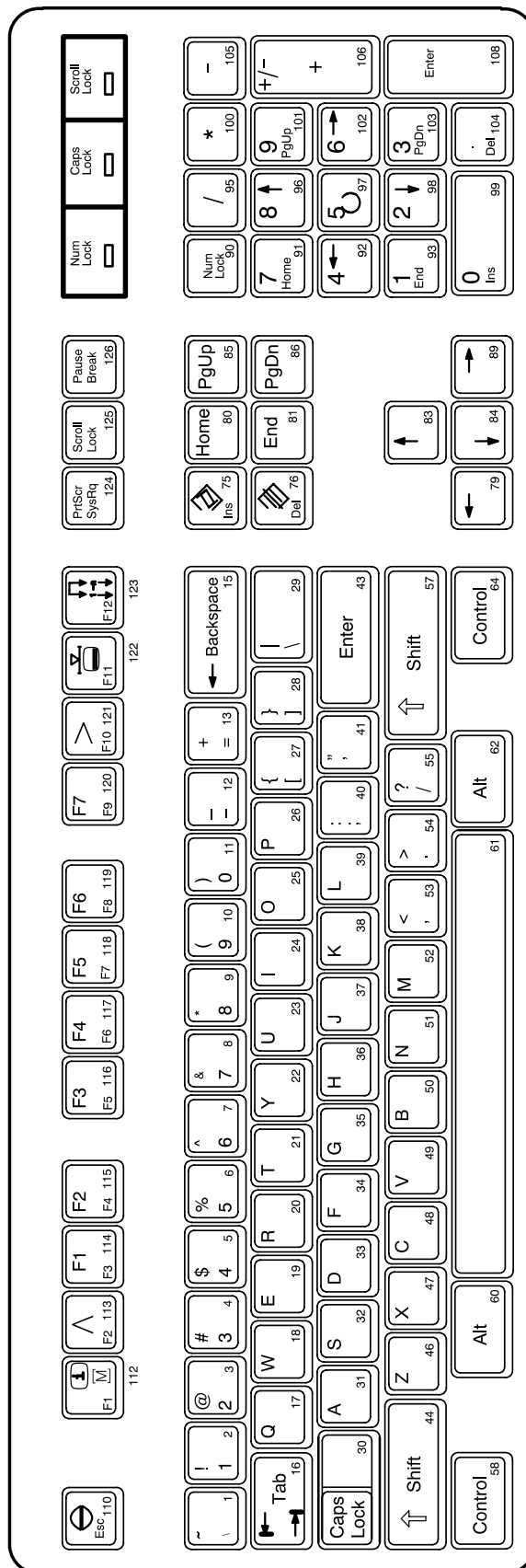


Fig. 2.5 Full keyboard

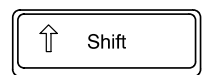
### 2.2.3 Operating elements of the operator panel keyboard/full keyboard



Softkey: Key which is assigned a variable function via a menu bar on the screen.

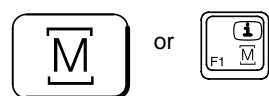
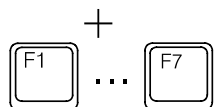


Horizontal softkey bar (full keyboard)



Vertical softkey bar (full keyboard)

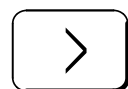
The designation F1 to F7 corresponds to the designation on the additional key caps of the standard keyboard (see also Figure "Full keyboard").



Machine area



RECALL key: Return to the higher level menu



ETC key: Extension of the softkey bar in the same menu



<sup>1)</sup> Area switchover

With this key, you can switch over to the area menu bar from every area. When an area is deactivated the control "remembers" it. It is activated again by pressing the area switchover key twice.



<sup>1)</sup> Shift: Switching over keys with double allocation; not self-retaining, i.e. two keys must be activated.



Lower case letter a ... z, activated with SHIFT

Upper case letter A ... Z

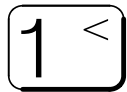


Underscore

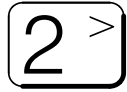
Blank, space

<sup>1)</sup> US layout

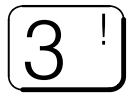
## 2.2.3 Operator panel keyboard / full keyboard



Special character: less than  
Digit 1



Special character: greater than  
Digit 2



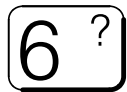
Exclamation mark  
Digit 3



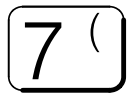
Square bracket  
Digit 4



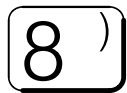
Square bracket  
Digit 5



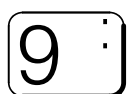
Question mark  
Digit 6



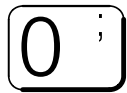
Round bracket  
Digit 7



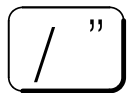
Round bracket  
Digit 8



Colon  
Digit 9



Semicolon  
Digit 0



Inverted commas  
Oblique stroke, division

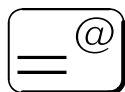


Apostrophe  
Multiplication

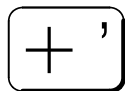




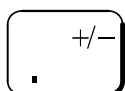
Special character  
Hyphen, subtraction



Special character  
Equal sign

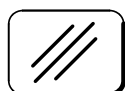


Comma  
Plus, addition



Change of sign  
Fullstop (point/period)

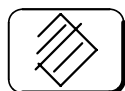
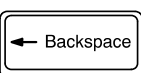
or



CLEAR

<sup>1)</sup> Delete character/entry

or



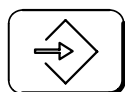
DELETE

<sup>1)</sup> Delete word



ALTER

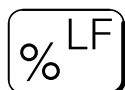
<sup>1)</sup> Edit word



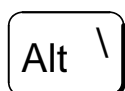
INPUT

<sup>1)</sup> Confirm input (store an edited value in the memory)

or



Line Feed; identifier for end of block  
Percent sign; identifier for beginning of main program



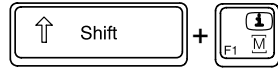
Backslash  
Alt

<sup>1)</sup> US layout

## 2.2.3 Operator panel keyboard / full keyboard

*Help*

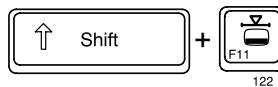
or



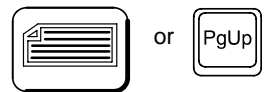
- 1) This key can be used to call up explanations and information on the current operating state. Pressing it a second time brings you back to the previous display (but not in the graphic programming system). The "i" in the softkey bar indicates the possibility of calling up information by pressing the "Help" key.

*Actual position in large characters*

or



- 1) When you operate this key, the CRT display of "Actual position" is shown in double-height characters. Operating the key again brings you back to the previous full CRT display (with standard size characters).

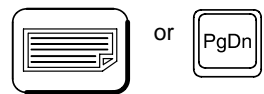


or



You "page" up by one display.

You change the current screen display by operating one of these keys if further screen displays of the same layout (lists) are available.



or



You "page" down by one display. In a part program you can page down the display (towards the end of the program), or up (towards the beginning of the program).



or



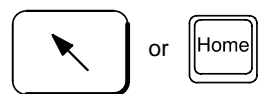
With these keys, you move the cursor on the screen to the left or to the right, upwards (back) or downwards (forward).



or



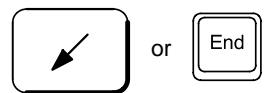
With the cursor keys, you can move the cursor in the input fields (toggle fields, etc.) or in the various editors (ASCII editor; DIN editor). Further information is given in the respective Sections of this Operator's Guide.



or



Change to another input window



or

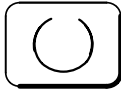

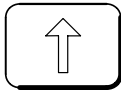

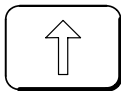



Activate/deactivate cursor in the input line



With this key you activate the cursor in the input line (NCK only), move it by means of the cursor keys, in order to correct or insert. When operating the key again, you deactivate the cursor.

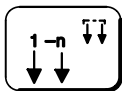

---

1) US layout

-   <sup>1)</sup> Selection key/Search key  
(toggle fields select/mark text in editor/search text)
-   SHIFT + selection key  
(reset a toggle field)
-   SHIFT + ALTER  
Copy the contents of the screen into a file (hardcopy!)  
You will find the TIF file "HCFn" (n = 0, 1, ...) in directory "Services/  
Screen shots"  
Memory requirement on the hard disk:  
mono approx. 300 KB / screen  
color (compressed) approx. 20 kB / screen



### Acknowledge alarm

-   <sup>1)</sup> By pressing this key you acknowledge the information from the NC monitoring system displayed in the alarm line such as:
- Alarm text and
  - Alarm number for CANCEL alarms
    - The machine manufacturer can configure whether this key is to act on all channel-specific CANCEL alarms or only on the channel-specific CANCEL alarms of the current channel.

-   <sup>1)</sup> Change of mode groups  
Channel switchover

### Change of mode groups

The NC channels are grouped together in mode groups. With the SINUMERIK 840C up to 6 mode groups can be selected.

	Please note: The channels are assigned to mode groups by the machine manufacturer.	
---	---	---

### Channel switchover

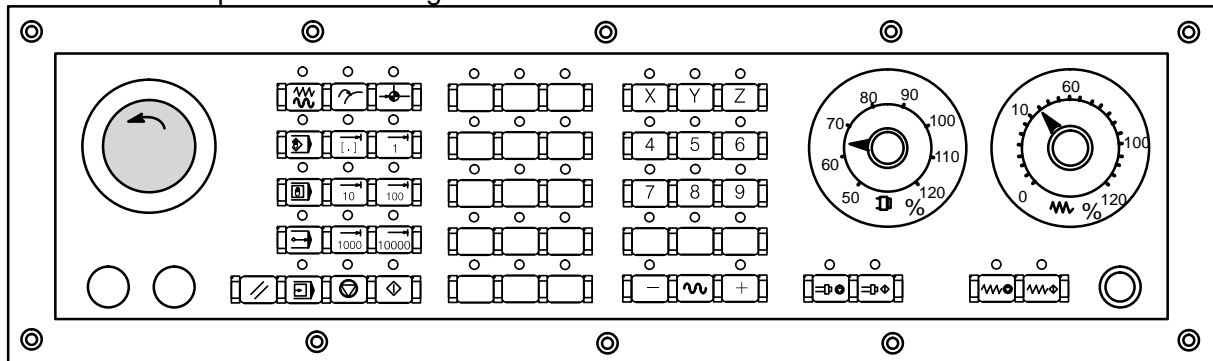
- Pressing this key once switches to the next higher channel number, referred to the number displayed in the channel status field.
- Pressing the key again switches on to the next channel or back.
- The channel can be selected directly by entering the channel number and then operating the key.
- The NC area of the SINUMERIK 840C is divided into a maximum of six channels.

<sup>1)</sup> US layout

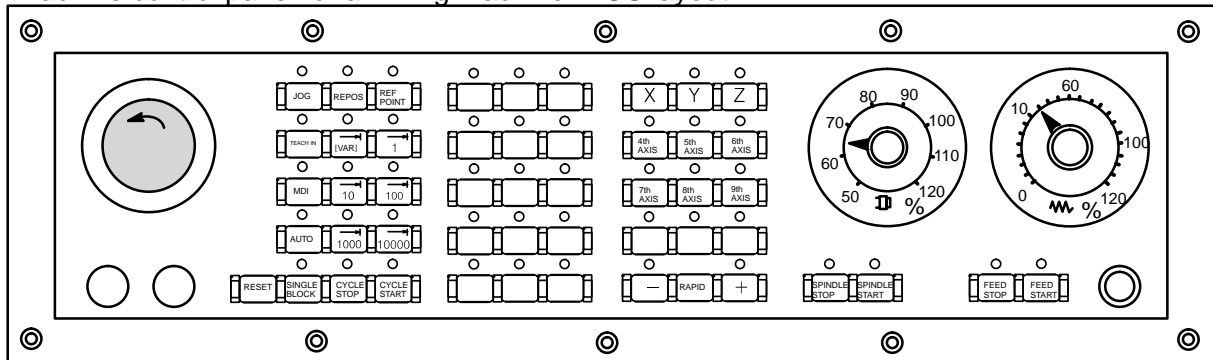
## 2.2.4 The operating elements of the machine control panel

## 2.2.4 The operating elements of the machine control panel

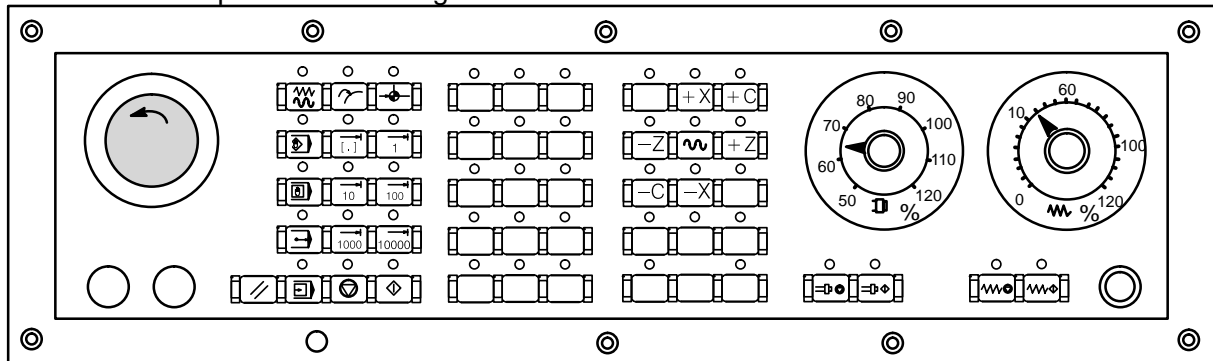
Machine control panel for a milling machine



Machine control panel for a milling machine in US layout



Machine control panel for a turning machine



Machine control panel for a turning machine in US layout

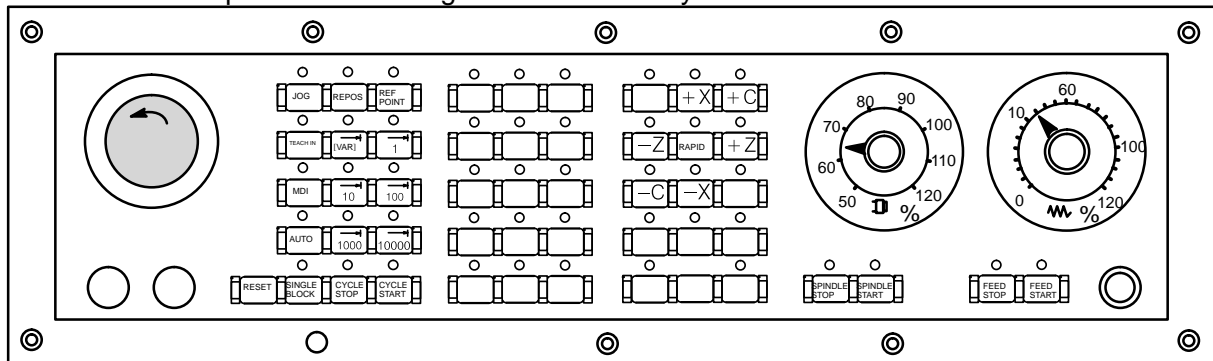


Fig. 2.6 Machine control panels

## 2.2.4 The operating elements of the machine control panel

Machine tool operations such as traversing of the axes or program start can only be triggered via a machine control panel.

The machine tool can be equipped with a standard Siemens machine control panel or with a special machine control panel from the machine tool manufacturer.

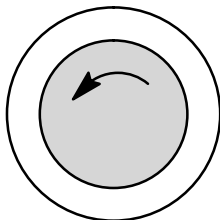
A maximum of two mode groups is possible.

The standard Siemens machine control panel is described. Should another machine control panel be used, please refer to the Operator's Guide of the machine tool manufacturer.

The standard Siemens machine control panel has the following operating elements:

- Emergency stop button
- Operating modes with function keys
- Spindle control
- Feed control
- Direction keys with rapid override
- Keyswitch
- Reset key
- Program control



#### *Emergency stop button*



You operate the red button in emergency situations:

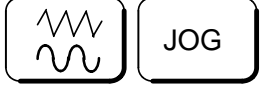



- When there is a danger to life.
- When there is a danger that the machine or the workpiece could be damaged.

Operation of the "Emergency stop" button generally brings all drives to a stop with maximum braking torque.

	For further or other reactions to "Emergency stop", refer to the machine tool manufacturer's documentation.	
---	---	---

## 2.2.4 The operating elements of the machine control panel



*Operating modes*

Key symbol	Definition	Designation of operating mode
 <sup>1)</sup>	The axes are traversed continuously in JOG mode using the direction keys or incrementally using the direction keys or the handwheel.	<b>JOG</b> Set-up ( <b>J</b> ogging)
 <sup>1)</sup>	Interactive creation of programs	<b>TEACH IN</b>
 <sup>1)</sup>	The machine is controlled by executing a block or a series of blocks. The operator panel is used to input blocks.	<b>MDA</b> Manual <b>D</b> ata <b>A</b> utomatic
 <sup>1)</sup>	The machine is controlled by automatic execution of programs.	<b>AUTOMATIC</b>

If a mode key is pressed, the corresponding mode is selected and all other modes and functions are cancelled.

The active mode is indicated and confirmed by the associated LED lighting up.

*“REPOS” and “Approach reference point” functions*

Key symbol	Definition	Designation of function
 <sup>1)</sup>	Repositioning, approach contour again in <b>JOG</b> mode	<b>REPOS</b> Reposition
 <sup>1)</sup>	Approach reference point in <b>JOG</b> mode	<b>REFPOINT</b> Approach reference point

The “REPOS” and “Approach reference point” functions are only active in JOG mode and can be selected only in JOG mode.


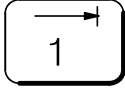
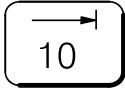
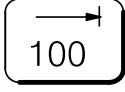
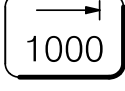
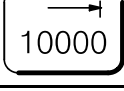
The active function is indicated by the associated LED lighting up.

The function can be cancelled by pressing the function key again. The function is cancelled automatically if there is a change of operating mode.

---

<sup>1)</sup> US layout

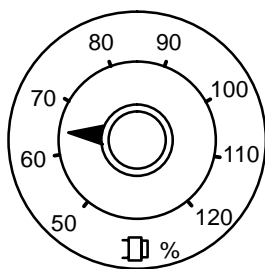
*INC function*

Key symbol	Definition	Designation of function
	Increment mode with variable step size (setting data). <sup>1)</sup>	<b>INC VAR</b> Incremental Feed variable
	Increment mode with fixed step size of 1 increment	<b>INC</b> Incremental Feed
	Increment mode with fixed step size of 10 increments.	
	Increment mode with fixed step size of 100 increments.	
	Increment mode with fixed step size of 1000 increments.	
	Increment mode with fixed step size of 10 000 increments.	

The increment size depends on the display resolution that has been set.

The INC functions can be activated in conjunction with the following modes:

- JOG mode
- TEACH IN mode.

*Spindle override switch*



- The rotary switch with 16 notched positions (of which 15 are used) enables you to reduce or increase the programmed spindle speed S (corresponds to 100%).
- A machine data determines whether the switch function is active, i.e. it is determined by the machine manufacturer.
- The set spindle speed value S is displayed as an absolute value in % on the CRT display.

Control: 50% to 120% of the programmed spindle speed

Step size: 5% from position to position

<sup>1)</sup> US layout

## 2.2.4 The operating elements of the machine control panel

	The given step size and the control range are valid for standard machine data (MD). These can be altered by the machine tool manufacturer to suit a specific application.	
---	---	---

*Spindle stop*

When you press the SPINDLE STOP key:



- <sup>1)</sup>
- The spindle speed is reduced down to zero
  - The associated LED lights up as soon as SPINDLE STOP is accepted by the control.

Example for use of SPINDLE STOP



- To effect tool change
- To enter S, T, H, M functions while setting up (overstoring)

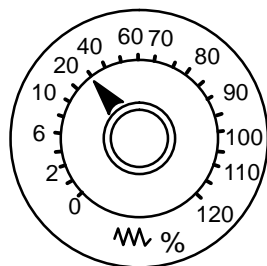
*Spindle start*

When you press the “Spindle start” key:



- <sup>1)</sup>
- The spindle is enabled. If the spindle has been stopped with SPINDLE STOP in a program, it can be started again with SPINDLE START.
  - The associated LED lights up as soon as SPINDLE START is accepted by the control.

	<p>The following values are specified by machine data or setting data:</p> <ul style="list-style-type: none"> <li>• The maximum spindle speed</li> <li>• The values for spindle override positions (refer to the machine tool manufacturer's documentation)</li> </ul>	
---	--	---

*Feedrate control***Feedrate/rapid override**

The rotary switch with 23 notched positions allows you to reduce or increase the programmed feedrate value F (corresponds to 100%).



The set feedrate value F is indicated on the CRT display in %.

Control range: 0% to 120% of the programmed feedrate.  
The 100% value is not exceeded in rapid traverse.

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

<sup>1)</sup> US layout



	<p>The given increment sizes and the control range are valid for standard machine data (MD) These can be altered by the machine tool manufacturer to suit a specific application.</p>	
---	---	---

**Feed stop**

When you operate the FEED STOP key:

- 1) • The program being executed is stopped
- The feed drives are brought to a controlled stop
- The associated LED lights up as soon as FEED STOP is accepted by the control.



Examples for use of FEED STOP:

- During application in MDA mode, a block with a fault is discovered
- To effect tool change.

**Feed start**

When you operate the FEED START key:

- 1) • The part program continues in the current block
- The feedrate is increased to the value specified by the program
- The associated LED lights up as soon as FEED START is accepted by the control.

	<p><b>The following is configured by the machine manufacturer:</b></p> <ul style="list-style-type: none"> <li>• The speed and rapid traverse rates</li> <li>• The values for feedrate and override positions</li> <li>• Whether the feedrate override switch is also active for rapid traverse</li> <li>• The axis names (see machine manufacturer's specifications)</li> </ul>	
---	---	---

**Direction keys, turning machine**

You traverse the axis marked X.



You traverse the axis marked Z.



You traverse the axis marked C.

1) US layout

## 2.2.4 The operating elements of the machine control panel

*Rapid traverse override*

<sup>1)</sup> When you operate this key at the same time as any of the keys above, the axis is traversed in rapid traverse mode.

*Direction keys, milling machine*

You select the axis marked X, Y, Z etc.



You traverse the selected axis in positive or in negative direction.

You traverse other assigned axes in the same way.

If an INC function has been set and the direction key is pressed (whether for a long or a short time), the axis traverses by only one step (1/10/100/1000/10000 increments depending on the setting).

If an INC function is not selected, the default setting “continuous” is active. The axis traverses as long as the direction key is pressed.

Please take into consideration that when the safety interlocks are enabled only the simple traverse movement via the JOG keys or via the handwheel is permitted.

*Rapid traverse override*





<sup>1)</sup> When you operate this key at the same time as any of the keys above, the axis is traversed in rapid traverse mode.



---

<sup>1)</sup> US layout

**Keyswitch**

The SINUMERIK 840C keyswitch has 4 positions which are assigned with functions via the control's operating system. Three keys of different colours belong to the keyswitch and these can be turned and removed from the following positions:

Position of switch	Withdraw position	Function
Position 0 	–	Data on the hard disk are only displayed
Position 1 	0 + 1 key 1 black	Generate, edit and delete workpiece data
Position 2 	0 + 1 + 2 key 2 green	Generate, edit and delete interface data, archive lists and setting data
Position 3 	0+1+2+3 key 3 red	Siemens MMC standard data is loaded into the system when powering up. All data in the directory is password-protected. Note: Use only for start-up and servicing!

	The key positions can be assigned with additional functions by the machine manufacturer. Please read the machine manufacturer's Operator's Guide.	
---	---	---

**Note**

If the PLC is in the STOP state the input display on the machine control panel is not scanned. In this case the control can be powered up via keyswitch position 3.



**Reset key**

<sup>1)</sup> When you operate the "Reset" key:

- Execution of the current part program is interrupted if the key triggers a mode group Reset (= Reset of all the channels of the mode group).
- Messages are cleared from the monitoring system unless they are POWER ON or acknowledgment alarms.
- The control is switched to the "Reset" state, i.e.
  - The NC control remains synchronized with the machine.
  - All buffer and user memories are cleared (but the contents of the part program memory are retained).
  - The control is in the Reset state and ready for a new program run.

1) US layout

## 2.2.4 The operating elements of the machine control panel

	The machine manufacturer can configure the RESET key such that it acts on channel/mode group or on the whole NC.	
---	--	---

*Single block*

This function allows you to execute a part program on a block-by-block basis.<sup>1)</sup>

The SINGLE BLOCK function can be activated in the AUTOMATIC, TEACH IN and MDA modes.

When SINGLE BLOCK operation is active:

- The SBL (**S**ingle **B**lock) message is shown on the CRT display
- The current block of the part program is executed only when you press the “NC start” key
- When the current block has been executed, processing is stopped
- The following block can be executed by pressing the “NC start” key again.

The function does not work with calculation blocks. Calculation blocks are part program blocks which execute programmed calculations (R parameters calculation operations), but do not output anything to the machine or to the PLC.

If SINGLE BLOCK is activated, the corresponding LED lights up on the machine control panel.

The function can be deselected by pressing the key again.



*NC Stop*

If you press the NC STOP key, processing of the current part program is interrupted. All axis movements are brought to a controlled stop. The reactions of the H and M functions in the case of NC STOP are configured by the machine manufacturer. For more information refer to the machine manufacturer's documentation. The associated LED lights up.

Processing can then be continued by pressing NC START.



*NC Start*

If you press the NC START key, the part program called is started at the current block; the associated LED lights up. Processing of a part program interrupted with NC STOP is continued at the point of interruption on pressing NC START.<sup>1)</sup>

	The axis positions programmed in the current block are approached with linear interpolation after the “NC START” key has been activated. <b>Caution:</b> Risk of collision!	
---	--	---

<sup>1)</sup> US layout

## 2.2.5 Operating elements on the hand-held unit

	<p>The display and key functions can be configured by the machine manufacturer. For further details refer to the machine manufacturer's documentation. The standard keyboard is described below.</p>	
---	--	---

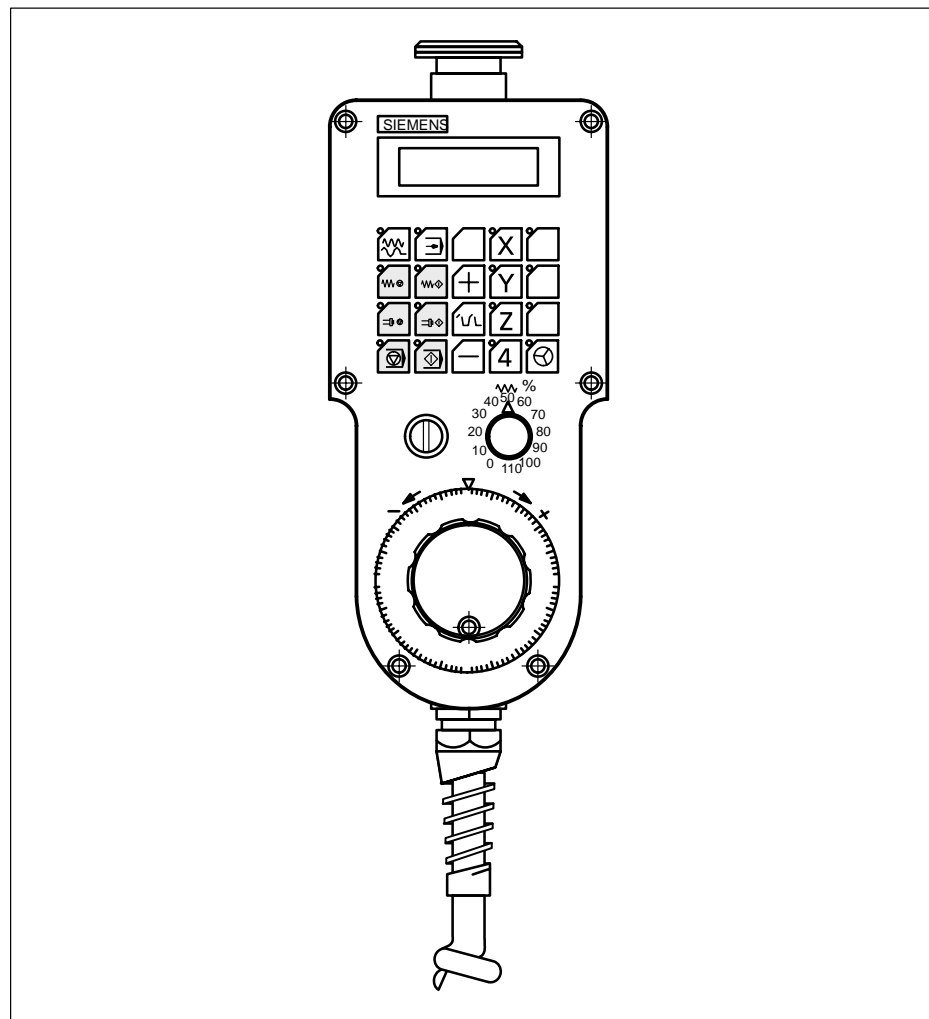




Fig. 2.7 Operating elements of the hand-held unit

## 2.2.5 Operating elements on the hand-held unit

*Operating and display elements*

- 20 user-assignable keys; symbol labels can be replaced (hand-held unit must be opened to access the labels).
- 1 rapid traverse/feedrate override switch
- 2 enabling devices
- 1 keyswitch
- 1 emergency STOP pushbutton
- 1 electronic handwheel
- 16 user-controllable LEDs
- 1 two-line numerical display (2 x 16 characters)

**2.2.6 Operating elements on the A-MPC hand-held unit***Operating modes*

Key symbol	Definition	Operating mode
	Jogging the machine by a continuous movement of the axes using the direction keys or by incremental movement of the axes using the direction keys or the hand-held unit	<b>JOG</b> set-up (Jogging)
	Creating programs interactively with the machine	<b>TEACH IN</b>

Pressing a particular operating mode key selects the corresponding operating mode, all other operating modes and functions are deselected.

The LED for the selected operating mode lights up to indicate and confirm that the operating mode is active.

*Feed stop*

When you press the FEED STOP key:

- Processing of the current program is stopped
- The feed drives are stopped under control
- The corresponding LED lights up as soon as FEED STOP has been accepted by the control.



Example applications of FEED STOP:

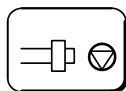
- An error is detected while running a program in MDA mode
- To carry out a tool change.

**Feed start**

Press the key FEED START:

- The part program is continued in the current block
- The feedrate is increased to the value defined in the program
- The corresponding LED lights up as soon as FEED START has been accepted by the control.

	<p><b>The machine manufacturer configures the following:</b></p> <ul style="list-style-type: none"> <li>• The feed and rapid traverse velocities</li> <li>• The values for the feedrate override settings</li> <li>• Whether the feedrate override switch is also active for rapid traverse</li> <li>• The axis names (see information supplied by machine-tool manufacturer)</li> </ul>	
---	--	---

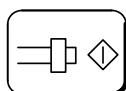
**Spindle stop**

Press the SPINDLE STOP key:

- The spindle speed is reduced to zero speed
- The corresponding LED lights up as soon as SPINDLE STOP is accepted by the control.



Example application of SPINDLE STOP

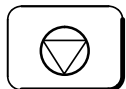
- To carry out a tool change
- To enter S, T, H and M functions during set-up (overstore)

**Spindle start**

Press the SPINDLE START key:

- The spindle is enabled. If the spindle has been stopped with SPINDLE STOP in a program, it can be started again with SPINDLE START.
- The corresponding LED lights up when "Spindle start" is accepted by the control.

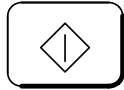
	<p>The following is defined in the machine data or setting data:</p> <ul style="list-style-type: none"> <li>• The maximum spindle speed</li> <li>• The values for the spindle speed override settings (see information supplied by the machine-tool manufacturer)</li> </ul>	
---	--	---

**NC Stop**

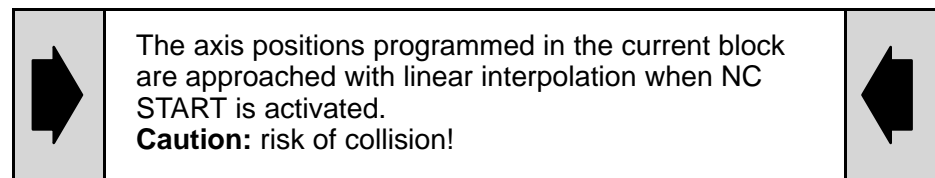
If you press the NC STOP key processing of the current part program is interrupted. All axis movements are brought to a controlled stop. H and M function responses on NC STOP are configured by the machine-tool manufacturer. Please refer to the documentation supplied by

## 2.2.6 Operating elements on the A-MPC hand-held unit

the machine-tool manufacturer. The corresponding LED lights up. You can then continue machining with NC START.

*NC Start*

Press the key NC START to start the part program you have called at the current block; the corresponding LED lights up. After interrupting processing of a part program with NC STOP, processing is continued with NC START at the point of interruption.



You select the axis marked X, Y, Z etc.



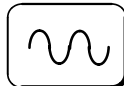
You traverse the selected axis in positive or in negative direction.

You traverse other assigned axes in the same way.

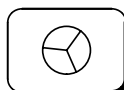
If an INC function has been set and the direction key is pressed (whether for a long or a short time), the axis traverses by only one step (1/10/100/1000/10000 increments depending on the setting).

If an INC function is not selected, the default setting “continuous” is active. The axis traverses as long as the direction key is pressed.

Please take into consideration that when the safety interlocks are enabled only the simple traverse movement via the JOG keys or via the handwheel is permitted.

*Rapid traverse overlay*

When you operate this key at the same time as any of the keys above, the axis is traversed in rapid traverse mode.



*Handwheel active*

This handwheel function must be configured by the machine-tool manufacturer.



*Keyswitch*

The SINUMERIK 840C keyswitch on the hand-held unit has 4 positions which can be assigned functions by the machine manufacturer. Three keys of different colours belong to the keyswitch which can be turned and removed in the stated positions (see “Position of switch” and “Withdraw position” for the machine control panel!).

	The key positions can be assigned with functions by the machine manufacturer. Please read the machine manufacturer's Operator's Guide.	
---	--	---



*Note*

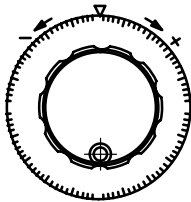
If the PLC is in the STOP state, the input display on the machine control panel is not scanned.

*Feed control*

Feedrate override



	The increment values displayed and the control range only apply to standard machine data. Please refer to the documentation supplied by the machine-tool manufacturer.	
---	--	---

*Electronic handwheel*

## 2.2.7 Screen layout

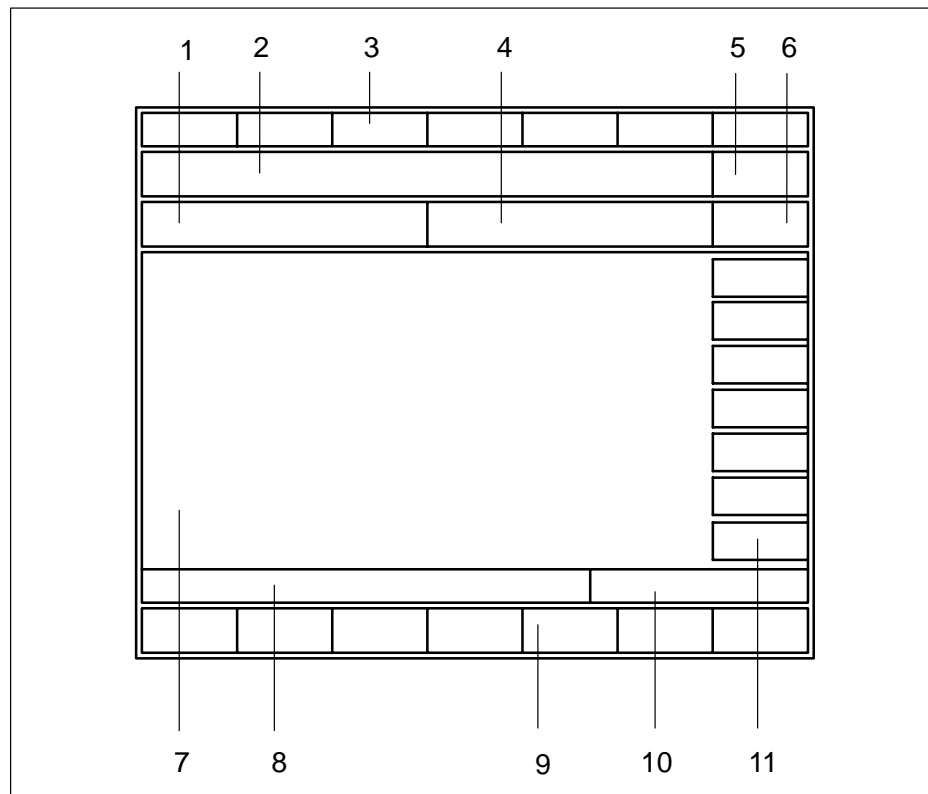


Fig. 2.8 Screen layout

### Meaning of screen boxes

- 1 – Mode, operating function, channel state (area-specific)
- 2 – Alarms, messages, comments
- 3 – Display of the selected area
- 4 – Channel status, program modification display (area-specific)
- 5 – Clock, clear condition for present alarm, channel-independent status display by means of icons
- 6 – Mode group and channel number (area-specific)
- 7 – Operating window, NC displays, texts, input keys
- 8 – Interactive text (area-specific)
- 9 – Softkey bar with 7 softkey functions
- 10 – Input (area-specific)
- 11 – Softkey bar with 7 softkey functions (area-specific)

END OF SECTION

## 3 Machine

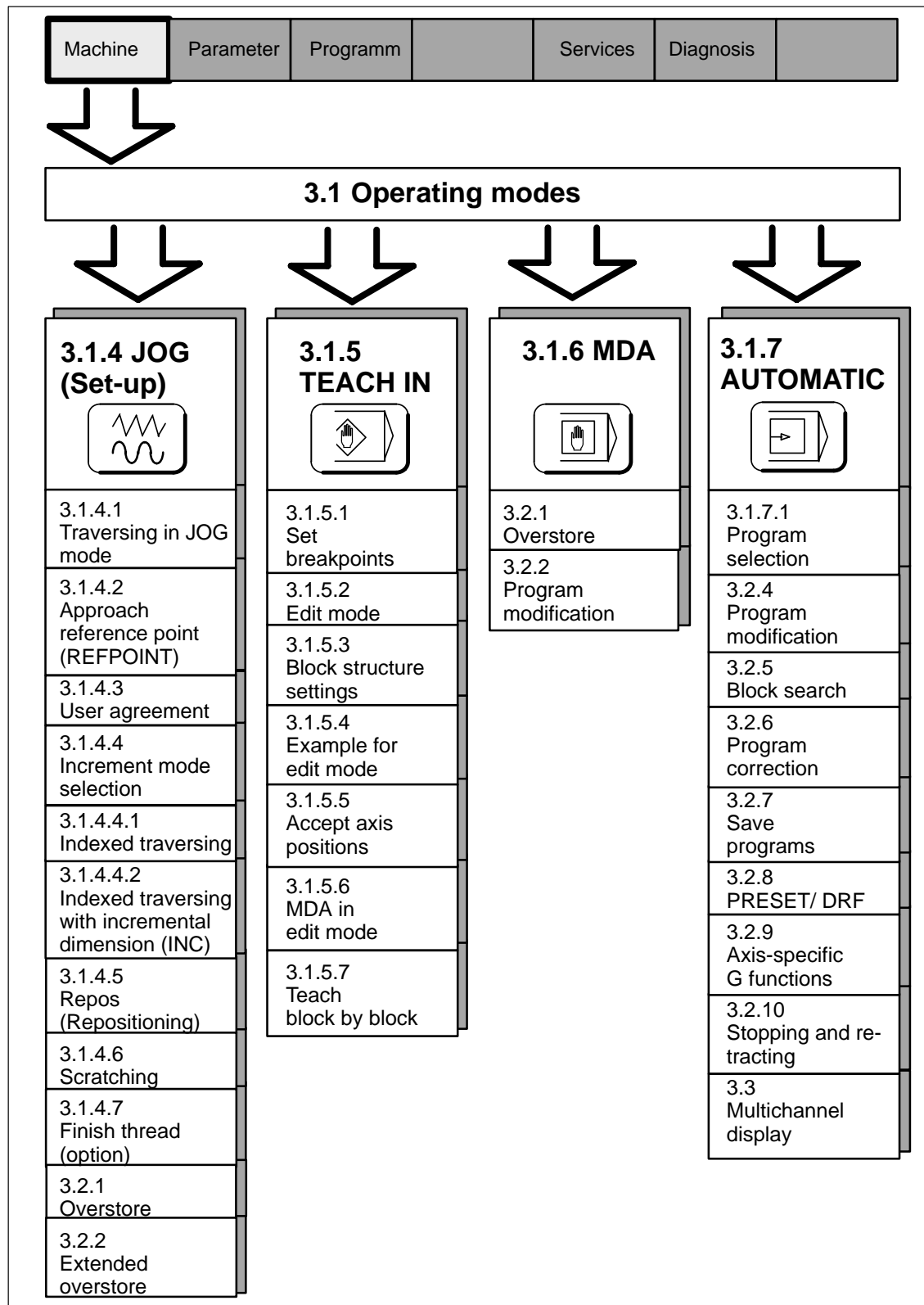
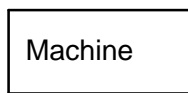


Fig. 3.1 Machine



If you activate the MACHINE softkey in the area menu bar



or the machine area key, you switch over to the machine area.  
With the machine area key you can return to machine operation at any time from any area.

## 3.1 Operating modes

The machine area includes all functions and influencing variables resulting in operations at the machine tool or detecting its state.

In the machine area there are four modes:

- JOG (set-up)
- TEACH IN,
- MDA (Manual Data Automatic)
- AUTOMATIC.

The mode can be selected via the machine control panel.

The operating mode can be changed at any time provided the machine and the control are switched on.

You select the mode separately for each mode group via the appropriate machine control panel.

The control has a maximum of six mode groups. When selecting the operating mode, it must be distinguished whether the machine is equipped with:

- One mode group and a related machine control panel
- Two mode groups and two machine control panels
- Two mode groups and one machine control panel.

### *One mode group – one machine control panel*



Select the desired operating mode by pressing the corresponding key on the machine control panel. The associated LED lights up.

### *Two mode groups – two machine control panels*



If you have two mode groups, and if each is assigned to a machine control panel, you select the operating mode by pressing the mode key on the appropriate machine control panel.

The selected operating mode is not displayed on the screen unless the relevant mode group has been selected.



### Six mode groups – one machine control panel

If you have several mode groups and one machine control panel, you must first assign the machine control panel to the desired mode group.

When the operating mode is changed, the mode group is automatically switched over for screen display.

Now select the operating mode for the preset mode group.






	<p>The machine tool manufacturer's guide provides information on:</p> <ul style="list-style-type: none"> <li>• Number of mode groups</li> <li>• Number of machine control panels</li> <li>• Assignment of machine control panels to the mode groups.</li> </ul>	
---	---	---



In each mode group you can select four operating modes JOG, TEACH IN, MDA and AUTOMATIC.

If you activate the operating mode key, the corresponding operating mode is selected, all other operating modes and functions are deselected.

The operating mode which is active is signalled and acknowledged by the associated LED.

Machine	Parameter	Programm.		Services	Diagnosis	
						12 : 48   
AUTOMATIC		Program reset				M.grp : 1 Channel : 1

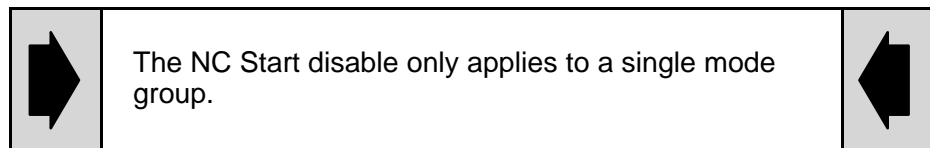
Display of the currently selected operating modes depends on the preset mode group

	<p>Switchover of the machine control panel is configured by the machine manufacturer. Please read the machine manufacturer's specifications.</p>	
---	--	---

### 3.1.1 Operating states – Operating state changes on change of operating mode

At the present time, each operating mode has one of the following operating states:

<i>Program reset</i>	The machine is in the reset state, e.g. after switching on or after end of program (M30).
<i>Program running (not in <b>JOG</b> mode)</i>	A program has been started and program execution is running.
<i>Program stop</i>	A program in execution has been interrupted.
<i>NC Stop</i>	A program in execution is interrupted. All state changes which occur during NC Stop (drift of one axis in the follow-up mode or traversing an axis in another channel) are taken into consideration after NC Start.  Here, a program can be a main program, a subroutine, a cycle or a number of NC blocks (e.g. in MDA mode).
<i>NC Start disable</i>	NC START is disabled in the relevant mode group in certain error states, e.g. if machine data are incorrectly parameterized.



*Example* If, for example, channels 1 and 2 of mode group 1 are disabled for NC Start and channel 3 belongs to mode group 2, it is still possible to process a program in channel 3 with NC START.

If the mode changes, the operating state can also change.  
The following table shows how the operating states change on change of the operating mode.

## 3.1.1 Operating states – Operating state changes on change of operating mode

To From		AUTOMATIC			JOG		TEACH-IN			MDA		
		Pro-gram reset	Run-ning	Stop	Pro-gram reset	Stop	Pro-gram reset	Run-ning	Stop	Pro-gram reset	Run-ning	Stop
AUTO-MATIC	Pro-gram reset				X		X			X		
	Run-ning					X	X			X		
	Stop					X	X			X		
JOG	Pro-gram reset	X					X			X		
	Stop			X			X			X		
TEACH-IN	Pro-gram reset	X			X					X		
	Run-ning	X			X					X		
	Stop	X			X					X		
MDA	Pro-gram reset	X			X		X					
	Run-ning	X			X		X					
	Stop	X			X		X					



### 3.1.2 Machine functions – overview

The operating modes are supplemented by machine functions. The machine functions are technology and machine independent.

All machine functions are listed below. Depending on the operating state, you can select some of the machine functions in every operating mode. The table below shows which machine functions can be selected in which operating mode and in which operating state.






Operating mode Functions	AUTOMATIC			JOG		TEACH-IN			MDA		
	Pro-gram reset	Run-ning	Stop	Pro-gram reset	Stop	Pro-gram reset	Run-ning	Stop	Pro-gram reset	Run-ning	Stop
Correction block			X					X			
Program modification	X	X	X			X	X	X	X	X	X
Overstore	X		X	X	X	X		X	X		X
Block search	X					X					
Preset offset DRF	X										
Program selection by TEACH IN						X					
Approach reference point				X							
REPOS					X	X		X			
x = Function selectable											

### 3.1.3 Status displays

#### 3.1.3.1 Channel-independent status display by means of icons

*Icon*



Three icon fields are provided underneath the clock. Three of the 5 existing standard icons can be displayed. The following statuses can be displayed by means of icons:

- |   |   |  |
|---|---|--|
| 1 |    | RS232C(V24)/Data in-out, as soon as data exchange with the I/O devices has been activated.   |
| 2 |    | Override = 0%, if no traversing movement is active in any axis and the override is set to 0% on at least one machine control panel.  |
| 3 |    | Axis moves as soon as a traversing command is active.  |
| 4 |   | Program is executed when in one channel the signal "Program running" is active.  |
| 5 |  | Block search, program interrupt if the signal "Program running" is not active in any channel and if a program interrupt is initiated by means of STOP, operating mode change, or M00/M01 active, or if block search has been triggered, or if the last block has already been executed in single block mode. |

Icon 1 is only displayed in the 1st icon field

Icons 2 or 3 are displayed in the 2nd icon field

Icons 4 or 5 are displayed in the 3rd icon field

	Icons 2 to 5 must be configured by the machine manufacturer. See the machine manufacturer's documentation.	
---	--	---

### 3.1.3.2 Spindle utilization display

*"Utilization"*  
bar display

This display indicates what percentage of the leading spindle capacity (e.g. S1) is being utilized in the currently selected operating mode. It is output only when the leading spindle has a digital drive.

Act. value	Override	Position	Set value							
F = 0.00	70%		F = 0.00 M							
S1 = 0	85%	0.00	S1 = 0							
Utilization (%)	<div></div>									

*Note*

Name of bar display

- SW 5.2 and lower: Power
- SW 5.4 and higher: Utilization

Configuration of bar graphs

- Digital drive  
Actual speed, torque, etc. can be read and displayed with WS800A
- Analog drive  
Configuration possible

### 3.1.4 JOG: Set-up mode



After switching on the control, the basic display of the JOG mode appears, or you select JOG with this key, provided the control has not been configured otherwise by the machine manufacturer.

Using the direction keys or the handwheels, you can traverse the axes of the machine in JOG mode.

It is possible to set the effect of the direction keys or the handwheels by way of the INC function. The following settings are possible:

INC VAR, INC1, INC10, INC 100, INC1000, INC10000. (handwheel MAX. INC100).

The feedrate is set by machine data. A feedrate override (in the range 0% to 120%) is only possible if certain interface signals are transmitted from the PLC to the NC.

After a program interruption, the distance of the point of interruption is displayed as "REPOS offset".

You traverse to the point of interruption until the REPOS offset shows zero.

Machine	Parameter	Programm.		Services	Diagnosis	
						11:09
JOG						Mode Gp 1 Channel 1
Program stop						
Actual value		REPOS offset		Program pointer		
X	-307.196		0.000	<div></div>		
Y	-89.600		0.000			
Z	-250.000		0.000			
Act. value		Override	Position	Set value		Auxiliary functions
F=	0.00	120%		F=	0.00M	T = M =
S 1=	0	115%	0.00	S 1=	0	D = M = H = M =
						G functions
Overstore	INC variable					

Fig. 3.2 Basic display in JOG mode

### 3.1.4.1 Traversing in JOG mode

#### Preconditions

- JOG mode (basic display) has been selected.
- Feed stop (FST) must not be active.  
Feed stop can be active, for example, when
  - there is no servo enable (see machine manufacturer's documentation)
  - the feedrate override switch is in the 0% position.

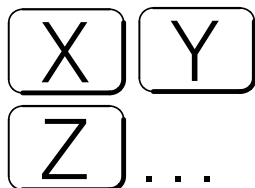
#### Operating sequence

##### 840C machine control panel for turning machine



Press the direction keys to traverse the axes.

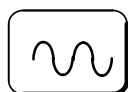
##### 840C machine control panel for milling machine





Select the axes.



Press these keys to traverse the selected axis in the respective direction.



If you press the key RAPID OVERRIDE as well, you traverse the selected axes at rapid traverse speed.

	The axis speed and rapid traverse speed are specified by the machine manufacturer. The axis speed override can also be effective for rapid traverse in the range of 0% to 100%.	
---	---	---



### 3.1.4.2 Approach reference point (REFPOINT)

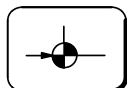
After switching on the control, you approach the reference points for the individual axes. In this manner, the control is synchronized with the machine (this does not apply for machines with absolute encoders).

#### Preconditions

- The machine area has been selected.
- JOG mode has been selected.
- The mode group and channel have been preset.

#### Operating sequence

	The machine manufacturer can change the reference point approach by means of the configuration. In this case, the following description does not apply.	
---	---	---



In JOG mode, “Approach reference point” may be selected by pressing the appropriate key.

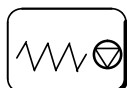
Now press the direction keys on the machine control panel. In the case of the machine control panel for a milling machine select the axis first.

The control checks the selected direction for traversing before the start:



If the wrong direction key has been pressed (e.g. + instead of –) the operation is not accepted and no movement takes place.

When the reference point has been approached, this is displayed on the screen for the appropriate axis.

The control is synchronized with the machine once the reference point has been approached for all axes.



Using the FEED HOLD key, you can stop the selected axis before the reference point has been reached.

	The position of the reference point and the traversing speed are set by the machine tool manufacturer by way of machine data.	
---	---	---

### 3.1.4.3 User agreement (“Safety Integrated” option)



User agreement can be implemented only when the SINUMERIK Safety Integrated option is installed. The user agreement function (protected via keyswitch) allows an appropriately authorized person to confirm that the currently displayed actual value of an axis tallies with the actual position on the machine. When the user agreement function is activated, the safe end positions (SE) and/or safe cams (SN) are active and safe.

User  
agreement

The display USER AGREEMENT can be selected in the JOG mode via softkey USER AGREEMENT.

#### Preconditions

- Keyswitch position 3
- “Axis referenced” state is reached
- Message 1340\*/300950 “Axis is not safely referenced” is active

	Please note the machine tool manufacturer's specifications with respect to referencing (using incremental/absolute encoder).	
---	--	---

#### Operating sequence

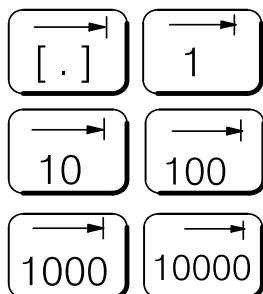
- Traverse the axis to a known position (e.g. visual mark).
- Compare this position to the NC actual value and the safety integrated actual value.
  - If the position tallies with the actual values, your agreement as the user can be given by means of a toggle softkey “Axis in position” “yes/no” in the User Agreement display.

The message 1340\*/300950 “Axis is not safely referenced” disappears. The axis is now safely referenced and the safe end positions (SE) and/or safe cams (SN) are active and safe.

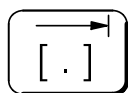
- If the position does not tally with the displayed actual values, then your agreement as the user cannot be given. The axis settings must be checked again.

### 3.1.4.4 Increment mode – selection

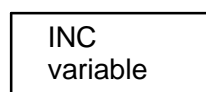
#### Operating sequence



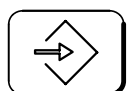
Before traversing the axes using the direction keys, select the appropriate incremental weighting on the associated machine control panel.



This key (INC variable) is used for traversing an increment previously set:



For setting the variable increment, activate this softkey and enter the value you require in the input field.



Press the INPUT key for entering the value in the “Incremental weighting for INC variable” window.

Use the direction keys to traverse the axes incrementally in JOG mode. In the case of milling machine controls, select the axis first.

You can reselect this function by pressing the active incremental weighting key; JOG mode is automatically effective.

#### Note

The selected increment function is displayed in the mode field.



The direction keys can act in two different ways:

- Modal
- Non-modal.

Modal means that the axis is always traversed by one increment (corresponding to the setting 1, 10, 100, 1000, 10000  $\mu\text{m}$ ) regardless of whether the key is pressed briefly or held down.

Non-modal means that the axis is traversed only as long as the key is held down and the set increment is reached. When the key is released, the traversing movement is stopped – even if the set increment has not yet been reached.

If the key is pressed again, the increment is traversed **anew**.



	Whether the increment is traversed modally or non-modally is determined by machine data.	
---	--	---

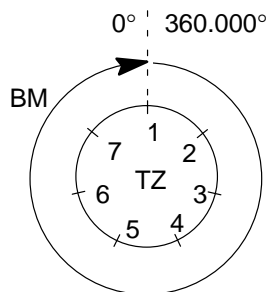


### 3.1.4.4.1 Indexed traversing

If this function is assigned to the axis to be moved and the direction key is pressed, the axis is moved as though it were a normal axis. After releasing the direction key, the axis moves to the next indexing position in the direction of traversing.

Using the “Indexed traversing” function, NC axes can be positioned at particular grid points.

	<p>The machine manufacturer uses machine data to assign the “Indexed traversing” function to individual axes. The parameters required are also preset by machine data. Please also refer to the machine manufacturer’s operating guide.</p>	
---	---	---



Example of “Indexed traversing” with a rotary axis:

Number of divisions: 7

Number of divisions per absolute dimension

Absolute dimension: 360.000° corresponds to the path to which the number of divisions refers.

### 3.1.4.4.2 Indexed traversing with incremental dimension (INC)

If the “Indexed traversing” function is assigned to the axis to be moved and one of the direction keys is pressed, the axis traverses by one division increment.

It is irrelevant which INC function (1, 10, 100, 1000 or 10000) is selected.

A rotary axis can be traversed by one division increment only by pressing the INC 1 or INC variable key and entering “1”. The INC10, INC100, INC1000 and INC10000 keys have no effect.

### 3.1.4.5 Repos (Repositioning)

After a program interrupt and changing from AUTOMATIC to JOG, it is possible to leave the contour in JOG mode.

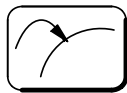
In this case, "AUTOMATIC" mode is **not** terminated; i.e. the control is not brought to the reset position by an automatic "RESET".

When leaving the contour, the paths covered are recorded by the control. The distance from the point of interruption is stored and displayed as a "REPOS **offset**".

#### Preconditions

- Program execution in AUTOMATIC mode has been interrupted.
- You travel away from the contour in JOG mode.

#### Operating sequence



Press the REPOS key on the machine control panel.



By pressing the appropriate direction keys on the machine control panel, you approach the point of interruption. On the machine control panel for a milling machine, select the axis first.



After the contour has been reached, select AUTOMATIC mode and press the NC START key.

#### Note

- If you want to reposition after a program interruption triggered by a RESET, you must first perform a BLOCK SEARCH:
  - Select AUTOMATIC mode
  - Softkey [ > ] [block search]  
[accept progr. pointer]  
[block search with calculation]
- Repos offsets can only be deleted with mode group reset.
- The REPOS function is cancelled by pressing the REPOS key.
- The direction key for the opposite direction is disabled and it is not possible to overshoot the initial position.
- When the point of interruption is reached, the "REPOS offset" display becomes zero and at the same time the direction keys have no effect.
- No more than 2 axes can be traversed at the same time.
- The feedrate override switch and the rapid override switch are active.

	<p><b>Caution:</b> After a tool change, the REPOS function can only be used with the same tool dimensions.</p>	
---	--	---

### 3.1.4.6 Scratching

If, for example, a tool brakes during program execution, it is possible to leave the contour for the purpose of tool change using JOG or INC.

After entering the new tool length offset (the tool nose radius remains the same), the new tool is returned to any point on the contour with the interrupted block ("Scratching").

#### Preconditions

- A tool change has not been carried out.
- You are in JOG mode.



**Operating sequence** Using the direction keys, approach a point which is **within the interrupted block**.

On the milling machine control, first select the axes.

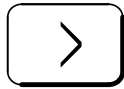


Select AUTOMATIC mode and press the NC START key.

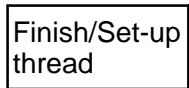
The block end point is approached and program execution continued.

	<p>In a block with circular interpolation (G02, G03) scratching must always take place within a very narrow range. This range is fixed by machine data (see also the machine tool manufacturer's specifications). On moving outside the range: alarm 3018!</p>	
---	--	---

### 3.1.4.7 Finish thread



Press the ETC key to expand the softkey bar.

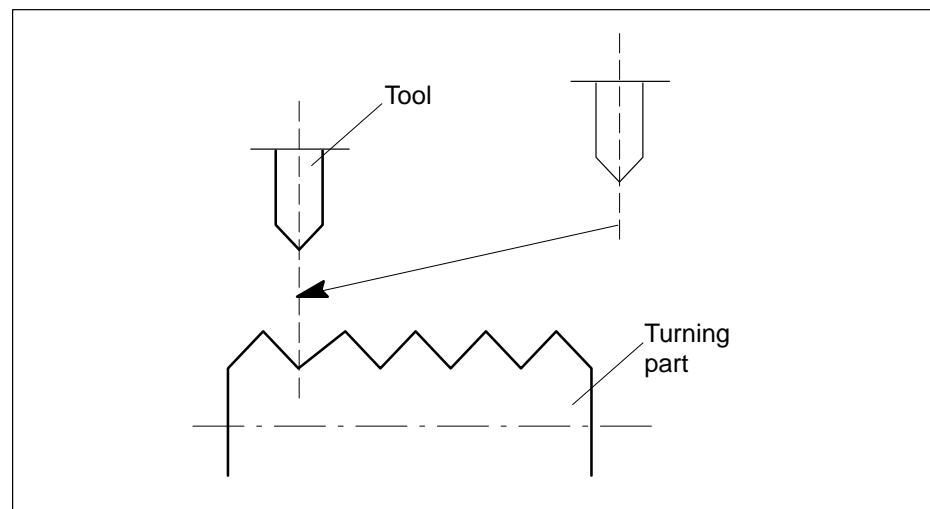


With this function you can load a thread that has been already cut and finish it.

Only threads with constant pitch can be finished.

#### *Operating sequence*

- Load workpiece.
- Position the tool on the thread in such a way that the tip of the tool corresponds to the thread root.
- Press the “STORE POSIT.” softkey.  
The “Offset angle” and the “Starting angle (G92A)” are thus set to zero.
- NC program start (Automatic or MDA).  
The offset angle is calculated and allowed for with the first thread set.  
The offset angle is considered until it is set to zero by pressing the DELETE OFFSET ANGLE softkey.



#### *Note*

The offset angle is not retained after POWER ON.

Maximum value: 359.99999 degrees

Resolution:  $10^{-5}$  degrees, without sign

### 3.1.5 TEACH IN mode

This mode can be used to create main programs and subroutines interactively. If a program has already been selected in AUTOMATIC mode, the selection also applies within TEACH IN.

TEACH IN **only** functions in the master channel of a mode group. You are given the following two options for teach-in of a part program:

- Edit mode and
- Teach block by block

#### *Edit mode*

A part program **cannot** be started in edit mode. There are therefore no limitations in editing. In addition to the standard editing functions such as edit, insert and delete, the position of the cursor can be manipulated by:

- the travel keys
- the handwheel and
- MDA operation in edit mode.

*Teach block by block* Teach block by block is a feature designed to assist the user in refining part programs. In this mode, you can start a part program and edit it or expand it only at the breakpoint. Advance decoding is interrupted at the breakpoint. This ensures that a decoding error does not occur when the teach-in program is executed later in AUTOMATIC mode, and that the teach-in contour is reproduced correctly. You can edit the program and accept the positions at the breakpoint.

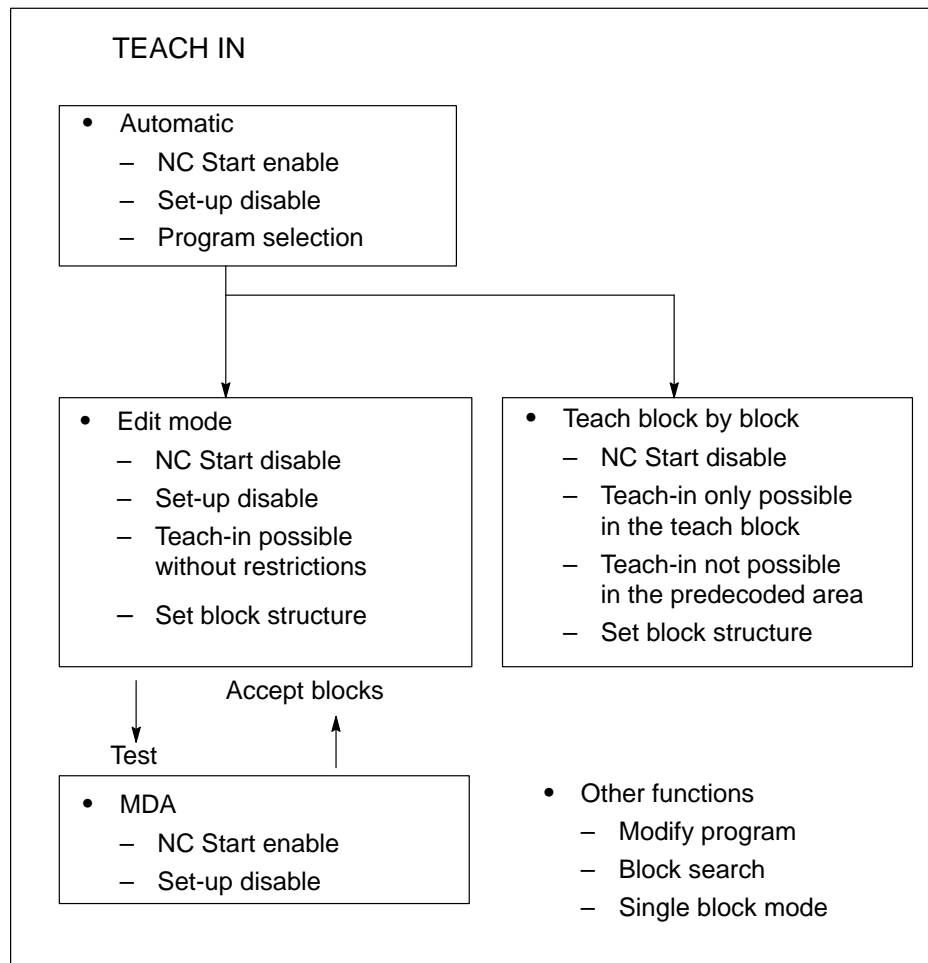
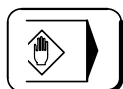


Fig. 3.3 TEACH IN

#### Preconditions

- The machine area has been selected.
- The mode group and the channel for which the program is to be created have been preset.
- The TEACH IN mode in the mode group has been selected.
- To create or modify a program, you enter a program number with the identifiers %, L, MPF or SPF.  
This is done by using the SELECT PROGRAM function.

#### Operating sequence



Press this key to select TEACH IN mode. The associated LED lights up.

Machine	Parameter	Programm.	Services	Diagnosis
				11:10
TEACH IN			Program reset	Mode Gp 1 Channel 1
Actual value		Feed/dist.-to-go	Program pointer	
X	-307.196	0.000	MPF5 0	
Y	-89.600	0.000		
Z	0.000	0.000		
Act. value		Override	Position	Set value
F=	0.00	120%		F= 0.00M
S 1=	0	115%	0.00	S 1= 0
MPF5		Workpiece		1
JOG / handwheel disabled				
Overstore	Edit mode	Program modific.	Mark target block	Teach blk by block
		Search	Select program	

Fig. 3.4 Basic display TEACH IN

The basic display of the TEACH IN mode appears on the screen.

Select  
program

If no program is selected, enter a program name (identifiers MPF, SPF, %, L) on the keyboard and press the SELECT PROGRAM softkey. If the program is not stored in the NCK memory, a new program is created. The program is displayed in the program pointer window. The first 3 blocks of an existing program appear in the editor box.

In the basic display of TEACH IN mode, you can start a part program using NC START as in AUTOMATIC mode. It is **not** possible to set-up a program here (teach-in, accept position, etc.) (Alarm "Set-up disable").

i

When the "Help" key is pressed, information on the TEACH IN mode is displayed on the CRT

Continue

Page through the information.

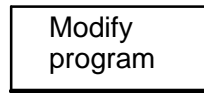
Note

You will find a description of the "Additional machine functions" of the editor and of "Program modification" in separate sections of this manual.

### 3.1.5.1 Set breakpoints

#### Breakpoint

The softkey function MARK TARGET BLOCK allows you to define a breakpoint in an existing program.



Use the arrow keys to select "Stop at target block" BRK.



Set the toggle field to YES. The status BRK is displayed in the status line.



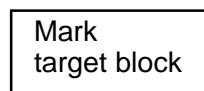
Select RECALL to return to the basic TEACH IN display.



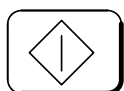
Use the arrow keys to position the cursor on the block where you wish to insert the breakpoint and



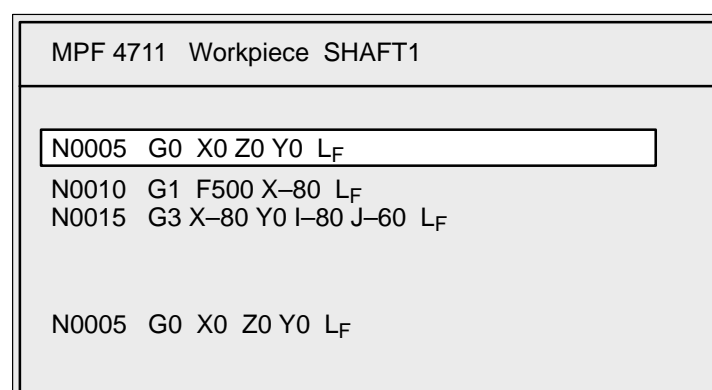
press the MARK TARGET BLOCK softkey. The target block is highlighted in colour.



press the MARK TARGET BLOCK softkey. The target block is highlighted in colour.



When you press NC START, the block containing the breakpoint is displayed. The started program executes up to and including the selected block with the breakpoint.



The block containing the breakpoint (N0005 in the example) is highlighted.

When the block with the breakpoint has been processed, you can interrupt program execution with PROGRAM STOP. You can continue program execution with NC START.

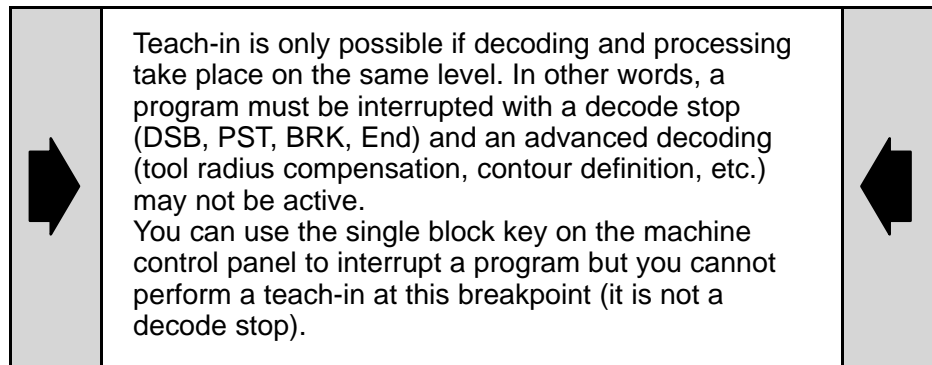


*Program step  
Decode single  
block*

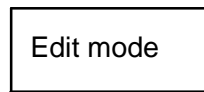
Stop at target block (BRK), program step (PST) and decode single block (DEC) function according to the principle of 1 of 3.  
For example, if BRK was active and PST is set to YES, BRK is automatically reset, etc. You can reset these functions by pressing the PROGRAM MODIFICATION softkey.

*Note*

You will find a description of the functions BRK, PST and DEC in the section entitled "Program modification".



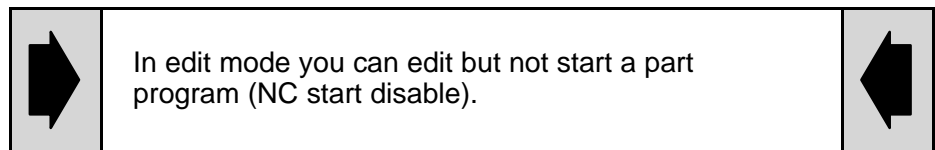
### 3.1.5.2 Edit mode



Press the softkey and the following softkey bar is displayed:

Accept position	Generate block	Accept MDA block	MDA mode	Insert/overwrite	Search	Block struct. settings
-----------------	----------------	------------------	----------	------------------	--------	------------------------

In edit mode there are no restrictions to editing and performing a teach-in at any point in the program.



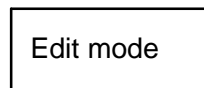
#### Note

It is only possible to switch between the basic display and edit mode in RESET state. This causes the program pointer to be reset to the start of the program when you exit edit mode. Decoding begins with NC start.

#### Preconditions

- TEACH IN is selected
- You have selected a program with the softkey function SELECT PROGRAM.
- The program is in the RESET state, i.e. you must interrupt a running program with the RESET key.

#### Operating sequence



Select the edit mode.

The selected program is displayed in the program pointer and editor windows.

### 3.1.5.3 Block structure settings

#### Block structure settings

The BLOCK STRUCTURE SETTINGS softkey function provides you with a convenient means of defining the block structure of your part program.

When you press the softkey the following display appears:

Machine	Parameter	Programm.	Services	Diagnosis
				11:12
TEACH IN Program reset				Mode Gp 1 Channel 1
Actual value		Feed/dist.-to-go		Block structure/settings
X	-76.006	0.000		Block number <input type="text" value="no"/>
Y	0.000	0.000		Next block no. <input type="text" value="5"/>
Z	0.000	0.000		Block no. diff. <input type="text" value="5"/>
				Settable zero offset <input type="text" value="none"/>
				Workpiece dim. <input type="text" value="none"/>
				Axes <input type="text" value="selected"/>
Act. value		Override	Position	Set value
F= 0.00		120%		F= 0.00M
S 1= 0		115%	0.00	S 1= 0
Axis selec.				
X <input type="text" value="yes"/>				
Y <input type="text" value="yes"/>				
Z <input type="text" value="yes"/>				
<input type="text"/>				
<input type="text"/>				
Edit mode WELLE1 MPF1012 Insert 9				
N025 G04 X2%				
N030 G90 G00 X150 Y150%				
N035 Z100%				
N040 G91 G01 X50 F100%				
N045 X50 Y50 F200%				
Select with the selection key				
Accept position	Generate block		Insert/overwrite	Search

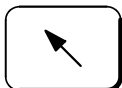
Fig. 3.5 Block structure settings

The following toggle fields (selection fields) are displayed in the "Block structure/settings" screen:

- Block number
- Settable zero offset
- Workpiece dimensions and
- Axes

and the following input fields:

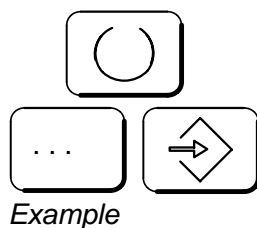
- Next block number
- Block number difference.



You can use the window switchover key to switch to the "Block structure/settings" screen. The active window is always highlighted by a yellow frame.



Use the cursor keys to select toggle or input fields.



Use this key to select the toggle field options.  
You enter values in the input fields, using the keyboard and the input key.

Block structure/settings		
Block number	<input type="text" value="Yes"/>	– Generated NC blocks are numbered
Next block number	<input type="text" value="10"/>	– Generated TEACH block is No. N10
Block number difference	<input type="text" value="5"/>	– Block spacing (N15, N20)
Settable zero offset	<input type="text" value="G53"/>	– G53 is selected
Workpiece dimensions	<input type="text" value="None"/>	– No switchover in inch/metric measurement system
Axes	<input type="text" value="traversed"/>	– During teach-in only the positions of the last traversed axes are accepted

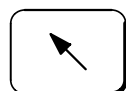
The “Settable zero offset” selection is used to calculate the axis position for SAVE POSITION and GENERATE BLOCK. The zero offset (G53, G54, etc.) is not saved in the part program. The axis positions, which are displaced by the zero offset value, are entered directly in the generated block. Before the block is generated, you should check which zero offset is to be used in the part program.

For the axes, you can choose between the settings “Traversed” and “Selected”. “Traversed” means that a block is generated with the axis positions defined in the “Axis selection” screen which have been traversed since the last block generation. “Selected” means that the block is generated with all of the axes defined in the “Axis selection” screen, regardless of whether the axes have been traversed or not.

The axis position change in the “Traversed” axes setting is effective



- after selection of the axis
- after the position is saved.

#### Note



The entries in the input fields “Next block number” and “Block number difference” also apply to the NCK editor.

Switch to the selection screen “Axis selection”. The axes configured for the machine are displayed along with toggle fields with the selection YES/NO. You use the toggle fields to decide which axes are used in the teach-in (see examples 1 and 2).

	The settings “Axes” and “Axis selection” are generally modified or overwritten by the PLC application program. Please refer to the manufacturer’s documentation for more information.	
---	---	---



Note

Press RECALL to return from the selection screens to the edit mode display..

The axes can also be selected in the input line independently of the “Axis selection” screen. If you select the axes using the input line, the parameters entered in the selection screen have no effect. The selection screen is not modified by the entries in the input line.

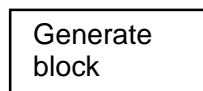
### 3.1.5.4 Creating a TEACH IN program in the edit mode

#### Preconditions

- The block structure has been set
- The axes have been selected
- Position the cursor **in front** of the last L<sub>F</sub>.



#### Operating sequence



Press the GENERATE BLOCK softkey.

An NC block is generated at the cursor position according to the selected block structure and the selected axes or entry in the input line (see examples).

#### Example 1

##### 1. With “Axis selection” screen

Axes	<input type="text" value="traversed"/>
Axis selection	
X	<input type="text" value="yes"/>
Y	<input type="text" value="no"/>
Z	<input type="text" value="yes"/>
A	<input type="text" value="yes"/>

If axes X, Z and/or A have been traversed during the teach-in, the axis positions are entered in the program with “Accept position”. If a selected axis has not been traversed, the position is not accepted.

Generated NC blocks:

- without traversing movement  
N10 L<sub>F</sub>
- with traversing movement on axes X and Y  
N10 X = 100 L<sub>F</sub>

The axis position of the Y-axis is not entered in the block as it has not been selected.

## 2. Entry in the input line

Enter the axis names X, Y and Z in the input line. Regardless of the selection in the “Axis selection” screen and regardless of whether a traversing movement has taken place, a block is generated with the axes in the input line: N10 X=100 Y=10 Z=50 L<sub>F</sub>

### Example 2

#### 1. With “Axis selection” screen

Axes	selected
Axis selection	
X	no
Y	yes
Z	yes
A	yes

The axis positions of axes, Y, Z and A are entered in the NC block with “Accept position”, regardless of whether a movement has taken place.

Generated NC block:

N10 Y = 50 Z = -100 A = 220 L<sub>F</sub>

#### 2. Entry in the input line

Enter the axis names X, Y and Z in the input line. Regardless of the selection in the “Axis selection” screen a block is generated with the axes entered in the input line: N10 X=100 Y=50 Z= -100 L<sub>F</sub>

### Note

The accepted positions are displayed in the format X=100 and not X100.



Once you have generated a block, you can insert G commands, technology data, etc. in the block using the keyboard.

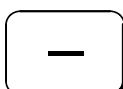
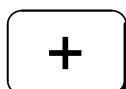
## 3.1.5.5 Accepting axis positions

### Milling machine

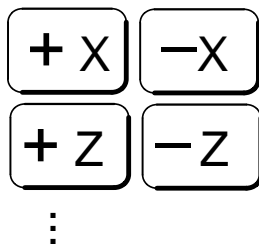


Select the appropriate axes and

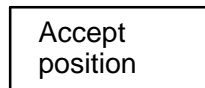
...



traverse these according to the desired dimension in positive or negative direction.

*Turning machine*

Traverse the axis according to the desired dimension.



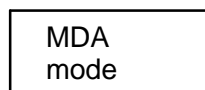
Pressing the ACCEPT POSITION softkey causes the current position on the traversed or selected axes (according to the selected block structure) to be entered in the NC block.

*Note*

If you enter the names of the axes whose positions are to be saved in the input line, the settings in the axis selection display **no longer** apply (see the section “Modifying an existing part program with block by block teach-in”).

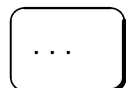
**3.1.5.6 MDA in edit mode**

MDA allows you to start NC blocks in edit mode/TEACH IN for purposes such as testing, setting up, etc. MDA mode provides a third way of reaching a position in addition to the use of the travel keys and the handwheel.

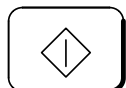


Press the MDA MODE softkey.

The MDA window is highlighted by a yellow frame.



Use the keyboard to enter blocks in this window.



You can press NC start to start the blocks and run them like a program.



Use the arrow keys to select the MDA block which you would like to save in the active TEACH IN program. The current line number is displayed.

MDA1	Insert	3
M19 S100 L <sub>F</sub>		
L200 L <sub>F</sub>		
<b>M3 S1025 M7 L<sub>F</sub></b>		
M4 M8 L <sub>F</sub>		



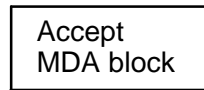
Press RECALL to return from the MDA mode to the edit mode.

*Note*

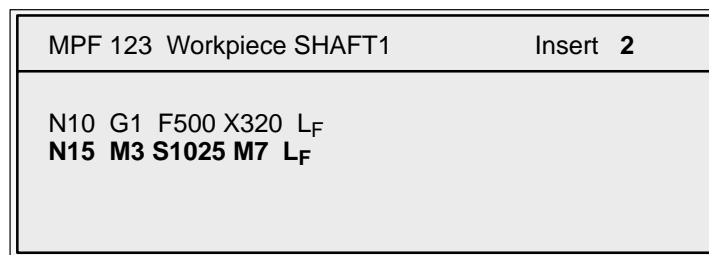
You should only exit MDA mode in the NC reset state. If the NC reset is not enabled the NC start is interlocked.



Move the cursor to the point in the TEACH IN program where you want to insert the MDA block.



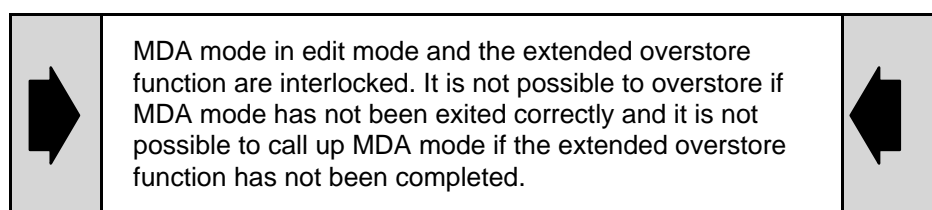
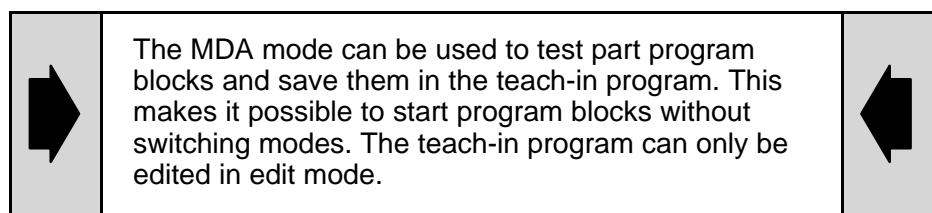
Pressing the ACCEPT MDA BLOCK softkey saves the selected MDA block in the TEACH IN program.



If you want to save more MDA blocks, switch to the MDA mode again. Use the arrow keys to select the next block, switch back to the edit mode and save the block in the teach-in program as described above.

*Notes*

- The functionality of the MDA mode in edit mode/teach-in is identical to the MDA mode. The CLEAR action is ignored in MDA mode under TEACH IN.
- If an MDA program has been executed, it can only be edited after the last existing MDA block.
- An MDA program can only be edited and started without restriction after a RESET.
- MDA 1 stands for the MDA program in channel 1, MDA 2 for channel 2 etc.





### 3.1.5.7 Block by block teach-in

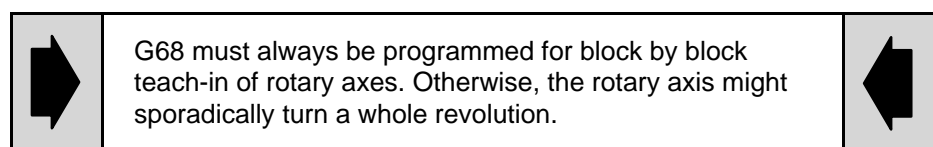
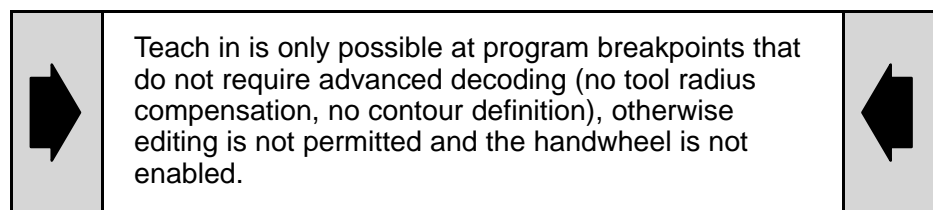
#### Preconditions

“Block by block teach-in” only functions in the NC stop state. If a new part program is opened, for example, you must first start it using NC start. Only then can you switch to block by block teach in otherwise the message “Editing error, no permission to edit” is output.

In “Block by block teach-in” you can start the program and only modify or extend it at breakpoints. The advance decoding is switched off at the breakpoint.

You can advance to a breakpoint by:

- RESET (program start)
- Traversing until stop at target block (BRK)
- Decode single block (DSB)
- Program step (PST)
- Block search with calculation and subsequent processing of at least one block in decode single block mode.
- Program end without M02/M30/M17 (TEACH IN and MDA programs stop even without end of program character).



The following modifications are possible at the breakpoint:

- Delete following block
- Edit following block
- Insert new block before following block
- Save axis positions in following block or insertion block
- Generate insertion block with axis positions (see description and examples for edit mode)

#### Note

You can interrupt program execution even within a subroutine. However TEACH IN is only possible in the main program level.

The axis positions can be approached using the travel keys or the handwheel if JOG/handwheel has been enabled.

When the modifications have been made in the block following the breakpoint, this position is approached after NC start. You can single-step or travel to the next breakpoint.



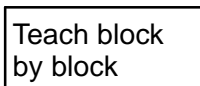
You can only switch between EDIT MODE and TEACH-IN BLOCK BY BLOCK after pressing RESET.

### 3.1.5.8 Modifying an existing part program with block by block teach-in

#### Preconditions

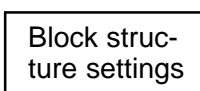
- An existing program has been selected in the TEACH IN basic display using SELECT PROGRAM.
- A program breakpoint has been selected (see the section “Setting breakpoints”).

#### Operating sequence



Press the TEACH BLOCK BY BLOCK softkey. The following softkey bar is displayed:

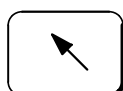
Accept position	Generate block	Enable JOG handwheel	Mark target block	Insert/overwrite	Search	Block structure settings
-----------------	----------------	----------------------	-------------------	------------------	--------	--------------------------



Press this softkey to enter the block structure settings. The “Block structure/settings” and “Axis selection” selection screens are displayed (see the section “Edit mode”). Since it is possible to interrupt the execution of a program in teach block by block and to edit the program at the breakpoint, the entire “environment” of the program is active. This means that in contrast to edit mode, you don’t need to set the zero offset and workpiece dimensions in the block structure. The actual values are used in the block by block teach-in.

#### Note

The block number and block number difference settings are also valid in the NCK editor.

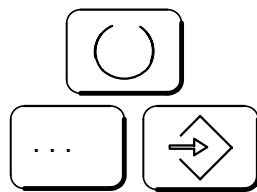


Switch to the “Block structure/settings” selection screen.



Use the arrow keys to move between the toggle fields:

- Block number
  - Axes
- and the input fields
- Next block number
  - Block number difference



Press the selection key to display the different settings in the toggle fields.

You can enter the values in the input boxes using the keyboard and pressing the INPUT key.



Switch to the “Axis selection” screen and enter the desired settings. You will find a detailed description of these selection screens in the previous sections.



Press RECALL to exit the selection screens.



Now start the part program by pressing NC START. The program executes until the selected breakpoint.

#### Note

The program can also be started from the TEACH IN basic display. You can subsequently switch to teach block by block.

The breakpoint block (N15 in the figure) is highlighted on a black background.

The teach block is highlighted in colour (N20 in the figure).

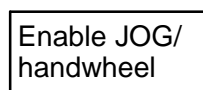
MPF 4711	Workpiece SHAFT1	Insert 4
N10 X0 Y0 Z0 G0 L <sub>F</sub>		
N15 Y125 G1 F500 L <sub>F</sub>		
N20 G0 Y0 L <sub>F</sub>		
N30 L1 L <sub>F</sub>		

You can now enter your changes in the teach block (N20 in the figure).

#### Edit

You can edit this block using the keyboard. Please refer to the section “Programming” for a description of the NCK editor

#### Accept position



Press the ENABLE JOG/HANDWHEEL softkey to cancel the set-up disable. The “JOG/handwheel disabled” dialog text disappears and the dialog text “JOG/handwheel enabled” is displayed.

#### Milling machine



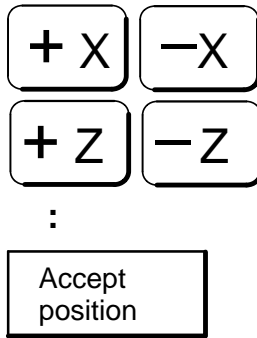
Select the appropriate axes and

...



traverse these according to the desired scale in positive or negative direction.

### Turning machine



Traverse each axis to the required position.

Pressing the ACCEPT POSITION softkey causes the current position on the traversed or selected axes (according to the block structure settings and the axis selection) to be entered in the NC block at the cursor position.

### Note

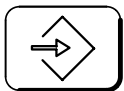
You can also accept the positions by entering the axis names in the input line. Press the HOME key to switch from the editor window ("MPF4711 Workpiece SHAFT1" in the example) to another window. Use the keyboard to enter in the input line the names of the axes whose positions you wish to save in the block (e.g. X Y Z ). Press the HOME key to switch back to the editor window and press the ACCEPT POSITION softkey. The axis positions defined in the input line are now accepted regardless of the settings in the "Axis selection" screen. You can also activate the input line with the END key. In this case you don't have to switch to the input window.

### Insert block

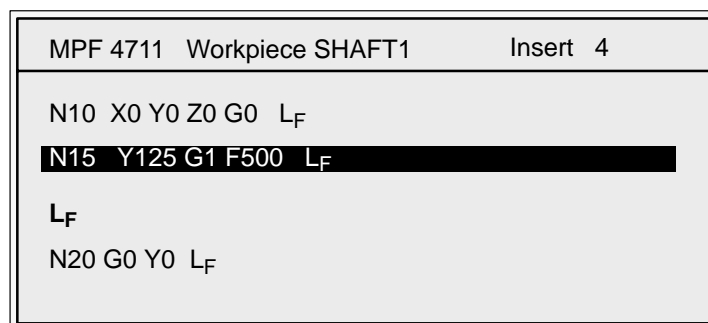
You can insert a block **before** the teach block.



Position the cursor on the first digit of the teach block (the 'N' of N20) and

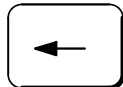


press the INPUT key.



You can edit or insert the axis positions with “Accept position” in this generated block.

Generate block



Position the cursor on the first digit of the teach block (the “N” of N20 in the figure).

Generate  
block

Press the GENERATE BLOCK softkey to generate a block **before** the teach block (N16 in the figure).

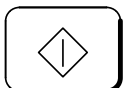
MPF 4711	Workpiece SHAFT1	Insert 4
N10 X0 Y0 Z0 G0 L <sub>F</sub>		
N15 Y125 G1 F500 L <sub>F</sub>		
N16 X= -25.600 Y= 68.300 L <sub>F</sub>		
N20 G0 Y0 L <sub>F</sub>		

Note

Please note that the function GENERATE BLOCK generates the block created in the block structure.

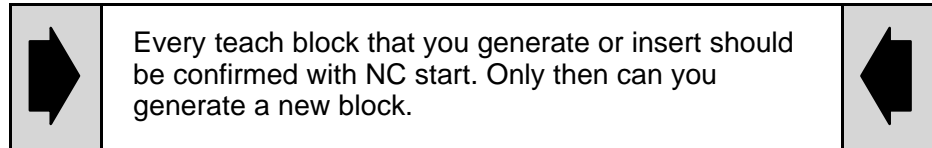
For the example in the figure, the following settings would be required:

Block structure/settings	
Block number	<input type="text" value="yes"/>
Next block number	<input type="text" value="16"/>
Block number difference	<input type="text" value="5"/>
Axes	<input type="text" value="selected"/>
Axis selection	
X	<input type="text" value="yes"/>
Y	<input type="text" value="yes"/>
Z	<input type="text" value="no"/>
A	<input type="text" value="no"/>



If you now press NC START, execution of the NC program will continue with the inserted or modified blocks.

You can set the next breakpoint **before** pressing NC start.



*Note*

To avoid collisions caused by operator errors, the following operating rules are built into the system:

1. If the travel key or handwheel is used to move the position away from that of the breakpoint block, the new axis position must either be accepted or the position should be returned to the block start position. Otherwise the NC start is interlocked.  
The block start position can be reached using the REPOS function. You will find a description of the operator actions for the REPOS function in the section "JOG mode".  
If, after REPOS, the function SAVE POSITION or GENERATE BLOCK is activated and "Traversed axes" was selected, the repositioned axis is not saved in the part program. If further axes are to be traversed after REPOS, REPOS should be deselected first.
2. If the axis positions in the teach block have been accepted, the travel keys and the handwheel remain disabled until the block has been processed with NC start.

The following general rule applies:

You can abort the teach run at any time by:

- RESET or
- Operating mode change

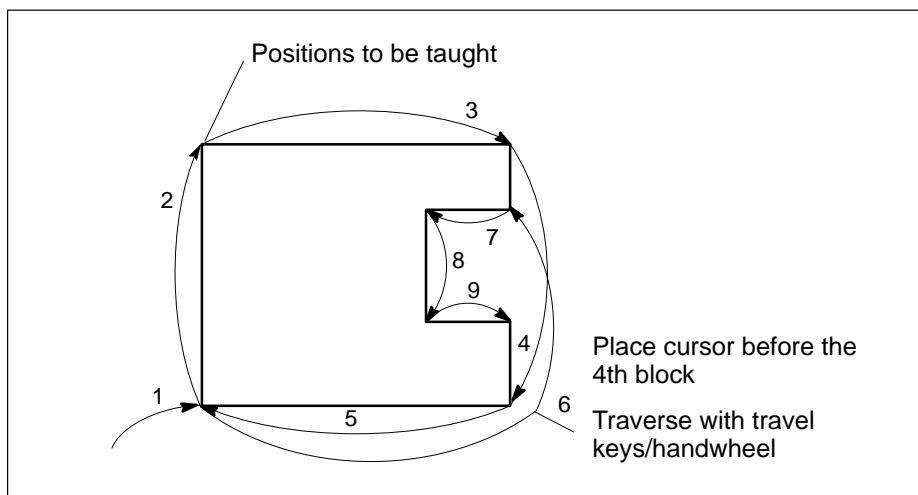
You will find a description of the functions overstore and block search in the sections of the same name.

*REPOS in Teach-In*

If the search is started in the teach-in mode and the program is to be subsequently processed in "Teach block by block", the DSB must be enabled before the search is performed. This is the only way of ensuring that processing and decoding of the selected part program are correctly coordinated. If DSB is not enabled, you can neither edit nor reposition with the Repos offsets.

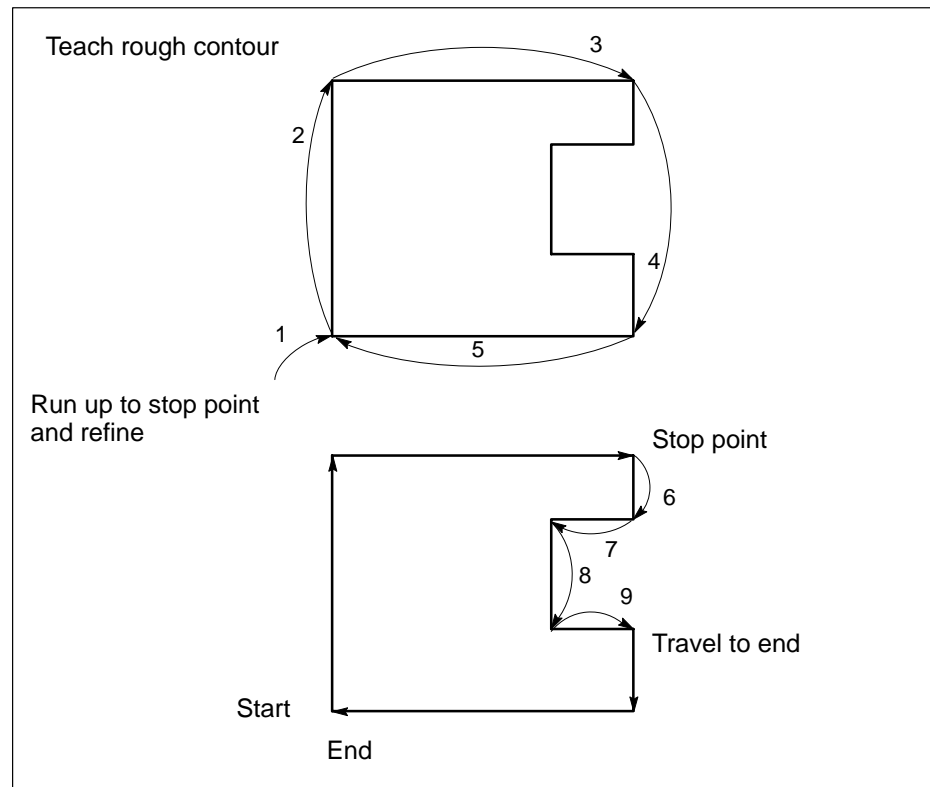
### 3.1.5.9 Examples

*Edit mode example* Positions 1 to 5 are approached using JOG keys and saved in the part program with the softkey GENERATE BLOCK. The editing cursor is placed on the block with position 3. The missing positions 7 to 9 are subsequently approached using JOG keys and similarly entered in the part program with the softkey GENERATE BLOCK.



It is possible to edit the part program at any point in any order without traversing.

First teach the rough contour. To do this approach positions 1 to 5 with the handwheel and press the GENERATE BLOCK softkey to save the positions in the part program. In the second step, run the program up to position 3. Positions 6 to 9 are **each** approached using the handwheel, inserted in the part program using the GENERATE BLOCK softkey **and executed** by pressing NC Start. Finally run the program until the end.

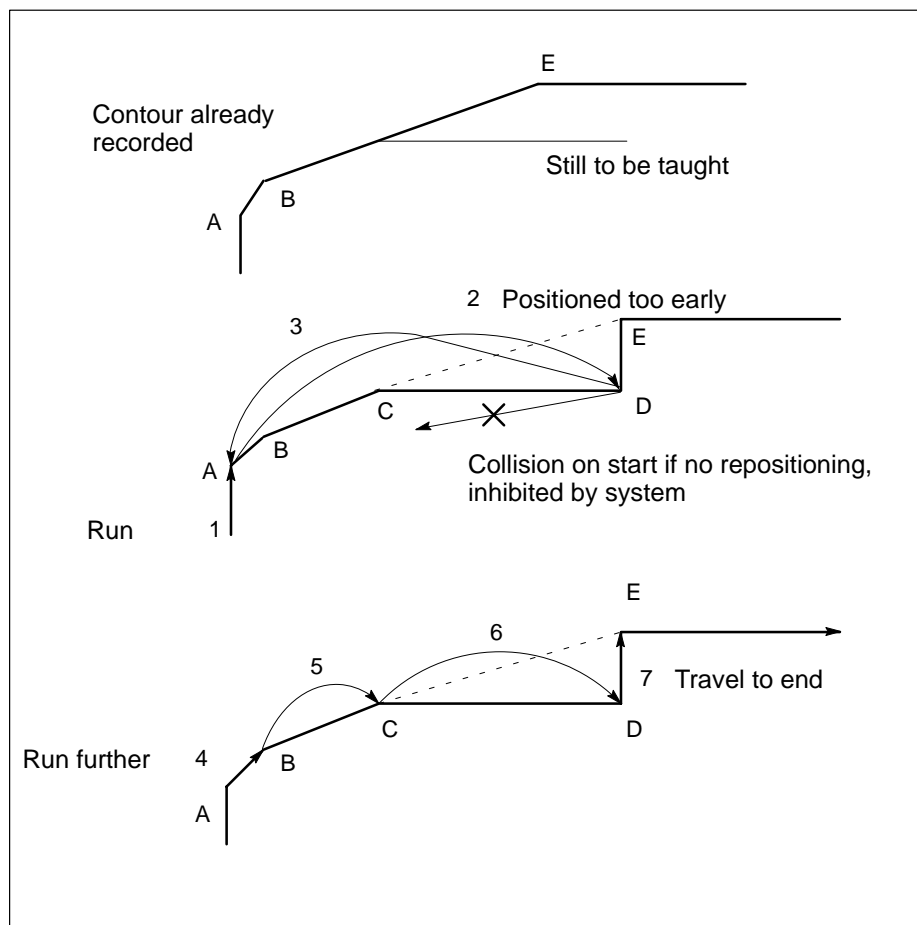


It is only possible to teach or edit in the following block. You must observe the machining sequence. Every block that you modify must be processed by pressing NC Start.



*2nd example for teach  
block by block*

A rough contour with the positions A, B and E which you have already recorded is now to be refined with the positions C and D. In step 1, the operator runs the program until position A. In step 2, he positions the handwheel on position D. He notices that he has interrupted the program one block too soon. He does not want to save the position, but to run the program until position B. But **NC Start is interlocked** because of the movement away from position A. In step 3 he **must** first move the position back to A. He can then execute the missing block to position B in step 4. Now he can teach position C and D.



### 3.1.6 MDA mode

Individual blocks or a sequence of blocks can be executed with the machine, without creating a program.

Blocks are input to the memory of the control via the keyboard. The system provides programs MDA1...6 for each channel (MDA1 for channel 1, MDA2 for channel 2, etc.).

When you press NC Start, the control processes the blocks. You can set CLEAR (CLR) under program modification to determine whether the block sequence is to be cleared after processing. If CLR is not active, the block sequence is retained after processing.

#### Application

Used, for example, in connection with operations in "JOG" mode.

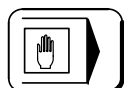
Machine	Parameter	Programm.	Services	Diagnosis
				11:14
MDA		Program reset	Mode Gp 1 Channel 1	
Actual value		Feed/dist.-to-go	Program pointer	
X -76.006		0.000	0	
Y -61.440		0.000		
Z 0.000		0.000		
Act. value	Override	Position	Set value	Auxiliary functions
F= 0.00	120%	0.00	F= 0.00M	T = M =
S 1= 0	115%	0.00	S 1= 0	D = M =
MDA1		Insert	3	G functions
X100 G1 F500%				
M03 S200%				
Z-10				
Overstore		Program modific.	Insert/ overwrite	Search Delete blocks

Fig. 3.6 Basic display in MDA mode

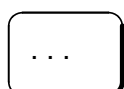
#### Preconditions

- The machine area has been selected.
- The mode group and channel for which the program is to be created have been preset.

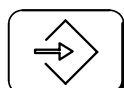
#### Operating sequence



To select MDA mode, press this key on the machine control panel.



Use the keyboard to enter one or more blocks in the MDA... input window.



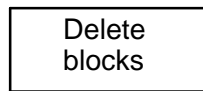
Press the INPUT key at the end of each block. An end of block character (LF) is generated automatically and the cursor moves to the next line.

**Note**

You will find a complete description of the operator actions for editing in the section entitled "Programming".



When you press NC START, the program blocks are processed. If the program modification clear (CLR) is active, all the blocks are cleared after processing, or the MDA blocks are cleared after pressing the RESET key.



If the program modification clear (CLR) is not active, you can use the DELETE BLOCKS softkey to clear the program blocks.

You can interrupt program execution in one of the following ways:

- Setting the DSB, BRK and PST program modifications
- Pressing NC STOP

The program is restarted by pressing

- NC START



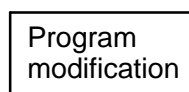
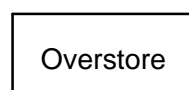
If you press the RESET key, execution is terminated. A RESET is also generated by changing the mode, i.e. execution is interrupted. Pressing NC START starts the program from the beginning. If clear (CLR) is active, RESET deletes the MDA blocks.

**Note**

While the program is being executed, the display "Program running" appears in the frame of the mode field.

The entire programming scope can be used without restriction.

If an MDA program is interrupted by decode single block (DSB) or program step (PST), you can edit without restrictions after the breakpoint block. When you press NC START, execution begins at the breakpoint position.



The machine functions

- Overstore
- Program modification

are described in the "Additional machine functions" section.

**Note**

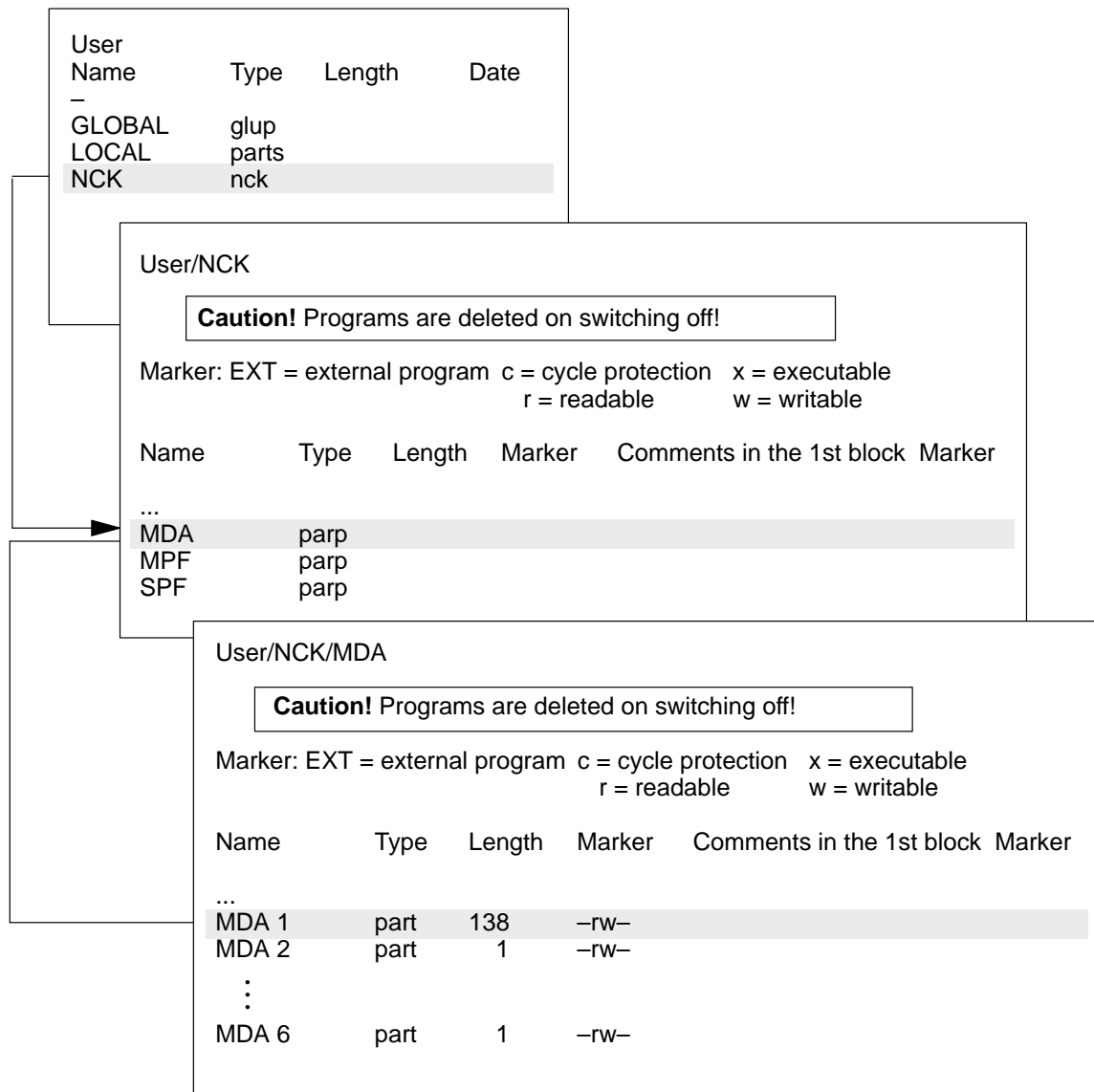
The contents of the MDA program are retained if clear (CLR) is not active and the softkey DELETE BLOCKS has not been pressed.

It is therefore possible to copy the contents of programs MDA 1 to 6 to a part program in the user area PROGRAMMING.

### 3.1.6.1 Copying MDA programs

Programm.

To do this, select the PROGRAMMING area and open directory NCK/MDA (see following example).



Copy

Now press the softkey COPY.

Now place the cursor on directory “MPF”. The path that you have selected is displayed in the display line “Copy to”. This display is updated.

Machine	Parameter	Programm.		Services	Diagnosis	
						13:17
<b>JOG</b>		Program reset				Mode Gp 1 Channel 1
Programming USER/NCK						
Copy from: USER/NCK/MDA/MDA1						
Mark: EXT = external program    c = cycle protect.    x = executable r = readable         w = writable						
Copy to: USER/NCK/MPF/???						
Name	Type	Length	Mark	Comment in the 1st block/date		
MDA	parp					
SPF	parp					
		Copy				

*Fig. 3.7 Copying MDA programs*

Now enter the program type and the program number, e.g. MPF 1234 and press the softkey COPY.

Main program MPF 1234 is created from the NC blocks in MDA 1.

*Note*

You can only use the COPY function within the directory NCK, e.g. from MDA to MPF.

Edit

Save  
under

To save onto hard disk it is recommended that you use the softkeys EDIT / SAVE UNDER (in area PROGRAM, directory NCK).

### Saving the program currently selected

with the softkey: **CORRECT PROGRAM / SAVE TO DISK**

or **CORRECT PROGRAM / SAVE UNDER**

is described in the Section “Saving programs”.

### 3.1.7 AUTOMATIC mode

Programs from the NCK part program memory can be executed automatically in this mode.

It is possible to select:

- Workpieces
- Main programs
- Subroutines.

To execute a part program in this mode, the control calls the blocks in sequence and evaluates them. Evaluation takes all the offsets addressed by the program into account. The blocks prepared in this way are processed in sequence.

The part program can be entered via the universal interface (e.g. by means of computer or PC), or through the keyboard, or it can be loaded from the MMC hard disk to the NCK memory. While one part program is being executed, another can be entered or read in simultaneously.

Machine	Parameter	Programm.		Services	Diagnosis	
						12:46
AUTOMATIC Program reset						Mode Gp 1 Channel 1
Actual value		Feed/dist.-to-go		Program pointer		
X	-76.006		0.000	MPF1 0		
Y	-61.440		0.000			
Z	0.000		0.000			
Act. value		Override	Position	Set value	Auxiliary functions	
F=	0.00	120%		F=	0.00M	T = M =
S 1=	0	115%	0.00	S 1=	0	D = M = H = M =
Current block					G functions	
Overstore	Preset DRF	Program modific.	Program view	Program correction	Select prog./job	

Fig. 3.8 Basic display in Automatic mode

### 3.1.7.1 Workpiece and program selection with the data selector

#### Operating sequence

 Select  
prog./job

Press the softkey SELECT PROG./JOB in the automatic basic display.

The “data selector” (file and directory selection) appears in the basic display for Automatic.

Machine		Parameter		Programm.		Services		Diagnosis		
									13:21	
AUTOMATIC Program reset									Mode Gp 1 Channel 1	
Actual value		Feed/dist.-to-go				Program pointer				
X		0.000		0.000		<div style="border: 1px solid black; width: 20px; height: 40px; margin: 0 auto;"></div>				
Y		0.000		0.000						
Z		0.000		0.000						
USER/LOCAL										
Name	Type	Length	Mark	Comment in the 1st block/date						
ACR            part ELG_LIN      part EXMP_T      part GIA_LIN      part IKA_BSP      part IKA_BSPG     part IKA_KOMP     part										
										Select prog./job

Fig. 3.9 Workpiece selection 01

#### Workpiece selection

The workpieces are stored in the directory LOCAL (it is not possible to load from the directory GLOBAL).

#### Prerequisite

In the example, the workpiece “Shaft 1” is to be loaded.

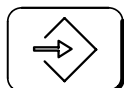
- A job list (load list) exists.  
Please refer to the Section “Programming” for more detailed information about the job list.

#### Note

If LOCAL has not yet been selected you must do this manually.

#### Operating sequence

Place the cursor on the directory LOCAL.



Now press the INPUT key,  
the workpieces in this directory (LOCAL) are listed.

Now position the cursor on the required workpiece.

Example:

Machine	Parameter	Programm.	Services	Diagnosis
				12:46
AUTOMATIC Program reset				Mode Gp 1 Channel 1
Actual value	Feed/dist.-to-go		Program pointer	
X	-76.006	0.000	MPF1 0	
Y	-61.440	0.000		
Z	0.000	0.000		
USER				
Name	Type	LengthMark	Comment in the 1st block/date	
LOCAL NCK	parts nck			
Select prog./job				

Fig. 3.10 Workpiece selection 02

Select  
prog./job

Now press the softkey PROG./SELECT JOG.

The following dialog text is displayed: "Load workpiece?"

Now press the softkey SELECT PROG./JOG and the workpiece is loaded together with programs and data from the hard disk of the MMC into the working memory of the NC.

*Note*

The program length display for the NCK and the hard disk do not correspond.

When the workpiece is loaded and if the part program in the MMC is, for example, 596 characters long, a length of only 564 characters is displayed in the NCK.





Machine		Parameter		Programm.		Services		Diagnosis	
12:50									
AUTOMATIC Program reset						Mode Gp 1 Channel 1			
Actual value		Feed/dist.-to-go				Program pointer			
X	-76.006	0.000				MPF1 0			
Y	-61.440	0.000							
Z	0.000	0.000							
USER/LOCAL									
Name	Type	LengthMark		Comment in the 1st block/date					
ALBER012	part								
ALBER051	part								
FRAES001	part								
FRAES002	part								
PEZZ01	part								
PEZZ03	part								
<div> <div>Program view</div> <div>Select prog./job</div> </div>									

Fig. 3.11 Workpiece selection 03

The workpiece name and the part program (if defined in the job list) is accepted in the selected channel in the program pointer.

The message "Workpiece data are being loaded" is displayed in the dialog line during data transfer.



	<p>The machine manufacturer can configure via machine data whether the workpiece last active is to be loaded again after restart of the control provided that a workpiece was loaded and selected before switching off the control. When the control is switched on, the workpieces are loaded in sequence starting with the workpiece in channel 1 to channel n. The STANDARD workpiece is always loaded first.</p>	
---	--	---

If a workpiece contains no job list or the standard job list and several or no main programs exist, **no** program is selected for processing and **no** workpiece name is displayed.

#### Remedy

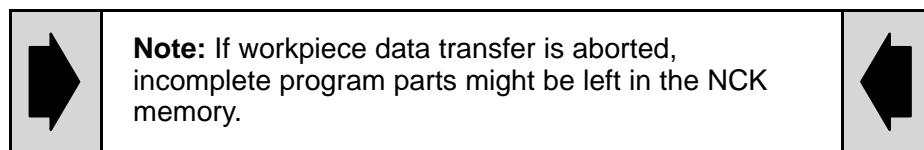
Activate the SELECT command in the job list, e.g.: SELECT MPF100 CH=1.

See also Section "Programming" under job lists.

	<p>Only workpieces from the LOCAL directory can be loaded in the NCK memory by means of the SELECT WORKPIECE function. Data already existing in the NCK memory is overwritten without warning.</p>	
---	--	---

Stop work-  
piece sel.

You can use this softkey to abort workpiece data transfer from the hard disk to the NCK memory.



### Dialog messages and alarms

If errors occur on execution of the softkey functions SELECT PROG./ JOB, the following dialog messages and alarms are generated:

Error	Dialog message
No directory has been called	Illegal directory
A directory but <b>no</b> workpiece has been selected	Please enter correct name
The softkey was pressed but the workpiece name contains illegal characters	Name illegal
The softkey has been pressed while a workpiece list is being processed.	Workpiece already being loaded
Workpiece selection disabled by PLC.	Workpiece selection disabled

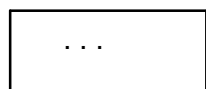
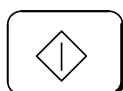
Error	Alarm
The workpiece does not exist on the MMC hard disk.	Workpiece not available
The job list on the MMC is faulty.	Job list faulty
An error occurs during job list processing.	Several alarms
A workpiece without job list and with several (or no) main programs has been transferred to the NCK.	Transfer workpiece, no program selected

Basically, the following applies

If an alarm occurs during workpiece transfer, the faulty files are not transferred. First, the job list syntax is checked. Should the job list contain syntax errors, it is not processed, i.e. a workpiece is not transferred.

The generation of load lists (job lists) is described in the "Programming" section.

Press the NC START key to start the part program.



Select the machine functions by pressing the corresponding softkey.

The individual machine functions are described in the “Additional machine functions” subsection.

### Selecting part programs

In some cases, several main programs and subroutines are loaded with the workpiece. A particular program can be selected with the data selector.

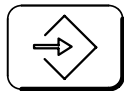
### Operating sequence

Select  
prog./job

Press the softkey SELECT PROG./JOB and open the NCK directory (same procedure as for workpiece selection).

The following can be selected:

...  
MDA  
MPF  
SPF



To select a main program, for example, the cursor is placed on the MPF directory and the INPUT key is pressed.

Now, all the main programs (MPF) in the working memory are listed.

Machine	Parameter	Programm.	Services	Diagnosis	
					13:39
AUTOMATIC Program reset					Mode Gp 1 Channel 1
Actual value		Feed/dist.-to-go		Program pointer	
X	0.000	0.000	WELLE1 MPF1012		
Y	0.000	0.000	0		
Z	0.000	0.000			
USER/NCK/MPF					
Name	Type	Length	Mark	Comment in the 1st block/date	
MPF1	part	280	-rwx	Zeichnungsnr.: 31K/112-12S	
MPF1012	part	219	-rwx	Z-Nr.: 34K100/12	
MPF11	part	307	-rwx	Zeichnungsnr.: 3	
MPF13	part	564	-rwx	MPF	
MPF22	part	167	-rwx	Grundstellung	
MPF23	part	103	-rwx	Lader	
MPF4711	part	98	-rwx		
Select prog./job					

Fig. 3.12 Program selection

Select  
prog./job

Place the cursor on the required program and press the softkey SELECT PROG./JOB.

The program is selected without any messages being displayed and is accepted by the program pointer.

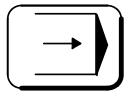
### 3.1.7.2 SELECT PROGRAM/JOB function

In addition to program selection with the DATA SELECTOR, it is also possible to select a program (MPF, SPF) by entering %... and L ... as is possible with other SINUMERIK systems.

#### *Preconditions*

- The machine area has been selected.
- The mode group and channel have been preset.
- The AUTOMATIC mode in the mode group is selected on the machine control panel.

#### *Operating sequence*



Press this key to select AUTOMATIC mode.

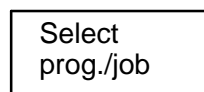
If you wish to execute a new program, enter the program number.  
Entering a program number:



Enter the program number and the corresponding identifier (% , L, MPF or SPF).



You can delete character by character from right to left in the input line with the DELETE INPUT key.

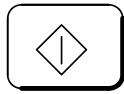


Press the SELECT PROGRAM softkey.

The selected program is accepted by the program pointer.

	<p>If a program is entered as % ... or L ..., it appears in the program display as MPF ... or SPF ... .</p> <p><b>Caution!</b> It is also possible to select programs that are not in the part program memory.</p>	
--	--	--

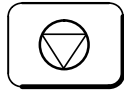
### 3.1.7.3 Starting and interrupting a part program



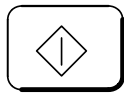
To start the part program, press the NC START key.

You interrupt execution of the current part program by pressing NC STOP.

With the START KEY you interrupt execution of the current part program provided program start has not been disabled. With FEED STOP ("FST" message) you can stop the axis movements. The NC program is not interrupted. With FEED START you enable the axis movements again.



NC STOP



NC START



FEED START



FEED STOP

#### Notes

- While the part program is being executed, a "Program running" display appears in the mode field.
- If, during program execution, FEED STOP is activated, FST (Feed stop) is displayed in the status field and the feed drives are brought to a standstill while maintaining the programmed path movement.
- In the case of NC STOP the NC STOP display appears in the mode field.

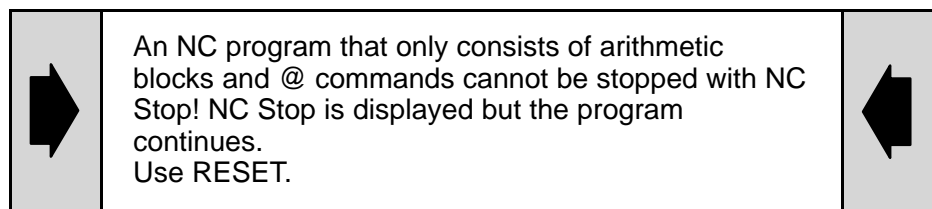
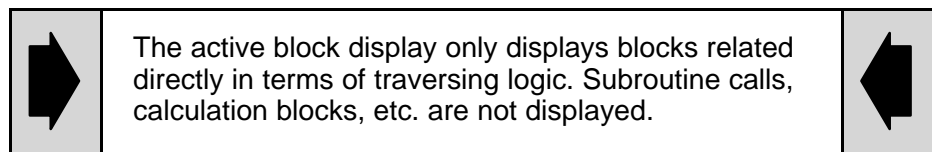


If this key is pressed, execution of the part program is terminated.

- The control is put into the "RESET" state:
  - The NC control remains synchronous to the machine
  - All buffers and active memories are cleared (however, the part program memory contents are preserved)
  - The control is in the Reset position and ready to run a new program.
- "Program reset" is displayed in the mode field.
- Error messages with the delete condition "Reset" are deleted.

A program can be interrupted in one of the following ways:

- Pressing the SINGLE BLOCK key on the machine control panel
- NC STOP key or internal signal
- PROGRAM STOP M01 or M00 program modification
- DECODE SINGLE BLOCK (DSB) program modification
- PROGRAM STEP (PST) program modification
- BREAKPOINT (BRK) program modification
- FEED STOP keypress or signal
- Switch to JOG mode



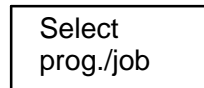
#### 3.1.7.4 Search for first letter function (from SW 6)

An option for fast selection of directories has been implemented with the Search for first letter function.

##### *Preconditions*

- The machine area has been selected
- The AUTOMATIC mode has been selected

##### *Operating sequence*



Press the SELECT PROGRAM/JOB softkey.



Enter the first letter desired in the entry line via the keyboard.



After operation of the selection key, the cursor is positioned on the search key. If no search key is found, the cursor remains unchanged.

### 3.1.7.5 Program view (from SW 6)

Program  
view

From SW 6, a **Program view** softkey is provided in the Automatic basic display.

When you call this softkey function, the following screen is displayed:

Maschine	Parameter	Programm.	Dienste	Diagnose															
000300 A1 Antriebskopplung aus <02BEH> <>				10:19															
<b>AUTOMATIC</b> Programm reset				BAG : 1 Kanal : 1															
Teileprogramm		Programmzeiger																	
N0005 G0 X50 Y50 D1% N0010 R10=1% N0015 R11=25% N0020 R12=50% N0025 R13=1% N0030 Q710 R0 R10% N0035 Q121 R13 K1 K-30% N0040 M30%		AF FE710 MPF1 0																	
		<table border="1"> <thead> <tr> <th>Istwert</th> <th colspan="2">Restweg</th> </tr> </thead> <tbody> <tr> <td>X</td> <td>0.000</td> <td>0.000</td> </tr> <tr> <td>Y</td> <td>0.000</td> <td>0.000</td> </tr> <tr> <td>Z</td> <td>0.000</td> <td>0.000</td> </tr> </tbody> </table>			Istwert	Restweg		X	0.000	0.000	Y	0.000	0.000	Z	0.000	0.000			
Istwert	Restweg																		
X	0.000	0.000																	
Y	0.000	0.000																	
Z	0.000	0.000																	
		<table border="1"> <thead> <tr> <th>Istwert</th> <th colspan="2">Overr. Pos.</th> </tr> </thead> <tbody> <tr> <td>F=</td> <td>0.00</td> <td>95%</td> </tr> <tr> <td>Vorschub F=</td> <td colspan="2">0.00M</td> </tr> <tr> <td>S 1=</td> <td>0</td> <td>105%</td> </tr> <tr> <td>Sollwert S 1=</td> <td colspan="2">0</td> </tr> </tbody> </table>			Istwert	Overr. Pos.		F=	0.00	95%	Vorschub F=	0.00M		S 1=	0	105%	Sollwert S 1=	0	
Istwert	Overr. Pos.																		
F=	0.00	95%																	
Vorschub F=	0.00M																		
S 1=	0	105%																	
Sollwert S 1=	0																		

In the Part program window, the program selected for execution is displayed. The display and the cursor run synchronously with the execution.



### 3.1.7.6 Expanded program view (from SW 6.3)

In SW version 6 and earlier, only program blocks on level 0 (Main Program Level) were displayed in the "Program view" screen. This function has now been expanded to allow the active part program on the currently selected program level to be displayed in addition to the level 0 blocks. The function expansion can be selected via MD 5189, bit 6.

The "Program view" screen must be reselected to activate the bit setting.

"Program view" can be opened by selecting the softkey of the same name in the "Automatic" machine area.

The active program on the currently selected program level is displayed in the "Part program" window. The display and cursor proceed in synchronism with the program run.

**CAUTION:**

If the part program cannot be displayed on an active subprogram level owing to a cycle disable, write/read authorization, etc., it will be displayed on the next-higher, enabled program level.

## 3.2 Additional machine functions

### 3.2.1 Overstore

One or more values in the buffer can be modified by means of the OVERSTORE machine function.

You can modify the following data:

- Tool number T...
- Tool offset number D...
- Spindle speed S...
- Auxiliary function H...
- Miscellaneous function M... (up to 3 values).

An invalid D number is not checked in the OVERSTORE mode.

The program will run with these values until a new value appears in the program for the overstore function or until you enter values modified by "Overstore".

Overstoring is only possible when the input window is active.

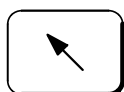
#### Preconditions

- The machine area has been selected.
- The mode group and channel in which you wish to overstore have been selected.
- One of the following four modes has been selected: AUTOMATIC, TEACH IN, MDA, JOG.
- For overstoring, a running program must be stopped.

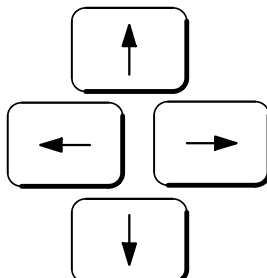
#### Operating sequence



Press the OVERSTORE softkey.



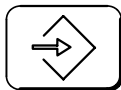
Activate the cursor in the "Overstore" input window.



Now, using the cursor keys, set the reverse video marking on a desired input field.

"Overstore" input window

Overstore			
S	■	=	M =
T		=	
D			
H		=	



Enter the correction values via the keyboard and conclude each value by pressing the input key.





Press the NC START key when all corrections have been carried out.

#### Note

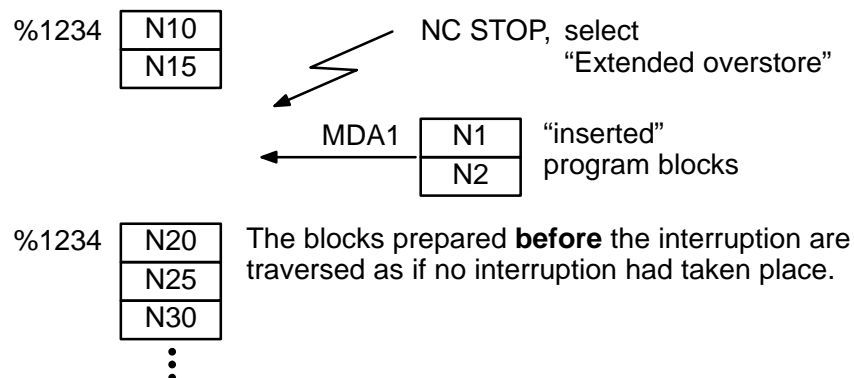
To simplify the input, you can enter the values in sequence. Assignment to the input field is performed automatically (e.g. S500 M4 H12).

### 3.2.2 Extended overstore

	The softkey function “Extended overstore” must be configured by the machine-tool manufacturer.	
---	--	---

“Extended overstore” makes it possible to overstore by executing whole NC program blocks or a single NC program.

With this function, traversing blocks or programs are executed when a channel is in the RESET or stop state by inserting MDA mode without making a mode change or RESET necessary. The program state and point of interruption are “noted”. The overstore program is processed and processing is then continued from the point of interruption.



#### Note

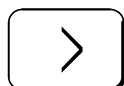
The “Extended overstore” function is enabled in the “Flexible memory configuration” in SW 5.4 and higher.

### Preconditions

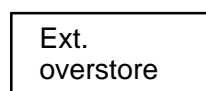
- The machine area is selected.
- The mode group and channel in which you wish to overstore are selected.
- One of the modes JOG, TEACH IN or AUTOMATIC is selected.
- The program must be stopped with NC STOP or RESET.

NC STOP is activated either by pressing the NC STOP key or defined by single block, program step, etc.

### Operating sequence

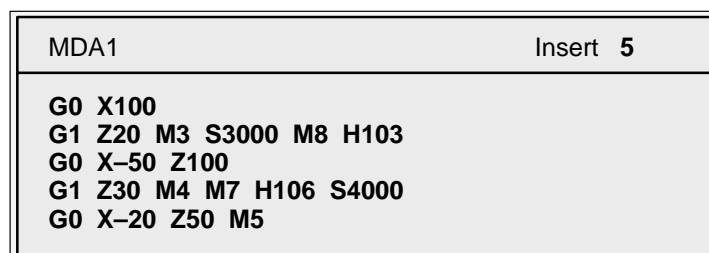


Switch to the extended softkey bar by pressing key ETC.

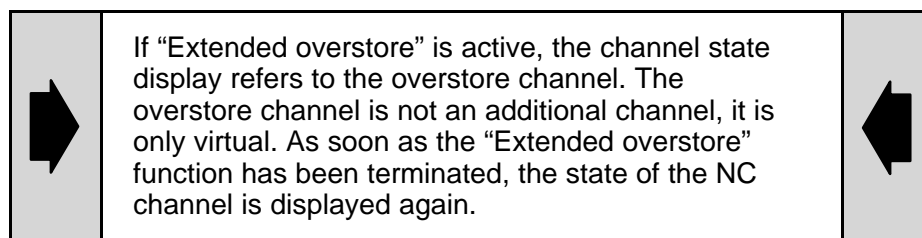


Select EXTENDED OVERSTORE.

The "MDA1" window is inserted in the "Current block" window.

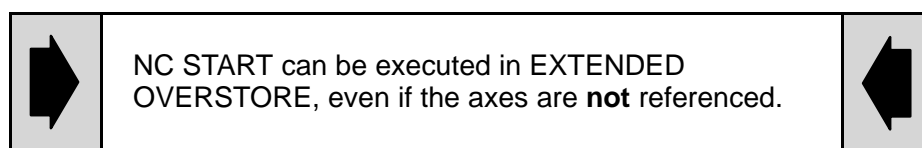


The text "Extended overstore" is inserted on the channel state display.



You can enter the required block sequence in the MDA1 window. The NCK editor is available to you here.

The entered blocks can be executed with NC START. A complete program can be executed by calling a subroutine.



Press the RECALL key to exit "Extended overstore". This action takes you back to the aborted channel. The interrupted program can be continued at the point of interruption.

The “Extended overstore” function **cannot** be terminated as long as the MDA1 program is running. It is possible to leave the menu but the message “End extended overstore (NC Start disabled)” appears.

The MDA1 program must be in the RESET state in order to be able to exit “Extended overstore”.

#### Notes

- “Extended overstore” and “Overstore” are mutually exclusive.
- If no disables have been configured by the machine-tool manufacturer NC START is possible in “Extended overstore” without it being necessary to reference the axes.
- If a RESET or STOP is triggered on a mode change, the MDA1 program is aborted with active “Extended overstore” of this RESET or STOP. The state before “Extended overstore” was activated is not affected. “Extended overstore” is **not** deselected (see channel display).
- If “Extended overstore” is active and another operating mode is selected, “Extended overstore” can be selected again. The MDA1 window which was exited reappears. In this case, “Extended overstore” is only active in the new operating mode and only has to be terminated in that mode.
- The following table shows when certain actions are possible:

Operating mode	Channel state	Possible action
AUTOMATIC, JOG, TEACH IN	Running	No ext. overstore possible
	RESET	Ext. overstore
	NC STOP	Ext. overstore
	Program stop	Ext. overstore
Extended overstore	Running	Termination not possible Simple overstore not possible
	RESET	Termination not possible Simple overstore not possible
	NC STOP	Termination not possible Simple overstore not possible
	Program stop	Termination not possible Simple overstore not possible
Simple overstore	Of no significance	No ext. overstore possible
BA MDA	Of no significance	No ext. overstore possible

- If program control is changed during “Extended overstore”, it is only active for the “Extended overstore”: After returning to the point of interruption, the previous conditions again apply.
- In operating mode TEACH IN, MDA mode in EDIT mode and “Extended overstore” are interlocked.

### 3.2.3 Altering F and S values on-line

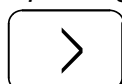
The feedrate value F (axis-specific and path feed) and the spindle speed value S (up to 6 axes) can be altered directly, i.e. "on-line", by entering F and S values directly.

In this way it is possible to enter more precise values than with the override function.

#### Preconditions

- The machine area is selected.
- One of the 4 operating modes MDA, JOG, TEACH IN and AUTOMATIC is active.

#### Operating sequences



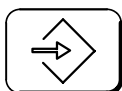
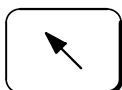
F/S  
on-line

Extend the softkey bar with the ETC key.

Press the softkey F/S on-line to call up the following display

Machine	Parameter	Programm.	Services	Diagnosis
13:07				
AUTOMATIC Program reset				Mode Gp 1 Channel 1
Actual value		Feed/dist.-to-go		Program pointer
X	-76.006	0.000	PART MPF13	
Y	-61.440	0.000	0	
Z	0.000	0.000		
USER/ACK/MPF				
Name	Type	Length	Mark	Comment in the 1st block/date
MPF1012	part	231	-rwx	Z-No.: 34K100/12
MPF11	part	299	-rwx	WKS-No.: 3
MPF13	part	564	-rwx	MPF
MPF14	part	442	-rwx	
MPF22	part	159	-rwx	GST
MPF23	part	106	-rwx	Portal
Select prog./job				

Fig. 3.13 Changing F/S values



Press the home key to switch to the S value and F value window. The active window is marked (with a yellow frame). Now enter the desired F or S value in the input line.

Transfer the entered value into the selected screen form with the INPUT key.

The values are active as soon as they are entered. Only the value for the F value is overwritten, the feed type cannot be altered.

If a block is executed with a new F or S value, this deletes the on-line value. The programmed value becomes active again and the value in the input screen form is reset to "0".

The on-line value is also reset with RESET

#### Note

A new feedrate for block-related simultaneous axes is only effective for the current block.



You leave the input screen forms for the function S/F ON-LINE with RECALL. The values entered remain active until they are again altered (see above).

### 3.2.4 Program modification

#### Preconditions

- The machine area has been selected.
- The mode group and the channel in which you wish to modify programs have been preset.
- One of the following modes is active: AUTOMATIC, TEACH IN or MDA.

#### Note

In JOG mode, program modification is only active if there has been a changeover from AUTOMATIC or MDA to JOG and the system is in the “Program stop” operating state.

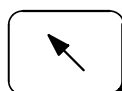
#### Operating sequence

Program  
modific.

Press the PROGRAM MODIFICATION softkey. The “Program modification” screenform appears on the CRT.

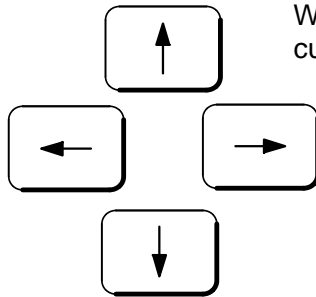
Machine	Parameter	Programm.	Services	Diagnosis
				13:40
AUTOMATIC		Program reset	DRY	Mode Gp 1 Channel 1
Actual value	Feed/dist.-to-go		Program pointer	
X	0.000	0.000	WELLE1 MPF1012	
Y	0.000	0.000	0	
Z	0.000	0.000		
Program modification				
Dry run feed	DRY	yes	Skip block	SKP/ no
Program stop	M01	no	Block skip level 1	SKP/1 no
Rapid override	ROV	no	Block skip level 2	SKP/2 no
Axis converter	ACR	no	Block skip level 3	SKP/3 no
DRF offset	DRF	no	Block skip level 4	SKP/4 no
Decode single block	DSB	no	Block skip level 5	SKP/5 no
Program step	PST	no	Block skip level 6	SKP/6 no
Stop at target block	BRK	no	Block skip level 7	SKP/7 no
Clear	CLR	no	Block skip level 8	SKP/8 no
Execute from external	EXT	no	Automatic save	SAV no
Prgrstart/end cyc from	NCY	no	Predoc.blocks	7 maximum 8
Select with the selection key				
Overstore	Preset DRF	Program modific.	Program correction	Select prog./job

Fig. 3.14 Program modification



Use the window switchover key to switch to the “Program modification” screenform. The window currently active is marked (by a yellow frame).

## 3.2.4 Program modification



With the cursor keys you select the program modification function. A cursor marks the current position.



The functions DRY, M01, ROV, ACR, DRF, DSB, PST, BRK, CLR, EXT, NCY, SAV and SKP can be selected/deselected in a toggle field by selecting yes/no.

All active functions are displayed in the status field with the corresponding abbreviation. The SINGLE BLOCK function acts mode group-specific. All other functions act channel-specific. They remain active until they are deselected or until the control is switched off.

From SW 6, a tick (✓) is added in the "Program modific." screen as reconfirmation.

The tick (✓) indicates that the PLC has carried out the "Program modific."

If the tick (✓) is missing, the PLC has not (yet) carried out the desired "Program modific.". The PLC can also execute a program modification automatically, as can be seen from the example of rapid override (ROV).



### 3.2.4.1 Description of the functions DRY, M01, ROV, ACR, DRF, DSB, PST, BRK, CLR, EXT, NCY, SAV, SKP and predec. blocks

This machine function can be used to set the following parameters and thus modify program execution correspondingly:

*DRY*

**Dry run feedrate;** Display: DRY (DRY RUN):

All those blocks in which a feedrate is programmed (G01, G02, G03, G33, G34, G35), are traversed at the "Dry run feedrate" preset by a setting data instead of at the programmed feedrate. The dry run feedrate then also applies in place of the "revolutional feedrate" G95 and the feedrate for thread cutting.

*M01*

**Programmed stop;** Display M01 (M function):

If an "M01" is present in the part program, the program is stopped. If the function is marked "NO" the "M01" is ignored

*ROV*



**Rapid override;** Display: ROV (RAPID TRAVERSE OVERRIDE)

If "NO" is set, the feedrate/rapid override switch is only active for "Feedrate". If the softkey function is set to "YES", the override set at this override switch is also active for rapid traverse.

*ACR*

**Axis converter;** Display: ACR (Axis Converter)

ACR is an optional function to convert existing axis addresses in a part program in other axis addresses. A screenform is available in the PARAMETER/SETTING DATA area for entering the names.



	This function must be enabled via machine and setting data.	
---	---	---



*DRF*

**DRF;** Display DRF (Differential Resolver Function).

You have enabled "DRF" (Differential Resolver Function). It is now possible to traverse an axis which has still to be selected, with the handwheel in AUTOMATIC mode.

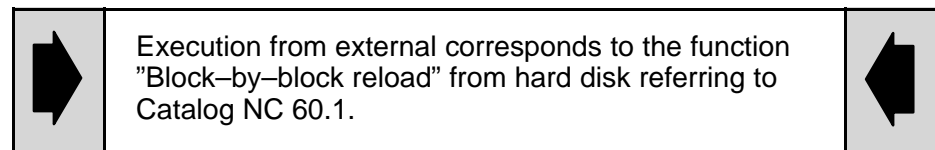
<i>DSB</i>	<p><b>Decoding single block;</b> Display DSB (DECODING SINGLE BLOCK). After NC START, only one block is executed, even if it does not contain a traversing movement or an auxiliary function (e.g. R parameter calculation).</p> <p>The signal acts at the end of the block preset by being decoded in the current part program.</p>
<i>PST</i>	<p><b>Program step;</b> Display: PST</p> <p>PST is a type of single block mode. Main program blocks are executed as individual blocks. A subroutine with all its nested subroutine levels is executed as a single unit. During subroutine execution the advance decoding is enabled. Machine tool compensation and contour definition are therefore possible within subroutines. When you return from the subroutine to the main program level, the advance decoding is switched off and the block sequence is interrupted. The main program block following the subroutine call is not executed. When PST is active, DSB and BRK are switched off and vice-versa (1 of 3 principle).</p>
<i>BRK</i>	<p><b>Stop at target block;</b> Display: BRK (break)</p> <p>This feature allows you to interrupt the program at a defined point. The target block is selected by means of the</p> <ul style="list-style-type: none"> <li>• Cursor functions and</li> <li>• MARK TARGET BLOCK softkey</li> </ul> <p>When the target block has been executed, the advance decoding is switched off and the block sequence is interrupted. When BRK is active, PST and DSB are switched off and vice-versa (1 of 3 principle).</p>
<i>CLR</i>	<p><b>Clear;</b> Display: CLR</p> <p>CLR is <b>only</b> effective in MDA mode (not in the MDA mode in TEACH IN mode). If CLR is active, the entered MDA blocks are deleted after program end, reset or mode change. If CLR is not active, the MDA blocks are retained.</p>

	The program modification signals (except for ACR) do not act directly, i.e. they can be configured by the machine manufacturer via the PLC program. See also the machine manufacturer's documentation.	
---	--	---

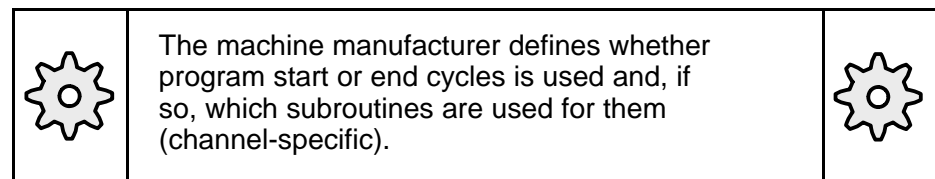
	When executing a program from the hard disk (EXT), the message 110114 (MPF ... not entirely transferred) may appear if the program is aborted with RESET. In this case the alarm has no effect and can be ignored.	
---	--	---

**EXT****Execution from external;** Display: EXT (Option)

This function makes it possible to execute an NC program via computer link (see also the job list description).

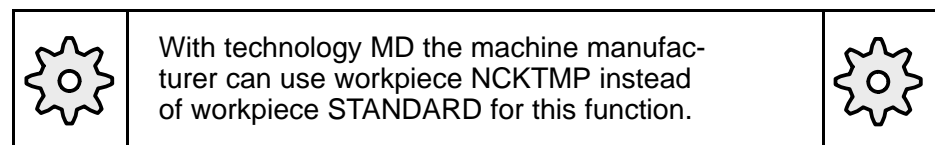
**NCY****Program start / End cycles off;** Display: NCY (no cycle)

Automatic execution of program start and end cycles can be disabled with this function.

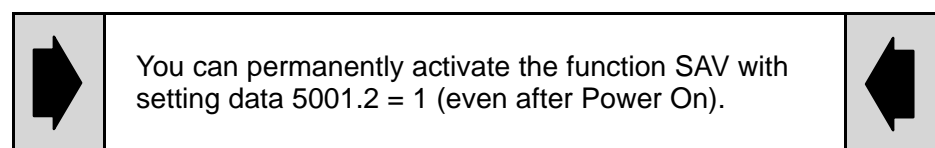
**SAV****Automatic saving;** Display: SAV in the NCK editor

If this function is active a backup copy of the part programs in the NCK is automatically made in the relevant workpiece after they have been edited (e.g. with softkey: PROGRAMS/Edit or MACHINE/Program structure).

If no workpiece is known (new program), the edited program is stored in director LOCAL / STANDARD.



To assign an original workpiece, use the softkey SAVE TO DISK and enter the workpiece name in the input line.

**Limitations:**

Automatic saving is not active in TEACH IN mode. A program is only saved if it has been altered or if you exit the editor or switch areas.

**SKP****Skip block;** Display: SKP:

Blocks in the program which are marked by an oblique stroke in front of the block number ("N...") are not taken into account when the program is run.

In addition to this skip block function, there are another 8 skip block levels /1 .../8.

The skipping criteria can be entered before program execution is started. They can also be configured by the machine manufacturer via the PLC program. See also the machine manufacturer's documentation.

*Predec. blocks* Displays how many blocks have been precalculated.

*Maximum* Here, the number of precalculated blocks is limited.  
Application: Faster decoding in single block mode.

### 3.2.4.2 Single block/Decoding single block

*Preconditions* The same preconditions apply as described in the previous section.

*Operating sequence*



The SINGLE BLOCK function is selected if you press the appropriate key on the machine control panel.



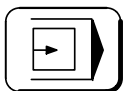
When the NC START key is pressed, one block is executed.

The SINGLE BLOCK function has no effect on arithmetic blocks. Arithmetic blocks are part program blocks which perform programmed calculations without outputting them to the machine or the PLC (R parameter operations).

The signal generated takes effect at the end of the current block. Once you have activated the SINGLE BLOCK function, the "SBL" display appears in the status field and the LED for the SINGLE BLOCK key on the machine control panel lights up.

After executing a block, "Stop single block operation" is displayed.

NC START must be pressed again to continue execution.





Pressing the SINGLE BLOCK key on the machine control panel a second time, deselects the function.

The table below shows with which blocks the "Decoding single block active" signal or the "Single block active" signal must be preset if a program is to be executed block by block.

The decoded single block mode can be activated using the DEC program modification. In contrast to single block mode (on the machine control panel) decoding is interrupted after every block. Each block (including calculation blocks) is decoded individually.

Block type	Single block	Decoding single block
Traversing blocks	X <sup>1)</sup>	X <sup>1)</sup>
Blocks without position data	X <sup>1)</sup>	X <sup>1)</sup>
Arithmetic blocks		X <sup>1)</sup>
Switching and auxiliary function blocks	X <sup>1)</sup>	X <sup>1)</sup>
Blocks generated within the control / CRC...) <sup>3)</sup>	X <sup>1)</sup>	X <sup>1)</sup>
Threading blocks without dry run feedrate	2)	X <sup>1)</sup>
Threading blocks with dry run feedrate	X <sup>1)</sup>	X <sup>1)</sup>
<ul style="list-style-type: none"> <li>• Blocks can no longer be stopped if they have been preprocessed in the buffer memory but not yet executed without the “Decoding single block” being present.</li> <li>• A “Decoding single block” can be modified by “Overstore”.</li> </ul>		
<hr/> <p>1) Single block  2) Single block: a stop comes into effect only at the end of a current block without thread.  3) With tool radius compensation</p>		

	With some functions (e.g. contour definition, tool radius compensation, soft approach to contour) additional blocks are inserted by the control. Depending on the number of insertions, the NC START key may have to be pressed several times.	
---	--	---

If the SINGLE BLOCK function has been deselected, the LED on the machine control panel goes out.

#### Note

Decoding single block:

In traversing blocks, and in particular when tool radius compensation is applied or in blocks containing only auxiliary functions, a “misleading” CURRENT BLOCK DISPLAY can occur because both the preprocessed and the traversed blocks are displayed.

#### Application

You wish to test a program that contains blocks only with R parameter calculations or CL800 (= @-) commands.  
Activate “Decoding single block” to check the results of arithmetic operations block by block.

### 3.2.5 Block search

#### *Preconditions*

- The machine area has been selected.
- AUTOMATIC mode has been selected.
- The control is in the reset state.

#### *Operating sequence*



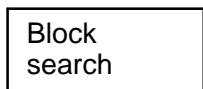
On pressing the extension key (ETC) in the AUTOMATIC or TEACH IN basic displays, the BLOCK SEARCH softkey is displayed.



On pressing the HELP key, the softkey functions are displayed.



Switch to the search window (if the window is active, it is marked).



Press the BLOCK SEARCH softkey. The following options are displayed:

- Block search with calculation
- Block search without calculation
- Block search with calculation from the last main block

You can use the block search functions to reach a breakpoint in a program (e.g. after a program error, tool break, etc.). This causes the program to be decoded until the breakpoint is reached. It is not necessary to process the entire program.

If the position at the breakpoint has been left in JOG mode (e.g. during a tool change), you can use the JOG/handwheel or REPOS function to travel back to the position.

#### *Note*

Only the functions

- Block search with calculation
- Block search with calculation from the last main block

are provided in TEACH IN mode.

- Output of auxiliary functions during block search:

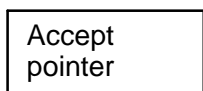
Auxiliary functions that are output to the interface during the block search are not shown in the AUTOMATIC basic display.

If you press the BLOCK SEARCH softkey the following search window is displayed:

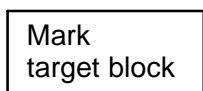
Search destination / Breakpoint			
0	MPF1	N30	
1	SPF20	P3	N45
2			
3			
4			
5			
6			
7			



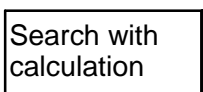
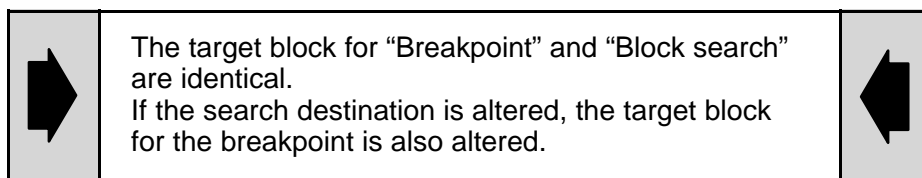
You can enter the search destination (program ID) the number of passes (for subroutines) and the breakpoint (block number) in the input boxes. Enter the line numbers if the program has no block numbers. The block search is performed with block and line numbers.  
(e.g. N10 : Block N10  
3 : Line 3)



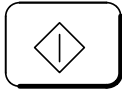
The ACCEPT POINTER softkey automatically enters the block in front of the breakpoint. The block search (after completion of a program correction or tool change) takes you to this precise position.



During program correction in the AUTOMATIC or TEACH IN mode, the function MARK TARGET BLOCK accepts the block underneath the editing cursor in the search pointer as the search destination.



Press the SEARCH WITH CALCULATION softkey. The block search is carried out by the control.



After NC START the program is started and executed.

During block search with calculation, the same calculations are carried out as in normal program operation but the axes do not move.

It is also possible to perform a block search in a nest of subroutines. For this, you enter in the main program, the number of the block in which the subroutine call is programmed. Then, you input the number, the number of passes and the block number of the subroutine and operate the softkey. Operating NC START triggers a compensatory movement with linear interpolation to the end point of the target block. When doing so, observe the feedrates and axis compensations configured by the machine manufacturer. If the target block is a block with G4 dwell, this is not activated. The compensatory movement then goes to the last position programmed. If G00 from the program is active in a target block, the compensatory movement is executed with G00. The display gives the G command which is active for the program.

In blocks with contour definition, tool radius compensation or soft approach to contour, the PROGRAM START key must be pressed several times before the axis moves or the message "Block search terminated" is displayed (cause: e.g. calculation of the intersection point of the tool radius compensation requires at least two blocks). With decoding single block it is possible that several blocks are traversed when pressing the PROGRAM START key once (e.g. tool radius compensation).

*Note*

Since the coordinate blocks of the spline blocks (spline interpolation) can only be programmed with block numbers, a block search with calculation can only be performed for coordinate blocks and **not** for coefficient blocks.

*DSB and block search*

Two applications are significant regarding the "Search with calculation" and "Search from main block" functions in conjunction with the Decoding Single Block (DSB) function:



- DSB function activated **before** search start

All blocks including the target block are calculated. The blocks after the target block are not decoded in advance. It is therefore possible to edit blocks directly after the target block. These blocks are effective as soon as NC START is pressed. Arithmetic blocks following the target block are displayed after NC START. If the functions "Contour definition", "Soft approach to/from the contour" and/or tool radius compensation are active, the signal "Block search terminated" is not output. The signal is not output before the next NC START.

- DSB function activated **after** reaching the target block **before** NC START.

Decoding is performed beyond the target block. Editing blocks following directly after the target block is not possible. Arithmetic blocks following the target block are not displayed after NC START.



	In the case of block search with calculation, C axis operation must be selected or deselected by means of overstoring before or at the latest in the target block.	
---	--	---

#### Search without calculation

Press the SEARCH WITHOUT CALCULATION softkey. The block search is carried out by the control.



After NC START the program is started and executed.

The target block (main block and subblock) must be in the main program level. Once the target block has been found, it is adopted as the current block.

#### Search from main block

Press the SEARCH FROM MAIN BLOCK softkey. The block search is carried out by the control.

The function includes a block search without calculation to the last main block before the target block and, from there on, a block search with calculation to the target block.







After NC START the program is started and executed.

The main block in question (= starting block of the search, e.g. :0010) must be in the main program.

The target block, however, will also be found in the subroutine; it can be both in the main block (e.g. :0045) and in the subblock (e.g. N0045).

Everything else is the same as for SEARCH WITH CALCULATION.

	It is optionally possible to set the output of the auxiliary function for BLOCK SEARCH by machine data. Depending on the setting on installation, the H, M, S, and T functions are output completely or partially or are completely suppressed.	
---	---	---

	If you want to perform a block search (e.g. to reposition) after an interrupted turning-milling operation (TRANSMIT) (e.g. because of a tool break), you must move the tool completely behind the turning centre before you start the block search.	
---	---	---

#### Note

When using a block search with TRANSMIT you should ensure that both the tool tip and the tool holder are positioned behind the turning centre (positive X direction!) before you start the block search. Otherwise the TRANSMIT machining operation might start in front of the turning centre only to be aborted with the alarm "Speed setpoint alarm limit".

This problem only arises with angle head cutters.

## 3.2.6 Program correction

If the control recognizes a programming error during execution of a program, the program sequence is stopped and a corresponding alarm is displayed.

If the program modifications BRK, PST or DSB are used to interrupt a program, the advance decoding terminates at the breakpoint. Any of the blocks following the breakpoint block can be edited. Pressing NC START causes execution of the program to begin after the breakpoint block, i.e. any changes become effective immediately. To use these program breakpoint features, you must define the breakpoint before starting the program. To do this, select the program modifications before pressing NC START. The target block must be marked before you select the function BREAKPOINT (BRK).

All other program interruptions (such as M00/M01, NC stop, feed stop, single block, etc.) do not interrupt the advance decoding.



It is only possible to edit at the breakpoint in these cases if the program execution has been aborted by pressing the RESET key. This resets the entire decoding. When you press NC START, execution begins from the start of the program.

### Note

Tool path  
simulation

This softkey is displayed in the PROGRAM CORRECTION basic display (AUTOMATIC mode).

This function is an option and is described in Section "Graphic Tool Path Simulation".

### Preconditions

- The machine area has been selected.
- AUTOMATIC or TEACH IN mode has been selected.
- The program sequence is stopped.  
Notes regarding the programming error are displayed in the alarm line.

### Operating sequence

Program  
correction

Press the PROGRAM CORRECTION softkey.

Program correction display

MPF 4711	Workpiece SHAFT1	Insert 4
N5	G0 X0 Y0 Z0 L <sub>F</sub>	
N10	G1 F5000 X-80 L <sub>F</sub>	
<b>N15</b>	<b>G32 X-80 Y0 I80 J-60 L<sub>F</sub></b>	
N20	G0 X0 Y25 Z1 L <sub>F</sub>	
N25	G01 Z-200 F500 L <sub>F</sub>	

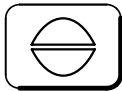
The block with the breakpoint or error is highlighted (N15 in the figure). If an error causes the program to be interrupted, the advance decoding is interrupted before the block containing the error. You can edit the block with the error and all the following blocks.

*Note*

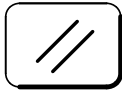
If the error is not detected until machining is started (e.g. tool radius compensation), the incorrect block is not marked but is shown by the block number in the alarm.

*Problem, remedy*

Neither a block is marked, nor a block number displayed in the alarm. Add block numbers to the program section in question and restart the program.



If a program is interrupted by a CANCEL alarm, execution of the program can be continued from the breakpoint by pressing NC START after the program error has been eliminated. When you press the CANCEL key you acknowledge the error message.



If a program is interrupted by a RESET alarm, the program can only be started after the RESET key is pressed. Execution begins from the start of the program.



*Note*



After a RESET interruption, you can use the block search function to calculate the breakpoint and go to this point. This means the complete program does not have to be processed (see the section "Block search").

You will find a description of the NCK editor in the section "Programming in the NCK memory area". Please refer to the section TEACH IN for the operator actions for setting a breakpoint.



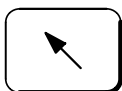
After the RESET key is pressed, you can edit anywhere in the entire part program. The selection of the correction block is cancelled.

	When a program is processed externally the program correction screen cannot be displayed.	
---	---	---

	The RESET function can be configured by the machine manufacturer. The RESET is generated internally, if required, so that acknowledgment by means of the key is not required. Read the machine manufacturer's documentation.	
---	--	---

*Note*

The PROGRAM CORRECTION function enables you to edit part programs in the machine area at any time (except while a program is running).



In the modes AUTOMATIC, TEACH-IN and MDA you can switch over to the part program input window and edit. The active window is marked (yellow frame).

## 3.2.7 Saving programs

In the TEACH IN and AUTOMATIC modes you can transfer programs from the working memory to the hard disk by means of the SAVE TO DISK and SAVE UNDER softkey functions.

### Preconditions

- The machine area has been selected.
- The TEACH-IN or AUTOMATIC mode has been selected
- There is a part program in the working memory

Program  
correction

Press the PROGRAM CORRECTION softkey in AUTOMATIC mode.

The softkeys for saving part programs are to be found in this operating branch because only here, i.e. in AUTOMATIC mode, can programs be altered. It is therefore only possible to save programs here.

The softkeys for saving part programs are to be found in the operating mode TEACH IN in the extension of the softkey bar which is called up with the ETC key.

A program can be saved in two ways.

The program can either be saved in the current workpiece, that is the SAVE TO DISK function or it can be saved using the DATA SELECTOR in any (legal!) directory. This function is called SAVE UNDER.

Save to  
disk

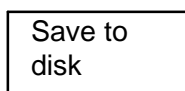
The softkey is to be found in the basic display of PROGRAM CORRECTION (operating mode AUTOMATIC).

Block search	Tool path simulation	Display block	Mark target block	Insert/overwrite	Search	Save to disk
--------------	----------------------	---------------	-------------------	------------------	--------	--------------

If the softkey SAVE TO DISK is pressed, the message “Store in workpiece?” appears and the current directory and workpiece is displayed.

Machine	Parameter	Programm.	Services	Diagnosis
				11:36
AUTOMATIC Program reset		DRY		Mode Gp 1 Channel 1
Actual value		Feed/dist.-to-go	Program pointer	
X	0.000	0.000	WELLE1	
Y	0.000	0.000	0	
Z	0.000	0.000		
Act. value		Override	Position	Set value
F=	0.00	95%		F= 0.00M
S 1=	0	50%	0.00	S 1= 0
WELLE1/MPF1012		Insert	1	
Z-Nr.: 34K100/12 )%				
( 25.10.94 Rn )%				
N0010 G90 G00 Z%				
N0015 Y0%				
Save / edit program?		WELLE1		
Block search	Tool path simulation	Display block	Mark target block	Insert/overwrite
		Search	Save to disk	

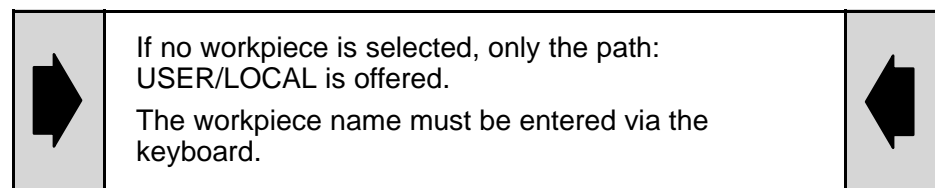
Fig. 3.15 Save 01



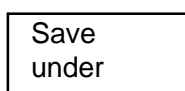
Press the softkey SAVE TO DISK again to save the program. While the program is being saved the message “Transfer running” is displayed.



If you press RECALL instead, the procedure is aborted without data transfer.

**Note**

You can also activate the function SAV (automatic save); See Program modification.



The softkey SAVE UNDER is to be found in the menu extension of the PROGRAM CORRECTION (AUTOMATIC mode). Now save using the DATA SELECTOR.

Block search	Tool path simulation	Display block	Mark target block	Insert/overwrite	Search	Save to disk
Cut						Save under

Now press the SAVE UNDER softkey to display the data selector.

Machine	Parameter	Programm.	Services	Diagnosis
				13:18
AUTOMATIC NC Stop				Mode Gp 1 Channel 1
Actual value	Feed/dist.-to-go		Program pointer	
X	163.413	0.000	PART MPF1012	
Y	150.000	0.000	MPF1012 N40	
Z	100.000	0.000		
USER				
Save from: USER/NC/MPF/MPF1012				
Save to:				
Name	Type	LengthMark	Comment in the 1st block/date	
LOCAL	parts			
NCK	nck			
Save to disk				

Fig. 3.16 Save 02

The workpiece can now be selected with the cursor in the LOCAL directory in which the program is to be saved.

Is also possible to enter a new workpiece name.

Save to  
disk

Now the softkey SAVE TO DISK must be pressed in order to save the program.

The message "Store in workpiece?" appears. The program is saved by pressing the softkey SAVE TO DISK again.

The message "Transfer running" is displayed while the program is being saved.



It is only possible to save in the LOCAL directory. It is possible to save programs from GLOBAL with soft-key: Services/NC/Save Start.



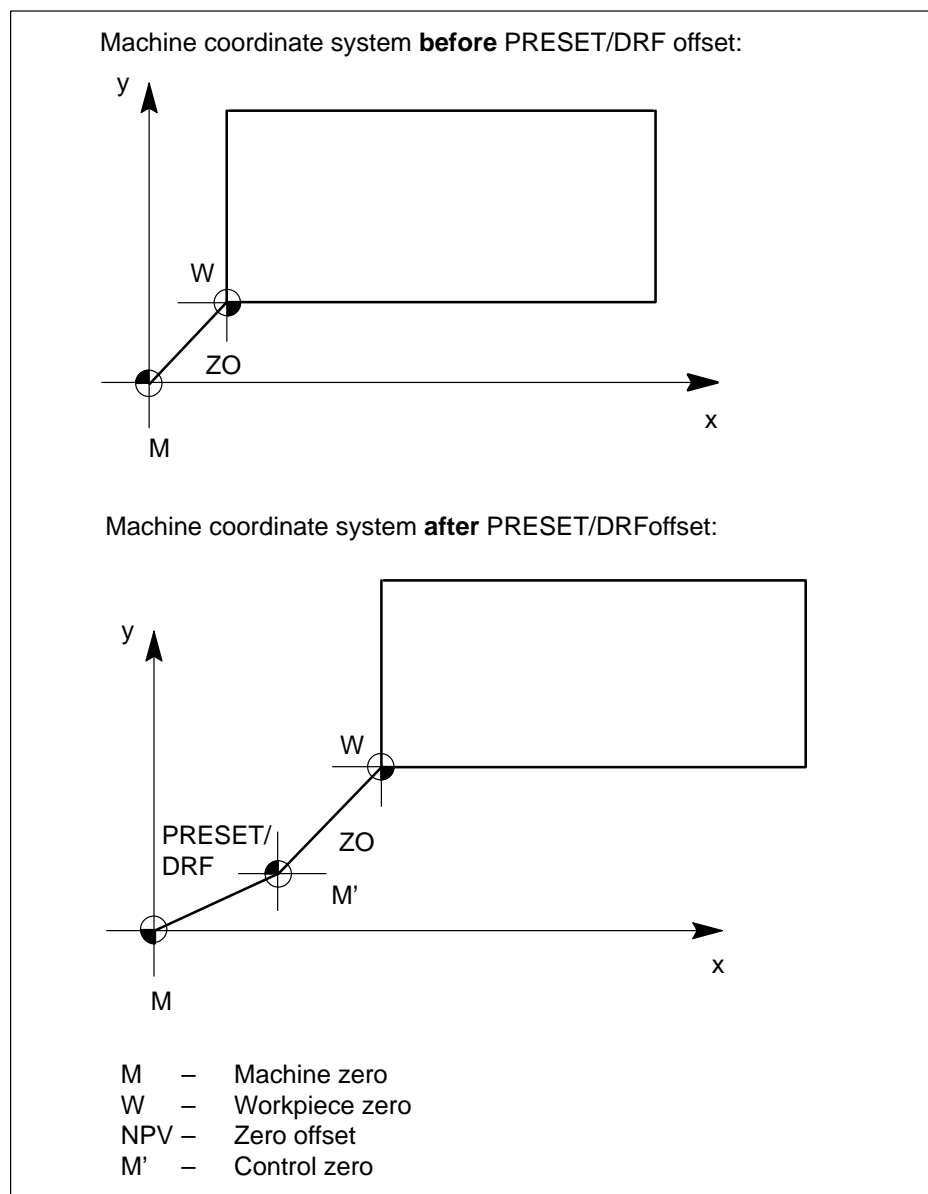
If the correcting and saving functions cause problems (e.g. when you reload the workpiece the old program version is called up again), check the job list. Refer to the notes in Section "Creating job lists / Syntax description for job lists". Pay special attention to the LOAD / GLOBAL / ... commands.



### 3.2.8 PRESET (Set actual value)/DRF

#### 3.2.8.1 PRESET – Offset

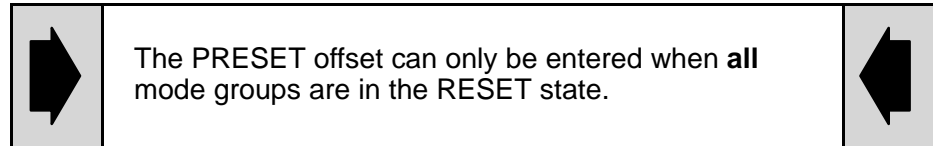
In the PRESET machine function, you can offset the control zero to any point in the machine coordinate system.



The PRESET/DRF offset is not visible in the actual value display.  
The software limit switches and the working area limitation act as in the original coordinate system, i.e. they are not offset.



## 3.2.8 PRESET (Set actual value)/DRF



Input a new actual value. The actual value memories of all axes present can be preset.

The presetting allows the control to calculate a PRESET offset which is displayed on the screen in the input window PRESET/DRF.

Optionally, a tool offset can be included in the calculation of PRESET offset.

Enter the tool offset data (correction number, correction direction and identifier) in front of "Set actual value".

When calculating the tool length compensation the identifier must be entered. Generally, the values 2 or 3 are entered here. Tool length compensations P2 or P3 are then calculated in the offset (see PARAMETER area).

The value then input is transferred to the actual value memory, taking account of the tool offset.

*Preconditions*

- The machine area has been selected.
- AUTOMATIC mode has been selected.
- The control is in the RESET operating state.

*Operating sequence*

Press this softkey to select the machine function.

*Including the tool offset*

In front of PRESET offset please enter the

- Identifying number
- Correction number and
- Correction direction.

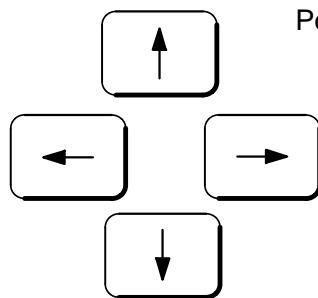


Switch to the upper window, if necessary.



*Input window PRESET/DRF*

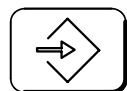
Preset	ID No.	DRF
0.000	0	0.000
0.000	0	0.000
0.000	0	0.000
TO area		
	1	
Correction number	0	0
Correction direction		0
(0 = pos./1 = neg.)		



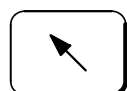
Position the cursor to the input field for the ID number.



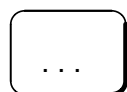
Enter the appropriate value via the keyboard.



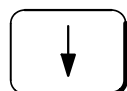
Confirm your input by pressing the INPUT key.



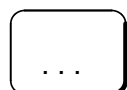
Switch to the lower input window.



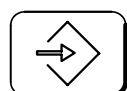
Enter the value for the correction number D... via the keyboard.



Position the cursor on the correction direction input field.



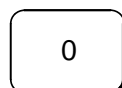
Enter the appropriate value (0 = positive, 1 = negative) via the keyboard.



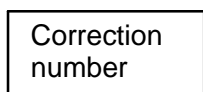
Confirm your input by pressing the INPUT key.

## 3.2.8 PRESET (Set actual value)/DRF

*Without including the tool offset*



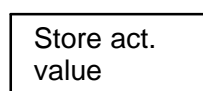
Enter the value 0 and press the CORRECTION NUMBER softkey.



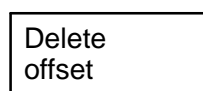
After having selected this data, enter the actual PRESET offset.



Enter the axis name and the new actual value for the axis.



Conclude the input by pressing the STORE ACTUAL VALUE softkey.



The DELETE OFFSET key is used to clear all PRESET offsets of the selected mode group.

The offset displayed includes the input value and the included tool correction.



If the "Help" key is pressed once the function has been selected, information on the function is displayed on the screen.

The PRESET offset remains stored:

- After "End of program"
- After RESET.

	<p>Machine data is used to determine whether the PRESET offset is deleted automatically:</p> <ul style="list-style-type: none"> <li>• When the control is switched on</li> <li>• When the reference point is approached (See the machine tool manufacturer's documentation).</li> </ul>	
--	---	--

### 3.2.8.2 DRF offset

The DRF offset allows an additional incremental zero offset to be made by the handwheel.

	<p>The DRF offset is displayed on the screen in the DRFinput window. In the actual value display, however, the DRF offset is <b>not</b> visible.</p>	
--	--	--

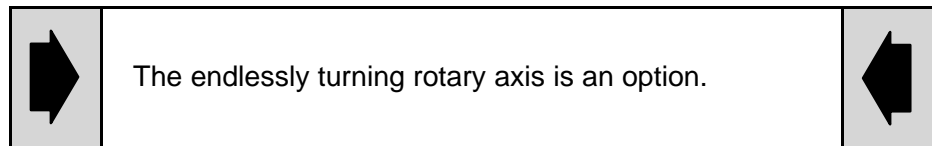
An active DRF offset cannot be deleted via the program (@434) but must be moved out again.

### 3.2.9 Axis-specific G functions

This is purely a display function.

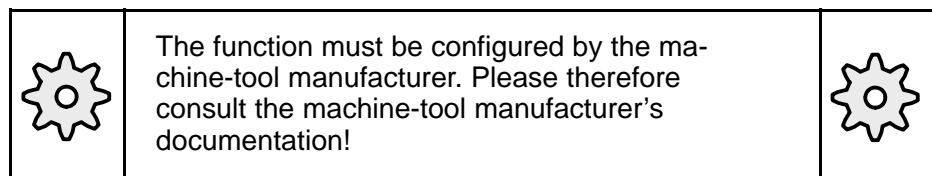
The “axis-specific” G functions are displayed instead of the “program pointers”.

The endlessly turning rotary axis is programmed with these G functions. This function is relevant for hobbing.

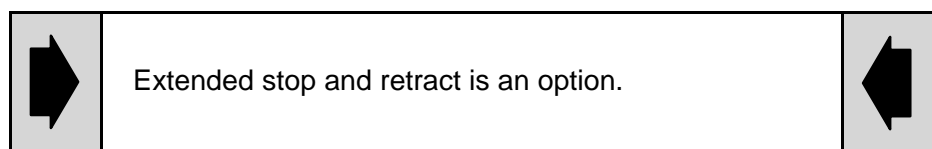


The G functions

- G 27 Handwheel override
- G 105 Endless rotation OFF
- G 103 Endless rotation ON clockwise
- G 104 Endless rotation ON counter-clockwise
- G 119 Oriented STOP
- G 195 Simultaneous feedrate F in mm/inch/degrees per revolution with reference to the speed/feedrate setpoint of the defined spindle or rotary axis.
- G 295 Simultaneous feedrate F in mm/inch/degrees per revolution with reference to the speed/feedrate actual value of the defined spindle or rotary axis.
- G 98 Simultaneous feedrate F in revolutions per min.





### 3.2.10 Extended stop and retract



The EXTENDED STOP AND RETRACT function is used to protect the workpiece and machine.

Responses such as approaching a programmable retraction position, stopping axes or the output of hardware signals can be triggered to detect certain faults.

## 3.2.10 Extended stop and retract

	<p>The function must be configured by the machine-tool manufacturer.</p> <p>It is, for example, dependent on the drives used.</p> <p>Please consult the machine-tool manufacturer's documentation!</p>	
---	--	---

Activation or deactivation of the monitoring and the response **can** be programmed via G commands.



Stop/  
retract

Press the STOP/RETRACT softkey to display the function-specific G functions (e.g. G425 G91 X10 F10000) with block functions.

This is only a display. It is not possible to enter G functions here.

The G functions:

- G420 Switch off; switch off "Extended stop and retract", all or selectively for axes and/or spindles.
- G421 Switch on; Activate monitoring sources and enable responses.
- G422 Configure generator mode
- G423 Configure stopping by control
- G424 Configure stopping by drive
- G425 Configure retraction by control
- G426 Configure retraction by drive

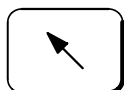
	<p>The EXTENDED STOP AND RETRACT function must be preconfigured.</p> <p>Please consult the machine-tool manufacturer's documentation!</p>	
---	---	---

### 3.3 Multichannel display

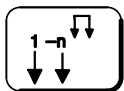
If the option "Double slide display" is active a maximum of 2 channels of the selected mode group are displayed on the screen. The double slide display is active in the JOG and AUTOMATIC operating modes.

Machine	Parameter	Programm.		Services	Diagnosis	
						14:39
AUTOMATIC Program running			Mode Gp 1 Channel 1	AUTOMATIC Program reset		Mode Gp 1 Channel 2
FST				ROU		
Program pointer				Program pointer		
WELLE1 MPF1012				WELLE1 MPF23		
MPF1012 N30				0		
Act. value Distance to go				Act. value Distance to go		
> X 0.000 150.000				X1 0.000 0.000		
> Y 0.000 150.000				Y1 0.000 0.000		
Z 0.000 0.000				Z1 0.000 0.000		
Act. value Overr. Pos. Set value				Act. value Overr. Pos. Set value		
F= 0.00 0% F= 11313.50M				F= 0.00 100% F= 0.00M		
S 1= 0 85% 0.00 S 1= 0				S = 0.00 85% S = 0.00M		
Current block				Current block		
N0025 G04 X2%						
Y0030 G90 G00 X150 Y150%						
N0035 Z100%						
Overstore			Preset DRF	Program modif.	Current G function	Program correction
				Auxiliary functions	Select prog./job	

Fig. 3.17 Multichannel display in AUTOMATIC mode



The "Home" key is used to switch from one channel display to the other. The channel display ("Active window") currently active is marked (e.g. yellow frame).

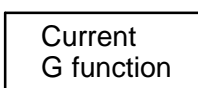


With the channel change key you switch through the channels in the active window.



With the mode group change key you change the mode group displayed.

There are additional softkeys in the AUTOMATIC display in conjunction with the multichannel display:



Instead of the display "Current block", the current G functions are displayed.

#### Note



From SW 6, the "Current G function" has been moved to the ETC menu bar. The "Program view" softkey has been installed in this position instead.



Auxiliary  
functions

Instead of the display “Current block”, the auxiliary functions are displayed.



The RECALL key is used to return to the display “Current block”.

	<p>Settings in the MD define which axes and spindles are displayed in each channel window (configured by the machine manufacturer).</p> <p>The manufacturer can also configure displays for more than 2 channels of a mode group (WS 800 A).</p>	
---	--	---

	<p>The machine manufacturer defines the effect of keys and rotary switches of the machine control panel on individual channels. For example, it is possible (on a machine control panel with several mode groups) to have the mode group keys influence the displayed mode group.</p> <p>Please consult the machine manufacturer's documentation regarding these points.</p>	
--	--	--

END OF SECTION

## 4 Parameter

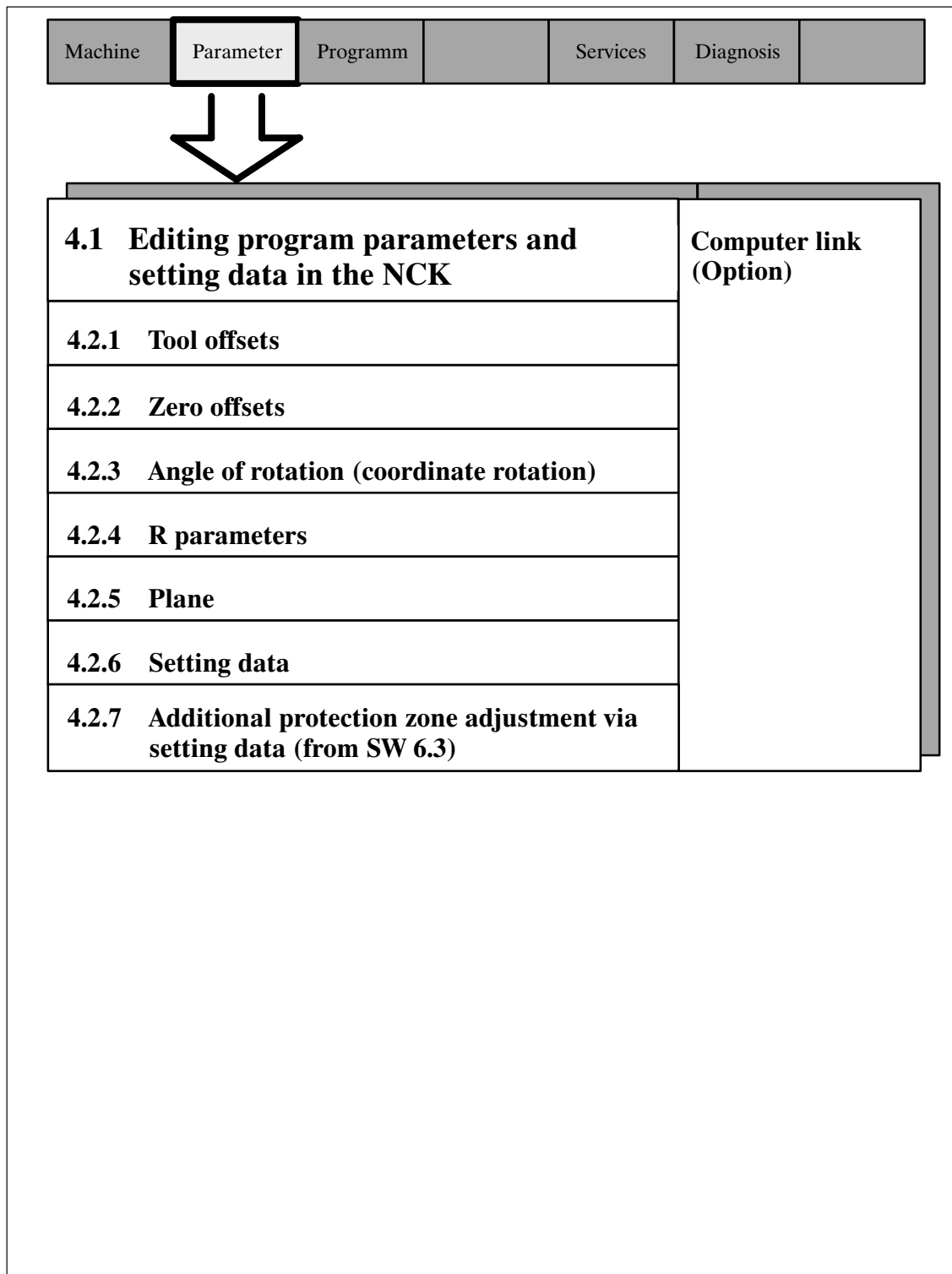


Fig. 4.1 Parameter

Parameter

You activate this area of the menu with the **PARAMETER** softkey in the area menu bar.

The **PARAMETER** area includes the functions displayed below:



Fig. 4.2 Basic display, parameter area

The **PARAMETER** area includes the program parameters and setting data in the NC data memory. This memory is buffered by means of a back-up battery, i.e. the data remains in the control after this is switched off. This memory area is accessed directly when processing a part program.

In the directories **LOCAL** and **GLOBAL**, however, the data are assigned to a workpiece and are stored on the hard disk. The program parameters and setting data assigned to a workpiece are not loaded from the hard disk to the NC data memory until the workpiece is loaded. The data can be saved to the hard disk again by means of the **SAVE NC** function.



## 4.1 Editing data in the PARAMETER area

You can edit the following data in the PARAMETER area:

- Program parameters (tool offsets, R parameters etc.)
- Setting data

The editing procedure is identical for all the data types in this area. The following sections describe the operating procedure for editing data via the NC keyboard.

*Note:*

With SW 4 and higher of SINUMERIK 840C, the function “Flexible memory configuration” is provided for altering the memory allocation.

You can configure the size of the following memory areas with this function:

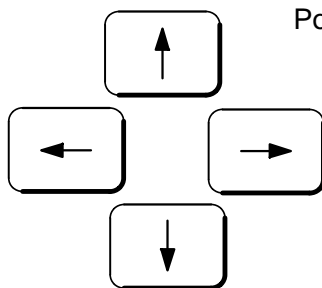
1. User data
  - Part programs
  - IKA data
  - UMS data
  - R parameters
  - TO data
2. Drive software
3. Number of axis-specific measured values
4. Number of block buffers in block memory

Please consult the machine-tool manufacturer's documentation.

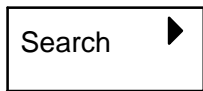
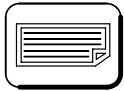
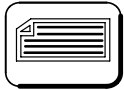
### 4.1.1 Selecting data

When you have selected the desired data range (tool offsets) via the softkey (TO geo + base), an overview of the corresponding data in the form of a table appears in the working field.

*Operating sequence*



Position in any input field with the cursor keys.



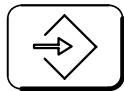
With these keys you page the screen display “down” and “up”.

If you are searching for a specific item of data, enter the desired number via the alphanumeric keyboard and then operate the SEARCH key (do **not** complete your entry by pressing the INPUT key). If the value you are looking for is within the selected data range, the cursor is positioned on this value.

### 4.1.2 Entering and correcting data



If the cursor has been positioned on the desired item of data, enter the desired value via the alphanumeric keyboard.



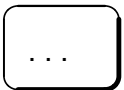
If the input value is correct, you conclude the input with this key. The value is transferred to the memory.



In the input line you can delete characterwise from the right to the left.

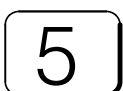


You can activate the cursor in the input window and move it with the cursor keys to insert and delete characters. If you press this key again, you deactivate the cursor.

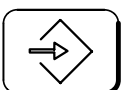
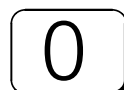


With the EDIT WORD key you can modify the current value by addition or subtraction (does not apply to setting data and R parameters).

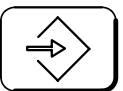
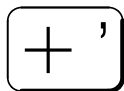
#### Examples



Current value: 50  
New value: 45



If you wish to delete the value:  
Overwrite the value with zero.



With the PLUS and INPUT keys you fix the cursor in the input line (e.g. with tool offsets). Press the keys before you enter the value. The cursor remains positioned on the selected input field even after you press INPUT.

Fixing of the cursor is deactivated with the next cursor movement.

### 4.1.3 Entering PLC data in ASCII format

This function is **not active as a standard**; it must be configured by the machine tool manufacturer. Please read the machine tool manufacturer's operator's guide.

If this function is configured correspondingly, you have the opportunity to input texts – as for example tool designations – in plain text in the control and edit them.

The following rules for operation apply:

- You cannot input more than 27 characters per line.
- With the INPUT key the current text is overwritten left-aligned with the contents of the input line.
- The text line is deleted with the CANCEL key.

Example of a configured display (not available as a standard):

Machine	Parameter	Programm.		Services	Diagnosis	
						14:47
<b>Automatic</b>		Program reset				Mode grp. :1 Chan. :1
<p>Test display for input in PLC texts</p> <p>Text 1 DB59, from DW 0:           TURNING TOOL1</p> <p>Text 2 DB59, from DW 30:        END MILL</p> <p>Text 3 DB59, from DW 60:        TAP</p> <p>Text 4 DB59, from DW 90:        </p>						
						MILL2
▲						

## 4.2 Program parameters

Program  
parameters

With the PROGRAM PARAMETERS softkey you select the area.

On selecting PROGRAM PARAMETERS, the new softkey bar appears.

▲ TO geo + base	TO wear	Zero offset	Angle of rotation	R param. central	R param. chan. spec	Search ►
--------------------	---------	----------------	----------------------	---------------------	------------------------	----------

Softkey bar: Program parameters

The following data is stored in the PARAMETER/PROGRAM PARAMETERS area:

- Tool offsets
- Zero offsets
- Angles of rotation
- R parameters (channel-specific and central)
- ASCII R parameters
- Plane.

### 4.2.1 Tool offsets

Program  
parameters

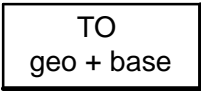
You press the PROGRAM PARAMETERS softkey. The tool offsets (geo + base, i.e. P0 to P4 and P8 and P9) are displayed on the screen:

Machine	Parameter	Programm.		Services	Diagnosis		
						13:29	
AUTOMATIC Program reset						Mode Gp 1 Channel 1	
Tool offsets		Current offset no.		D=	T0 area		1
P No	0	1	2	3	4	8	9
D No	Tool number	T typ	Length 1 Geometry	Length 2 Geometry	Diameter/radius	Length 1 Base	Length 2 Base
1	123	10	100	0	10	0	0
2	223	10	100	0	6	0	0
3	15	20	60	0	24.3	0	0
4	4567	20	150	0	180	0	0
5	0	0	0	0	0	0	0
6	0	0	0	0	0	0	0
7	0	0	0	0	0	0	0
8	0	0	0	0	0	0	0
9	0	0	0	0	0	0	0
10	0	0	0	0	0	0	0
11	0	0	0	0	0	0	0
12	0	0	0	0	0	0	0
13	0	0	0	0	0	0	0
14	0	0	0	0	0	0	0
15	0	0	0	0	0	0	0
16	0	0	0	0	0	0	0
17	0	0	0	0	0	0	0
18	0	0	0	0	0	0	0
▲T0 geo + base		T0 wear		Zero offset		Angle of rotation	
				R param. central		R param. chan.spec.	
						Search	

Fig. 4.3 Tool offsets



To display the other parameters, P5 to P7, you press this softkey.



In order to display the Geo + Base values again (P0 to P4 and P8 and P9) after making a selection, e.g. "TO wear", you press the "TO GEO + BASE" softkey.



The meaning of the parameters is described on the following pages. Editing data is described in the section "Editing data in the PARAMETER area".

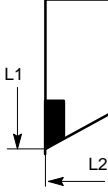
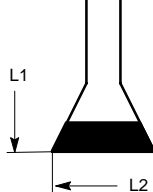
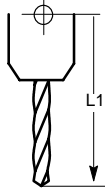
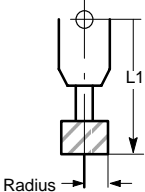
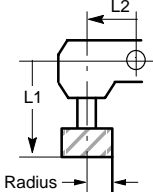
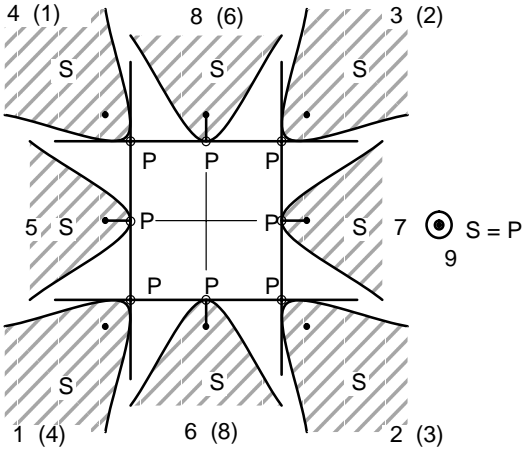
*D No.*  
*P No. (Ident. No.)*  
*TO parameter 0*

Number for tool offset memory  
Tool offset parameters P0 to P9  
Tool number.

This input field, marked by the identifier "0", is provided for entering a "Tool number", which may be up to 8 characters long. Leading zeros are ignored.

Normally, you do not need to perform an input; however, input may be necessary with flexible tool management.

	Please consult the machine tool manufacturer's documentation.	
--	---	--

Tool type	1. Lathe tool Example Turning tool Facing tool	2. Lathe tool Example Grooving tool (left-hand edge)	3. Tools with length compensation only  Example End mill	4. Tools with radius compensation and one length com- pensation  Example: End mill	5. Tools with radius compensation and two length com- pensations  Example: Angle head cutter
					
Type	1...9 (Machining in front of/behind the turning centre)		10	20	30
Lathe tool type: Machining behind turning centre  Figures in brackets: Machining in front of turning centre   S: Position of tool nose centre P: Reference point					

**TO parameters**

2 to 4

L1 geometry, L2 geometry, diameter/radius. See table for TO parameter 1.

**TO parameters**

5 to 7



L1 wear, L2 wear, diameter/radius. For the TO parameters “5” to “7”, it is possible (but not absolutely necessary) to enter the wear data of the tool in the input screenform.









**TO parameters**

8 and 9

L1 base, L2 base.

The TP parameters “8” and “9” are provided for special applications. The “Base dimension” makes a further tool length compensation possible.

	It is possible to disable input of the geometry and wear data of the tool by means of a keyswitch. Please refer to the machine tool manufacturer's documentation.	
---	---	---

	The default maximum wear value in the tool offset parameters P5 – P7 is 9.999 mm. This limit can be changed by the machine manufacturer.	
	The standard classification of TO parameters (P0 to P9) can be increased to as many as 32 by way of machine data. Please note the machine tool manufacturer's documentation.	
	It is possible to define that tool type 0 and tool type 20 are to have the same effect in the machine data. The radius and tool length compensation are then assigned in the same way as for type 20. The display for type 0 remains unchanged.	
	If the control has been equipped with flexible tool management, the D numbers are called by the PLC program of the machine manufacturer. Otherwise, the D numbers have to be specified in the part program as absolute values (see machine tool manufacturer's documentation).	

## 4.2.2 Zero offset

Zero  
offset

With this softkey in the PROGRAM PARAMETERS area, you select the input menu of the zero offsets. Editing is described in the "Editing data in the parameter area" section.

### Notes

The actual value memory and thus also the actual value display relate to machine zero "M" after having reached the reference point.

The workpiece machining program refers to workpiece zero "W". Machine zero "M" and workpiece zero "W" are not identical. Depending on the type of workpiece and the way in which it is chucked, the dimension between machine zero "M" and workpiece zero "W" can vary. This zero offset is taken into account when the program is executed.



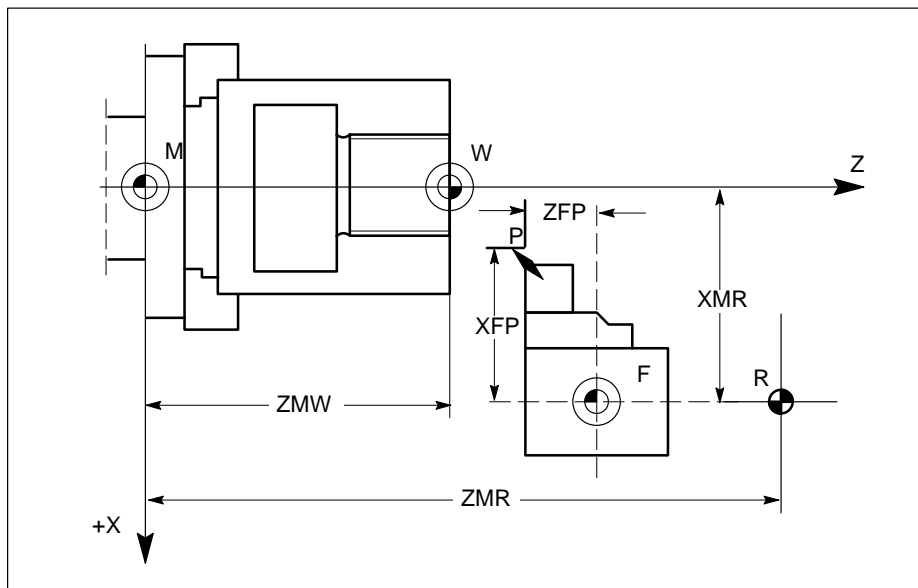


Fig. 4.4 Example: Lathe

You can use “G54” to “G57” to select 4 settable offsets for each axis.

➔	<p>Please note: On lathes, it is usual for the machine zero and the workpiece zero to coincide on the Z axis. In this case, setting a zero offset is meaningful only for the Z axis.</p>	➔
---	--	---

There are two settings in each case for “G54” to “G57”:

- A coarse offset and
- A fine offset (correction of the zero).

You input the values for the settable zero offset into the control as “Program parameters”. The zero offsets which are entered are activated in the part program called.

The values for the G58 and G59 offsets are stored in the program and can be modified only in the program.

Details on programming these zero offsets can be found in the Programming Guide.

The values for “External zero offset” come from the PLC and are displayed on the CRT.



With software version 5 and higher it is also possible to activate zero offsets with G58 Lx Ky or G59 Lx Ky that are programmed in a subroutine Lx under Ny.  
The number of zero offsets stored in this way is only limited by the size of the part program memory.



#### Example

Program MPF 100:	NPV in SPF 123:
N10	N10
:	:
N110 G58 L123 K40	N40 X20 Y5 Z70

#### Effect

As for G58 X20 Y5 Z70.

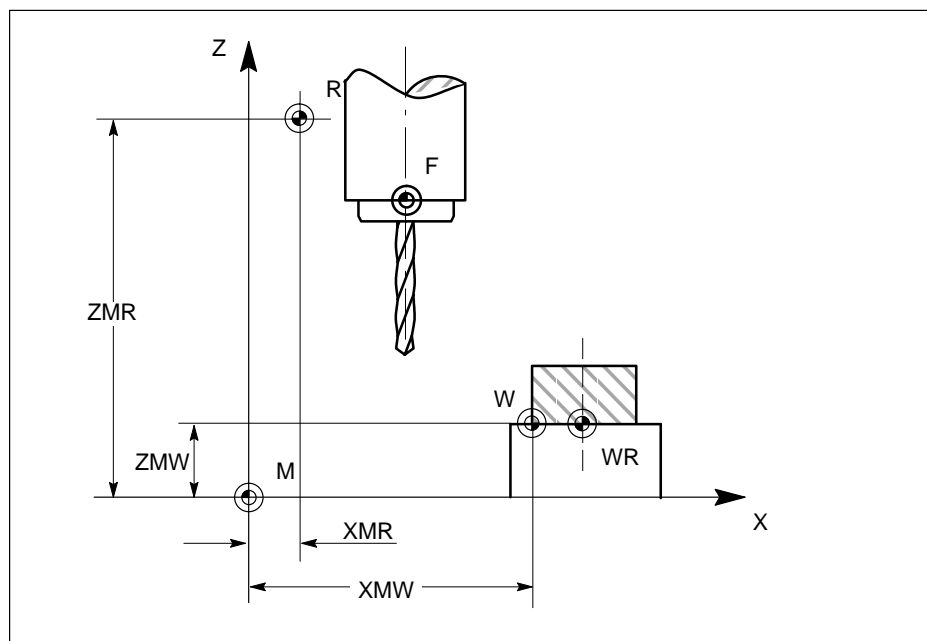


Fig. 4.5 Example: Drilling and milling machine

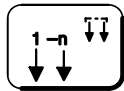
### 4.2.3 Angle of rotation (coordinate rotation)

Angle of  
rotation

With this softkey in the PROGRAM PARAMETERS area, you select the input menu for coordinate rotation. Notes on editing can be found in the section "Editing data in the parameter area". Enter the desired angles of rotation G54 to G57 (angles of rotation G58 and G59 are preset by programming).

## 4.2.4 R parameters

In a program, R parameters are used to represent a numerical value of an address. Within the program, you assign values to them and it is thus possible to modify a program for a variety of similar applications (e.g. different feedrates, spindle speeds for various materials, fixed cycles). Parameters are made up of the address R and a number comprising up to four digits (with the exception of ASCII-R parameters).



Select the desired channel for channel-specific parameters by pressing the CHANNEL SWITCHOVER key.

R param.  
central

Press the R PARAMETER CENTRAL key.  
The parameters R700 ... R1299 are displayed.

### Note

The standard number of 600 R parameters can be altered, see also the configuring notes further down!

R param.  
chan. spec.

You press the R PARAMETER CHANNEL SPECIFIC softkey and the channel-specific R parameters are called up (R0 ... R699).  
The input range is -99999999 to +99999999.



With the ETC key you can extend the menu bar and

Chan. spec.  
ASCII R P

the CHANNEL SPECIFIC ASCII R PARAMETERS softkey is offered.

You can enter any ASCII character in these R parameters. You enter the program name for the "Select part program for editing" function in plain text (e.g. MPF1). This function is not activated as a standard; it must be configured by the machine manufacturer. Please read the machine tool manufacturer's documentation.

Editing is described in the section "Editing data in the parameter area".



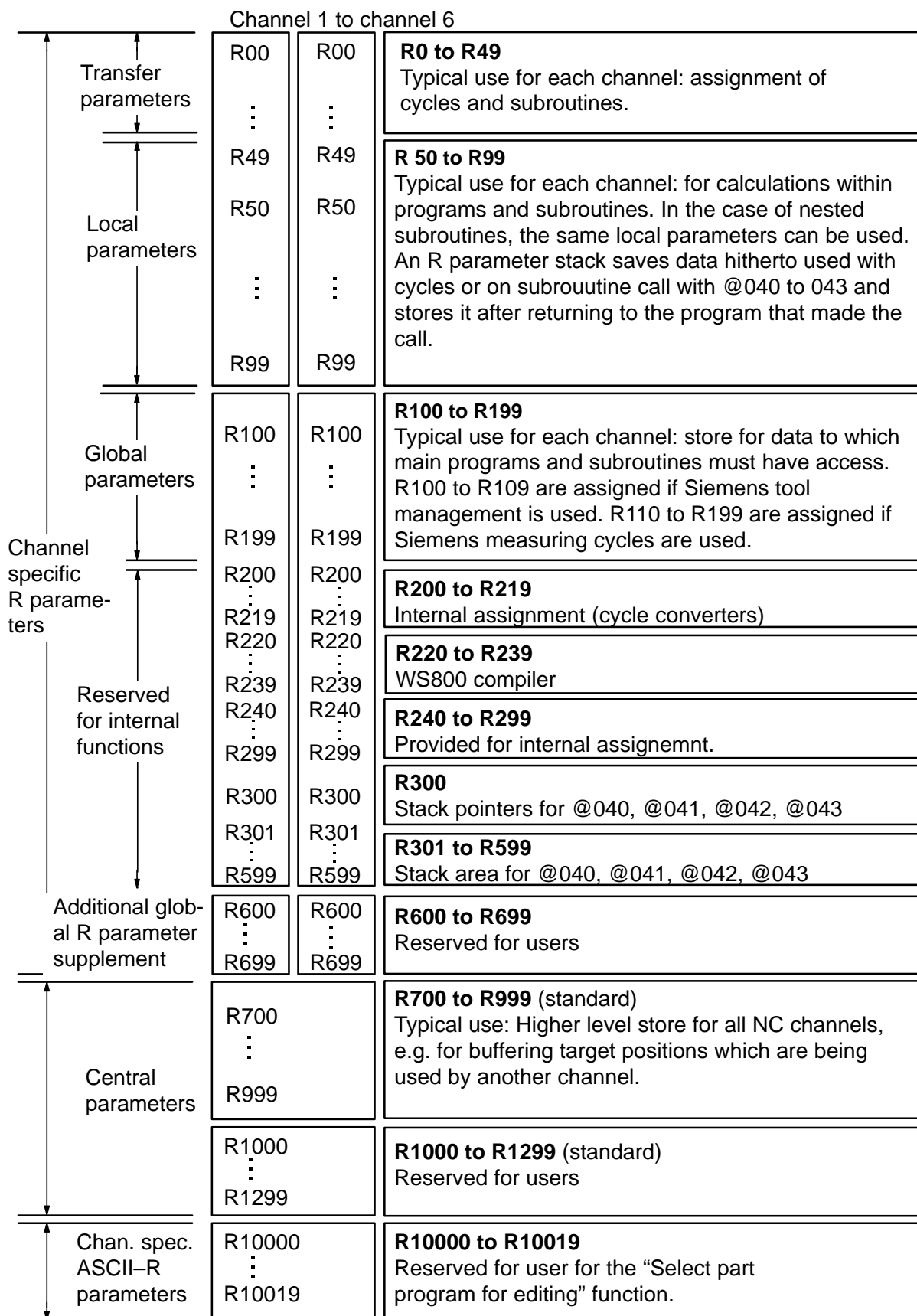
Please note:  
Only certain R parameters are freely available to the user. The parameter assignments are shown in the following overview.



The SINUMERIK 840C has a variable memory configuration. R parameter areas can be reconfigured (SRAM). Please consult the machine-tool manufacturer's documentation.



## Structure and use of R parameters



## 4.2.5 Plane

Plane

With this softkey in the PROGRAM PARAMETERS area, you select the display menu for plane selection. The planes are preset by the machine manufacturer in the machine data and are simply displayed here.

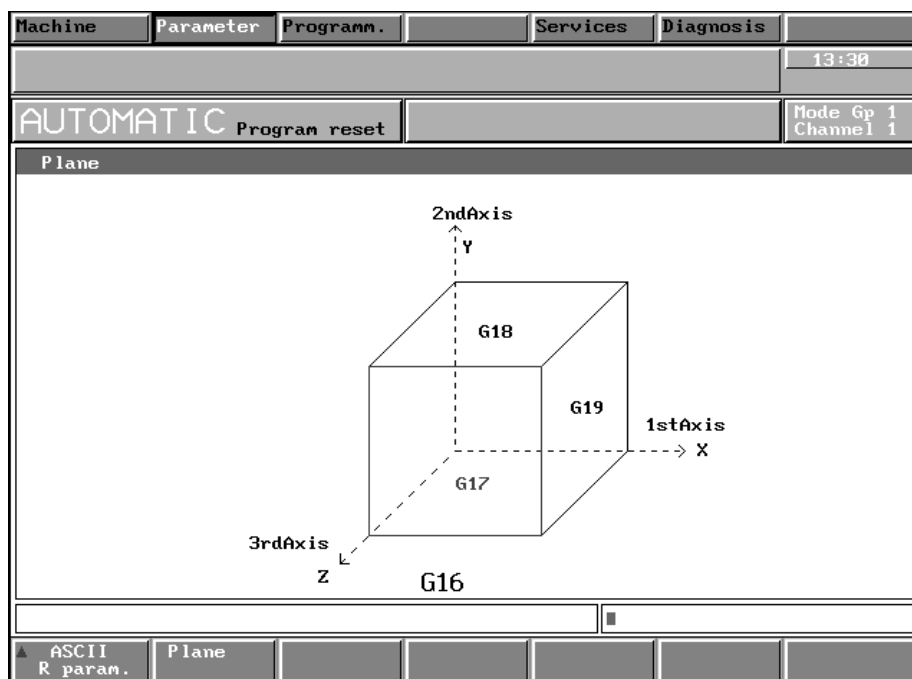


Fig. 4.6 Menu for plane selection in the parameter area

## 4.2.6 Setting data

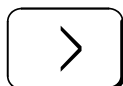
Setting data

With this softkey in the basic display of the PARAMETER area, you select the input menus for the setting data.

Editing is described in the section “Editing data in the PARAMETER area”.

The following menu appears in the softkey bar:

Working area limit.	General data	Spindle data	Scale	Setting bits	Protection zone	Position meas. sig.	up to SW 6.2
---------------------	--------------	--------------	-------	--------------	-----------------	---------------------	--------------



Press the ETC key to expand this softkey bar. The following softkeys are then displayed.

Axis and spind. conv.	Cycle data	Gearbox interpol.	Travel to fixed stop	Inch/metr. switchover	Search	up to SW 6.2
-----------------------	------------	-------------------	----------------------	-----------------------	--------	--------------

### 4.2.6.1 Working area limitation

Working  
area limit.

With this softkey, you select the input menu for the working area limitation.

You can enter the working area limitation setting data in the display.

Using the working area limitation in all operating modes, the traversing ranges can be limited in addition to the software limit switches. The axes are displayed for a specific mode group.

When switching over from the METRIC to the INCH input system (and vice versa), the values of the working area limitation are adjusted in the respective measuring system only if the maximum possible value is exceeded. If the value of the working area limitation is possible in the current measuring system, no adjustment is made.

### 4.2.6.2 General setting data

General  
data

With this softkey, you select the input menu for the general setting data. You can enter the following setting data for the GENERAL DATA display:

- Dry run feedrate
- INC VAR (traversing in incremental mode with variable increment)
- Smoothing constant for thread

*Dry run feedrate  
(SD0)*

The dry run feedrate (unit/min x 1000) entered here is effective if the dry run feedrate has been activated through program modification. All blocks for which a feedrate has been programmed are now traversed with the "Dry run feedrate" specified via the setting data instead of the programmed feedrate (but please observe the maximum feedrates defined by machine data).

*Increment for INC  
INC variable (SD9)*

Here you can store the value for the variable increment. This value also appears in the input field in JOG mode and can be edited there. The value is self-retaining.

*Smoothing constant  
for thread (SD1)*

This function reduces the wear on the drives for thread cutting and enables a constant speed. The feed ramp-up time until synchronization is reached with the running working spindle can be programmed with G92T ... or by entering the T value in setting data 1.

### 4.2.6.3 Spindle setting data

Spindle  
data

With this softkey, you select the input menu for the spindle data. You can enter the following data:

*Spindle speed  
limitation (static)  
(SD 403\*)*

The spindle speed limitation is used to limit the spindle to the speed entered. It can be modified in the program by means of G26 S... .

*Programmed  
spindle speed  
limitation (G92)  
(SD401\*)*

The programmed spindle speed limitation can be used in addition to the fixed spindle speed limitation to reduce the spindle speed in the program using the G92 function. The programmed spindle speed limitation is only active at a constant cutting rate (G96). The value is entered in the setting data automatically.

*Oriented  
spindle stop (M19)  
(SD 402\*)*

When M19 is programmed, positioning takes place at this angle (in degrees).

Example:

- With M19 S270 LF, the spindle is positioned to 270° and the angle entered in the setting data.
- With M19 LF, the spindle is positioned to the angle entered in the setting data.

*Start angle G92A  
(SD 204\*)*

When machining a multi-turn thread ("Extended thread package" option), a separate start angle must be entered for each thread turn. This can be defined in the setting data.

### 4.2.6.4 Scale

Scale

With this softkey you select the display menu for the scale data.

The values for scale factor and scale centre which are preset by programming are displayed on the CRT.

*Enabling bit*

If the "Scale" function is to be activated, the axes have to be "enabled". This serves on the one hand as a protective function and on the other as a means of manipulation. (For example, you can enable only one axis and thus convert a square into a rectangle).

The axis-specific enabling bits are included with the setting data bits (address 560\*). For this, bit 2 serves to enable the NC axes.

### 4.2.6.5 General setting data bits

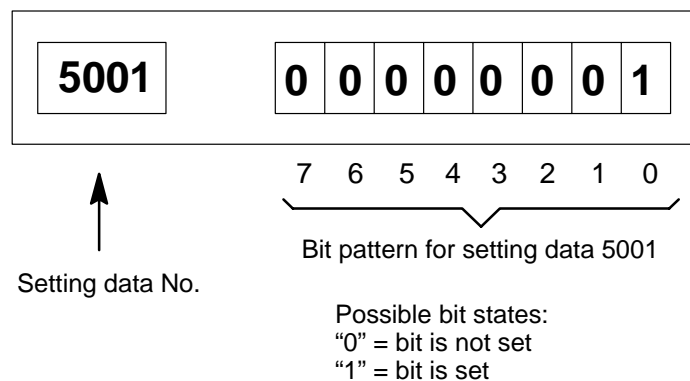
General  
bits

You select GENERAL SETTING DATA BITS.

The display with the setting data bits of setting data 5000 to 5599 appears on the CRT.

The setting data from byte No. 5000 (up to 5799) are made up of 8 bits each: 0 to 7.

*Explanation of the setting data structure:  
(e.g. for setting data (SD) 5001)*



You can set the desired bit pattern for any byte in the area from 5000 to 5799 on the CRT by way of an input screenform.

The following bits can be set:

SD 5000 bit 0,1,2 = 0 : Standard cycle up to UMS 02  
= 1 : Standard cycles as from UMS 03

SD 5000 bit 7 = 0 : No calculation of the overflow compensation  
= 1 : Calculation of the overflow compensation (see cycles description .L84)

SD 5001 bit 0 = 1 : Display of workpiece-related actual value system

SD 5001 Bit 2 = 1 : Automatic saving (SAV) active.

### 4.2.6.6 Axial setting data bits

Axial bits

You select the AXIAL SETTING DATA BITS.

The display with the setting data bits of the setting data 5600 to 5799 appears on the screen.

The following bits can be set:

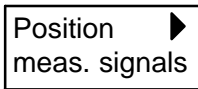
SD 560\* bit 0 = 1 : Override switch takes no effect on the axis feed.

bit 1 = 1 : Override switch takes no effect on the rapid traverse of the axis.

bit 2 = 0 : Scale factor G51 is effective during machining.



### 4.2.6.7 Position measuring signals

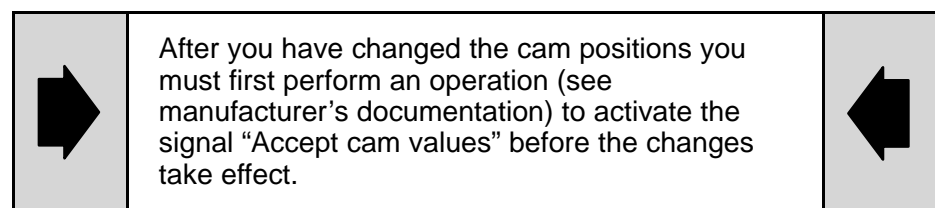
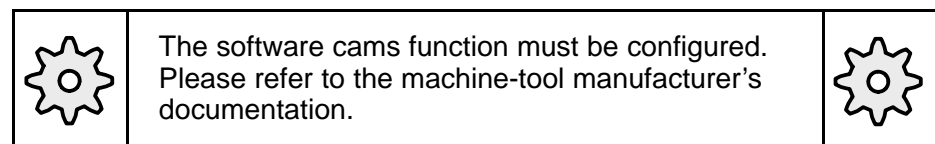
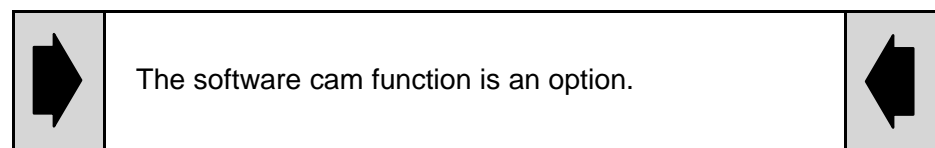


You select the softkey and the following display appears:

Machine	Parameter	Programm.	Services	Diagnosis																	
					15:01																
AUTOMATIC		Program reset		Mode Gp 1 Channel 1																	
Setting data values: Pos.meas.signals																					
<table border="1"> <tr> <td>Minus cam 1=</td> <td>-600</td> <td>Plus cam 1=</td> <td>250</td> </tr> <tr> <td>Minus cam 2=</td> <td>-100</td> <td>Plus cam 2=</td> <td>200</td> </tr> <tr> <td>Minus cam 3=</td> <td>-250</td> <td>Plus cam 3=</td> <td>250</td> </tr> <tr> <td>Minus cam 4=</td> <td>-50</td> <td>Plus cam 4=</td> <td>50</td> </tr> </table>						Minus cam 1=	-600	Plus cam 1=	250	Minus cam 2=	-100	Plus cam 2=	200	Minus cam 3=	-250	Plus cam 3=	250	Minus cam 4=	-50	Plus cam 4=	50
Minus cam 1=	-600	Plus cam 1=	250																		
Minus cam 2=	-100	Plus cam 2=	200																		
Minus cam 3=	-250	Plus cam 3=	250																		
Minus cam 4=	-50	Plus cam 4=	50																		

Fig. 4.7 Position measuring signals

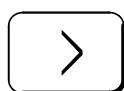
You can enter the values for 4 software cam pairs in the input fields. The SOFTWARE CAMS function allows you to generate position measuring signals in addition to the existing hardware cams.  
Input range:  $\pm 99999999$



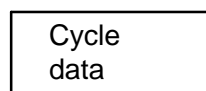
### Notes

- This function can only be used for linear axes.
- The software cam signals can only be activated after referencing the NC axes.
- The cam positions refer to the machine zero offset and are used in the active machine measurement system. The cam positions are not verified in the input display with regard to the maximum traversing ranges.
- The cams are combined in pairs. A pair of cams consists of a positive cam and a negative cam. The area **above** the positive cam position is assigned to the positive cam area and the area on the axis **below** the negative cam position is assigned to the negative cam area.
- For a detailed description, refer to the following documentation:  
SINUMERIK 840C  
SIMODRIVE 611-D  
Start-Up Guide  
Service Documentation  
Section 12.22

### 4.2.6.8 Cycle setting data



Press the ETC key to extend the SETTING DATA softkey bar.



You select CYCLE SETTING DATA.

The 1st display for cycle setting data (system data) appears on the screen.

Cycle setting data are separate for each channel. They are grouped into 4 system and user areas:

#### Cycle setting data

0 . 99	System data	Reserved
400 . 499	User data	Free
800 . 849	System bits	Reserved
900 . 949	User bits	Free

Use of the reserved system data is described in the corresponding cycle descriptions.

#### 4.2.6.9 Axis and spindle converter

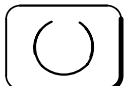
Axis/spind.  
converter

When you press this softkey, the input screenform for the Axis/Spindle Converter function is displayed.





Machine	Parameter	Programm.	Services	Diagnosis																						
				15:10																						
AUTOMATIC		Program reset	ACR	ROV																						
			Mode Gp 1 Channel 2																							
Converter																										
<div style="display: flex; justify-content: space-between;"> <div style="width: 48%;"> <p style="text-align: center;">Axes <span style="border: 1px solid black; padding: 2px 5px;">yes</span></p> <p>Programmed -&gt; machine</p> <table style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 50%; text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;">X</span></td> <td style="width: 50%; text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;">X1</span></td> </tr> <tr> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;">Y</span></td> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;">Y1</span></td> </tr> <tr> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;">Z</span></td> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;">Z1</span></td> </tr> <tr> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;"></span></td> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;"></span></td> </tr> <tr> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;"></span></td> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;"></span></td> </tr> <tr> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;"></span></td> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;"></span></td> </tr> <tr> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;"></span></td> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;"></span></td> </tr> <tr> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;"></span></td> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;"></span></td> </tr> </table> </div> <div style="width: 48%;"> <p style="text-align: center;">Spindles <span style="border: 1px solid black; padding: 2px 5px;">no</span></p> <p>Programmed -&gt; machine</p> <table style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 50%; text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;">S</span></td> <td style="width: 50%; text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;">S</span></td> </tr> <tr> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;">S</span></td> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;">S</span></td> </tr> <tr> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;">S</span></td> <td style="text-align: center;"><span style="border: 1px solid black; padding: 2px 5px;">S</span></td> </tr> </table> </div> </div>					<span style="border: 1px solid black; padding: 2px 5px;">X</span>	<span style="border: 1px solid black; padding: 2px 5px;">X1</span>	<span style="border: 1px solid black; padding: 2px 5px;">Y</span>	<span style="border: 1px solid black; padding: 2px 5px;">Y1</span>	<span style="border: 1px solid black; padding: 2px 5px;">Z</span>	<span style="border: 1px solid black; padding: 2px 5px;">Z1</span>	<span style="border: 1px solid black; padding: 2px 5px;"></span>	<span style="border: 1px solid black; padding: 2px 5px;"></span>	<span style="border: 1px solid black; padding: 2px 5px;"></span>	<span style="border: 1px solid black; padding: 2px 5px;"></span>	<span style="border: 1px solid black; padding: 2px 5px;"></span>	<span style="border: 1px solid black; padding: 2px 5px;"></span>	<span style="border: 1px solid black; padding: 2px 5px;"></span>	<span style="border: 1px solid black; padding: 2px 5px;"></span>	<span style="border: 1px solid black; padding: 2px 5px;"></span>	<span style="border: 1px solid black; padding: 2px 5px;"></span>	<span style="border: 1px solid black; padding: 2px 5px;">S</span>	<span style="border: 1px solid black; padding: 2px 5px;">S</span>	<span style="border: 1px solid black; padding: 2px 5px;">S</span>	<span style="border: 1px solid black; padding: 2px 5px;">S</span>	<span style="border: 1px solid black; padding: 2px 5px;">S</span>	<span style="border: 1px solid black; padding: 2px 5px;">S</span>
<span style="border: 1px solid black; padding: 2px 5px;">X</span>	<span style="border: 1px solid black; padding: 2px 5px;">X1</span>																									
<span style="border: 1px solid black; padding: 2px 5px;">Y</span>	<span style="border: 1px solid black; padding: 2px 5px;">Y1</span>																									
<span style="border: 1px solid black; padding: 2px 5px;">Z</span>	<span style="border: 1px solid black; padding: 2px 5px;">Z1</span>																									
<span style="border: 1px solid black; padding: 2px 5px;"></span>	<span style="border: 1px solid black; padding: 2px 5px;"></span>																									
<span style="border: 1px solid black; padding: 2px 5px;"></span>	<span style="border: 1px solid black; padding: 2px 5px;"></span>																									
<span style="border: 1px solid black; padding: 2px 5px;"></span>	<span style="border: 1px solid black; padding: 2px 5px;"></span>																									
<span style="border: 1px solid black; padding: 2px 5px;"></span>	<span style="border: 1px solid black; padding: 2px 5px;"></span>																									
<span style="border: 1px solid black; padding: 2px 5px;"></span>	<span style="border: 1px solid black; padding: 2px 5px;"></span>																									
<span style="border: 1px solid black; padding: 2px 5px;">S</span>	<span style="border: 1px solid black; padding: 2px 5px;">S</span>																									
<span style="border: 1px solid black; padding: 2px 5px;">S</span>	<span style="border: 1px solid black; padding: 2px 5px;">S</span>																									
<span style="border: 1px solid black; padding: 2px 5px;">S</span>	<span style="border: 1px solid black; padding: 2px 5px;">S</span>																									
Axis/spind converter																										

*Fig. 4.8 Axis/spindle converter*

With this function you can convert axis and spindle addresses into other axis and spindle addresses.



Move the cursor to the toggle field “Axes no”. If you can switch the contents to “yes”, the function can be used in this channel.

	<p>This function must be enabled and activated by the machine manufacturer for the channel in question (technology MD). The function must also be enabled in the channel-specific SD 540*.0.</p>	
	<p>The machine manufacturer can configure an input disable when the keyswitch is in the “0” position.</p>	

The following characters can be used for the axis names: X, Y, Z, A, B, C, U, V, W, Q, E as well as possible axis extensions X1 ... X15 etc.

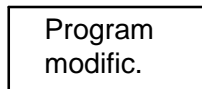
Up to 8 axis converter pairs per channel can be input. You cannot enter values directly for the spindle conversion. The values can only be entered via the PLC program.



In the input line you can delete character by character from the right to the left.

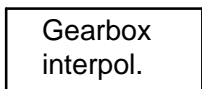


The contents of the selected input field are deleted.



In SW 5 and earlier, the axis converter function is activated in the Program modific. screen. From SW 6, a toggle field in the Axis/Spindle Converter screen can also be used to activate this function.

#### 4.2.6.10 Gearbox interpolation





You can use this softkey to select the input screen for gearbox interpolation from the parameter/setting data menu. It is possible to program the gearbox interpolation groups completely via the input field. The input field is also used to display the current status and configuration of the groups.

You select a gearbox interpolation group by entering the name of the following drive in the "Following axis" field. The input screen is then in DISPLAY mode and shows the current GI status of this following drive. If changes are made in any of the other fields, the control automatically switches to INPUT mode. The values which have just been entered only become active when the corresponding softkey is pressed. When the function has been performed, the system switches back to DISPLAY mode. This is a sign that the values have been accepted. The operator actions are identical to those of programming with G commands, i.e. each softkey represents a GI command. The G commands are described along with the softkeys.

Machine	Parameter	Programm.	Services	Diagnosis			
					15:15		
JOG		Program reset			Mode Gp 1 Channel 1		
Gearbox interpolation Input							
Name	Type	GI type	IKA	IKP	Coupl. factor Numer. Denom.	Mach. pos./ travel dir.	Status
L 1	X Axis	Linear	K1		3 1		LINK_ON
L 2	Y Axis	Linear	K1		2 1		LINK_ON
L 3	Z Axis	Linear	K1		1 1		LINK_ON
L 4							
L 5							
F	A Axis						
Direct input							
Delete configur.	Define configur.	LINK_OFF total	LINK_ON total	In-process synchron.	LINK_ON LINK_CH	Start pos. LINK_ON/CH	

Fig. 4.9 Gearbox interpolation

	<p>The function gearbox interpolation must be configured by the machine-tool manufacturer (machine data). Please consult the machine-tool manufacturer's documentation!</p>	
---	---	---

### Description of the fields

**Axis name** Input field for the axis/spindle names. When you enter the names, the system checks that the axes/spindles are also defined in the mode group. The system also checks whether following axes/spindles are real axes/spindles.

**Axis type** You cannot enter the axis type. The axis type (axis or spindle) is entered by the system according to the axis/spindle name.

**GI type** The toggle field GI type offers the following options:

- Linear (K01 ... K04)
- Curve (K11 and K12)

The link structure can be selected by toggling between the options in the right-hand field. The following options are provided:

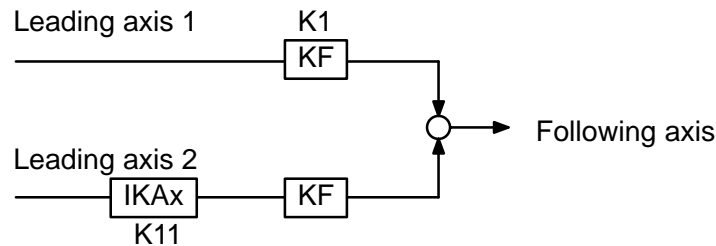
- K1: Setpoint position link
- K2: Actual position link
- K3: Setpoint position link with simulated actual values of the main axis/spindle, i.e. no compensation control possible.
- K4: Setpoint speed actual position link

A compensation controller can be activated for K1 and K2 from the PLC. Press the DEFINE CONFIG. softkey twice to activate the configuration.

The link structure for the GI type can be selected by toggling between the options in the right-hand field. The following options are available:

- K11: Setpoint position link
- K12: Actual position link

IKA relations in the gearbox interpolation can thus be calculated.



#### Link factor

Enter the link factor for the main axis by specifying the denominator and the numerator (8 decimal places + decimal point + leading sign). Activate the entry by pressing the LINK\_ON/LINK\_SWITCH softkey. The entered link factor is only accepted if a gearbox interpolation group has already been configured.

#### Machine position

Input field for the synchronization position for flying synchronization (8 decimal places + decimal point + leading sign). An absolute value must be entered. The reference system is always the machine coordinate system. Activate the entry by pressing the FLYING SYNCHRON. softkey twice. The machine position is only accepted if a gearbox interpolation group has been configured. After the link has been activated by pressing the softkey, the value disappears from the screen.

#### Direction of travel

If + or – is entered, the machine position is only accepted if the position entered has been crossed in the positive or negative direction. If no sign is entered, the direction is irrelevant.

#### IKA

Enter here the number of the IKA block used.

#### IKP

Enter here the control curve number of the IKA.

The entries for IKA and IKP are only relevant if a GI type curve has been selected.

#### Status

The current status of the gearbox link is entered here by the system. A toggle key allows you to activate or deactivate the link for each main drive. Activate by pressing the LINK\_ON/LINK\_SWITCH softkey.

The input screen for gearbox interpolation displays the following softkey bar:

Delete config.	Define config.	Link_off all	Link_on all	Flying synchron.	Link_on link_switch	Start pos. LINK_ON/SW
-------------------	-------------------	-----------------	----------------	---------------------	------------------------	--------------------------

### Explanation of the softkey bar

Delete  
config.

When you press a softkey, the prompt "Confirm input" is displayed. Press the key again to confirm.

Define  
config.

The gearbox configuration for the selected following axis is cancelled. You must first press LINK\_OFF. This softkey is identical to the G command "G401 FA".

Link\_off  
all

The configuration defined in the basic display by the parameter settings is subjected to an integrity check and the gearbox interpolation group is generated. The default setting for link factors is zero. No position is entered. The link is deactivated (status LINK\_OFF). This softkey is identical to the G command "G401 LA<sub>1</sub> K<sub>1</sub>.....LA<sub>5</sub> K<sub>5</sub> FA".

Link\_on  
all

Press this softkey twice to perform link\_off for the entire gearbox interpolation group selected by the following axis. The current link factors are retained. The positions of the main drives are cleared. The status is set to LINK\_OFF for every main axis. This softkey is identical to the G command "G400 FA".

Flying  
synchron.

Press this softkey twice to perform link\_on for the entire gearbox interpolation group selected by the following axis. The current entered link factors remain active. The positions of the main drives are cleared. The status is set to LINK\_ON for every axis. This softkey is identical to the G command "G402 FA".

Link\_on  
link\_switch

LINK\_ON for the entire gearbox interpolation group selected by the following axis. The entered link factors are active and flying synchronization is performed with the entered synchronization positions. This softkey is identical to the G command "G403 LA<sub>1</sub> LA<sub>1</sub>POS I J....LA<sub>5</sub> LA<sub>5</sub>POS I J FA FAPOS".

Start pos.  
Link\_ON/Sw

After the new link factors have been entered or the link status has been changed (toggle field) press this softkey twice to activate the entries or changes. New positions are not taken into account. This softkey is identical to the G command "G402 LA<sub>1</sub> I J.....LA<sub>5</sub> I J FA".

The entry made in the machine position column is evaluated when the softkey is pressed, i.e., after the position has been reached, link on, link switch are switched off depending on the state before the softkey was pressed.

### Note

On installation, the default setting for the transmission ratio is 0. If a new transmission ratio is entered it is stored in the static RAM, i.e. the GI data are not lost when the control is switched on/off. The GI data can be stored on the hard disk with the function SERVICES/NC/SAVE START (identifier GIA). GI data that are entered directly by hand or generated in the NC program are active immediately.



If a GIA block is loaded from the hard disk into the NCK, the GIA data are not active until POWER ON (safety function).

### Sequence of operations

The sequence of operations for activating a gearbox interpolation grouping is always:

1. Define following axis
2. Define leading axes and link type (for link via IKA tables the IKA data must be loaded)
3. Softkey: Accept configuration (2x)
4. Enter link factors
5. Softkey: LINK\_on / LINK\_switch (2x)
6. Enter starting position, if necessary
7. Softkey: Start pos. Link\_ON/switch if necessary (2x)
8. Softkey: LINK\_on all (2x)

The configuration can only be altered after softkey: LINK\_off All and softkey: Delete config.

	<p>The selected axis must be enabled as a GI following axis in the machine data (MD). The controller parameters of the compensatory controller must have matching values.</p> <p>The actions “New configuration”, “Switch link factor” and “Change start position” can be disabled independently of one another and axis-specifically.</p> <p>Gearbox interpolation is an option. Please consult the machine manufacturer’s documentation where necessary.</p>	
---	--	---



### 4.2.6.11 Travel to fixed stop

#### Travel to fixed stop

Press the softkey and the input screen is displayed with the configured axes.

You can use the function "Travel to fixed stop" to set-up defined forces for clamping workpieces, e.g. tail stocks, sleeves, grippers, etc.

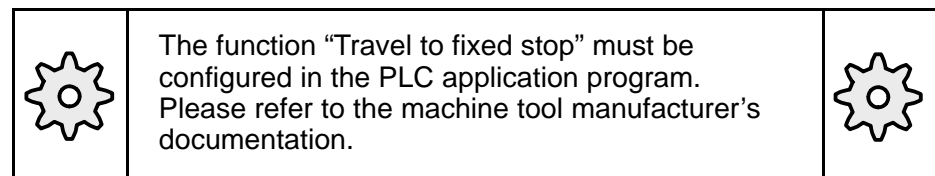
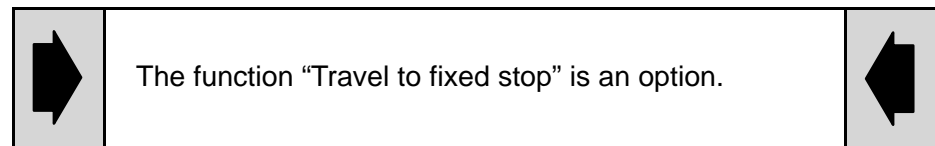
The clamping force for this function is entered in the input fields assigned to the axes.

Input range: 0 to 999999

Tolerance: 0.1% of max. power setpoint (motor torque)

#### Note

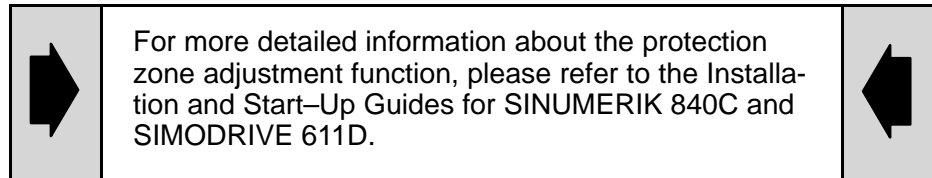
- The function can be used for all axes.
- Spindles which are to travel to fixed stop must first be switched to C axis mode.



## 4.2.7 Additional protection zone adjustment via setting data (from SW 6.3)

**4.2.7 Additional protection zone adjustment via setting data (from SW 6.3)***Selection*

The screens for protection zone adjustment are selected via softkeys **PARAMETER**, **SETTING DATA** and **PROTECTION ZONE**.



The "Protection zones" screen displays protection zone reference point P1 and diagonal point P7 for individual zones. The displayed dimensions are absolute with reference to the monitoring coordinate system.

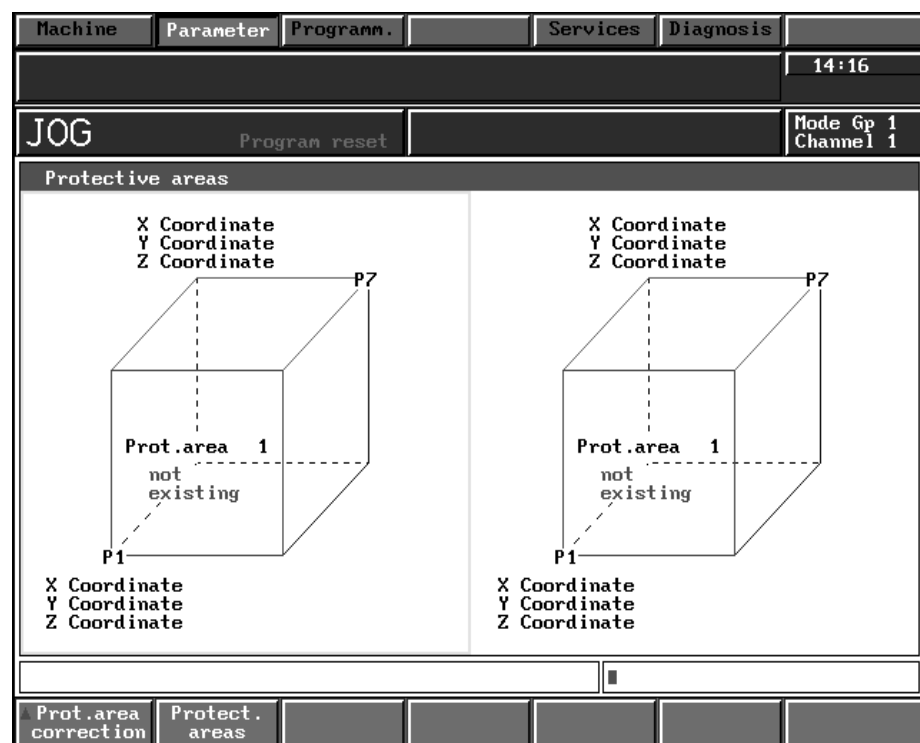


Fig. 4.10 Protection zone correction

A protection zone is uniquely defined by two points in space.

The control displays a diagram containing points P1 and P7 positioned along the spatial diagonal at the corners of a three-dimensional cube. The screen shows the X, Y and Z coordinates of these two points. The names of the motion axes are also displayed on the abscissa, ordinate and applicate.



Each zone coordinate is assigned two setting data, one for each coordinate direction. The setting data value is added to the basic dimension of the zone, thus extending the zone in the appropriate coordinate direction.

The values for the coordinates for additional protection zone adjustment must be entered in setting data 800\*, 804\*, 816\* and 820\*.

Machine	Parameter	Programm.		Services	Diagnosis	
						14:17
JOG		Program reset				Mode Gp 1 Channel 1
Protective area correction						
Prot.area	Coordinate					
	X+	X-	Y+	Y-	Z+	Z-
1	0	0	0	0	0	0
2	0	0	0	0	0	0
3	0	0	0	0	0	0
4	0	0	0	0	0	0
5	0	0	0	0	0	0
6	0	0	0	0	0	0
7	0	0	0	0	0	0
8	0	0	0	0	0	0
9	0	0	0	0	0	0
10	0	0	0	0	0	0
11	0	0	0	0	0	0
12	0	0	0	0	0	0
13	0	0	0	0	0	0
14	0	0	0	0	0	0
15	0	0	0	0	0	0
16	0	0	0	0	0	0
17	0	0	0	0	0	0
18	0	0	0	0	0	0
19	0	0	0	0	0	0
20	0	0	0	0	0	0
21	0	0	0	0	0	0
22	0	0	0	0	0	0
Prot.area correction						Protect. areas

Fig. 4.11 Protection zone correction

## 4.2.8 Inch/metric conversion (from SW 6.3)

	<p>For more detailed information about the Inch/metric conversion function, please refer to the Installation and Start-Up Guide for SINUMERIK 840C and SIMODRIVE 611D, Section 12.36</p>	
---	--	---

END OF SECTION



## 5 Programming

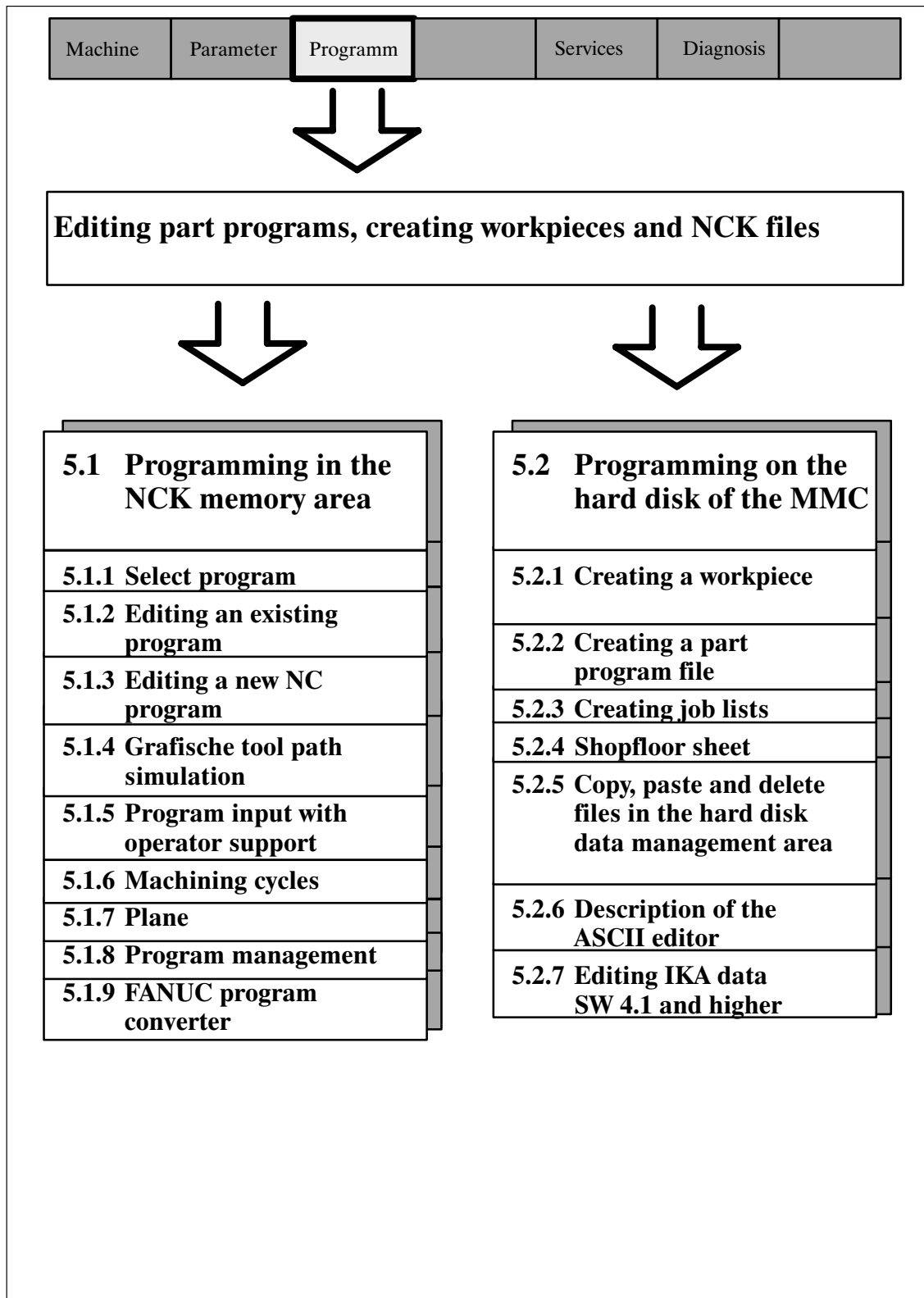


Fig. 5.1 Programming



*HD memory*

The free memory on the hard disk is displayed in KB = kilobytes (1 KB = 1024 Bytes) in the top right-hand corner of all screens in data management under LOCAL and GLOBAL.

## 5.1 Programming in the NCK memory area (directory NCK)

Select area NCK with the data selector. DIN programming is also automatically selected.

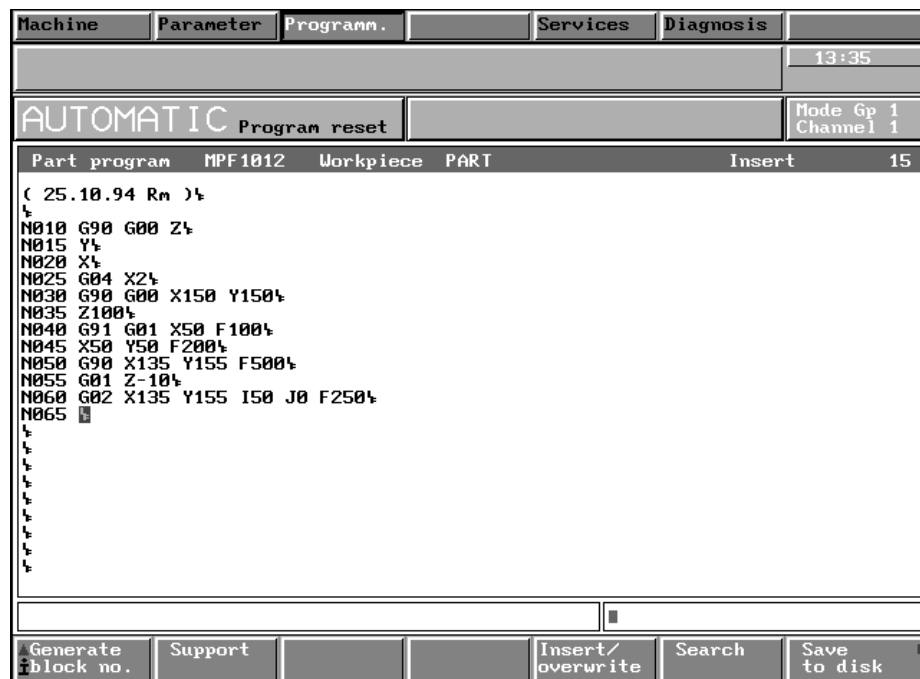
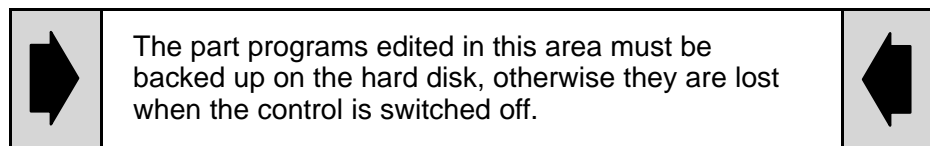


Fig. 5.3 Programming part programs in directory NCK

In this area you can now edit part programs in DIN format directly in the working memory.



If the letter "i" is displayed in the 1st softkey window (left) notes relating the selected area appear after pressing the HELP key.

## 5.1.1 Select program

Select directory NCK with the data selector and preferably select a main program or subroutine. (It is also possible to select an MDA program, e.g. to make a change before storing it as a part program).

Example:

Machine	Parameter	Programm.	Services	Diagnosis
				13:36
AUTOMATIC Program reset				Mode Gp 1 Channel 1
Programming USER/NCK/MPF				
Caution! Programs are deleted on switching off.				
Mark: EXT = external program    c = cycle protect.    x = executable r = readable    w = writable				
Name	Type	Length	Mark	Comment in the 1st block/date
MPF1012	part	250	-rwx	Z-No.: 34K100/12
MPF11	part	299	-rwx	WKS-No.: 3
MPF13	part	564	-rwx	MPF
MPF14	part	442	-rwx	
MPF22	part	159	-rwx	GST
MPF23	part	106	-rwx	Portal
Free part program memory: 686000 characters 1372 sectors				
<div> <div>Edit</div> <div>Modify rights</div> <div>Copy</div> <div>Set block number</div> <div>Rename</div> <div>Delete</div> </div>				

Fig. 5.4 NCK programming

Edit

Place the cursor on the program that you wish to edit and press the softkey EDIT. The part program is called up and can be edited.

If you wish to enter a **new** program, select MPF or SPF and then change to the keyboard and enter the desired program number (as %, L, MPF, SPF).

Edit

Then press the softkey EDIT.

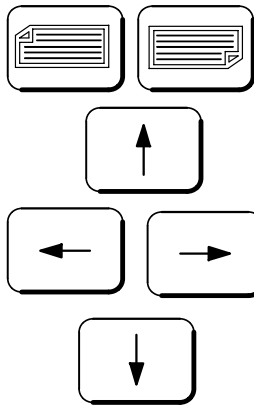
You can now edit the program.



## 5.1.2 Editing an existing program

### Note

If you try to select a protected part program for editing, the dialog text “Editor error: editing not permitted” appears (see the Section “Changing the access rights”).



Using the page up/down and the cursor keys, you can position the cursor at any position of a part program already stored. Pressing the key combination SHIFT + ARROW or SHIFT + PAGE moves the cursor to the start or end of the line or program file.

If you press RIGHT ARROW again at the end of a block, the cursor is positioned at the beginning of the next line. Pressing LEFT ARROW at the beginning of a block moves the cursor to the last character in the previous block.



In the input line you can delete character by character from the right to the left using the DELETE INPUT key.

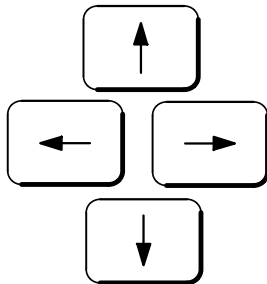


Press DELETE DATA to delete the character directly under the cursor. The result of a text search is deleted completely.

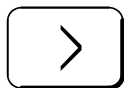
Select the function CUT if you wish to delete a larger program section or several blocks.



Use the selection key to select the first character of the character string that you want to delete. Pressing the key twice cancels the selection.

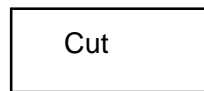


Use the arrow keys to select the desired text. You can press the Page Up or Page Down keys to select larger areas of text.

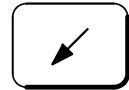
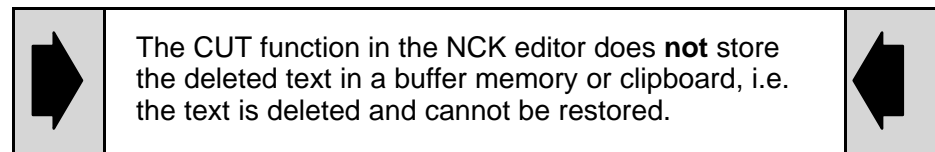


Press the ETC key to extend the softkey bar.

## 5.1.2 Editing an existing program

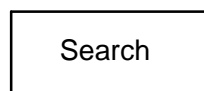


Press the CUT softkey to delete the selected text.



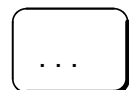
You can activate the cursor in the input line. The cursor is then inactive in the program, i.e. you can only enter characters in the input line (e.g. for the SEARCH function).

Pressing the key again deselects the activation.

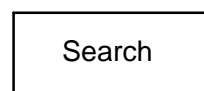


Press the SEARCH softkey to search for a block or a character string in the selected program. The message

“Please enter search text” appears.



Use the alphanumeric keyboard to enter the search term and then press the SEARCH softkey.

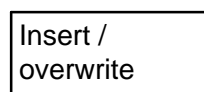


The cursor is positioned before the search term. The search is performed from the cursor position towards the end of the program.

#### Note

The search function is used to find character strings. This means that if you enter “N1” as the search text, the cursor finds the character strings “N10”, “N11”, “N100” etc. Spaces are also treated as characters. For example, if you enter the search term “N1 ” (space), the cursor will only find the block numbers “N1 ”.

When you press the SEARCH softkey again, the old search text appears in the input line. You can then perform the search again.



Press the INSERT/OVERWRITE softkey to change between the insert and overwrite modes. The cursor format changes as follows:

Insert: The character is highlighted in a different color to the rest of the text.

Overwrite: The character is displayed on a black background, the character itself is displayed in inverse format.

The active mode is displayed in the header line.

You can enter characters directly at the cursor position. The new characters are inserted where the existing characters are overwritten according to the selected mode.

Generate block number
--------------------------

This softkey enters the block number from the cursor position. When the SELECT program function is used to choose the name of the program, the block number is taken from the default setting in the "Editor settings" input screen. The next time you press the GENERATE BLOCK NUMBER softkey, the block number is generated from the entry in the "Block number difference" field.

Save to disk
-----------------

The softkey SAVE TO DISK and (in the softkey bar extension (with the ETC key))

Save under
---------------

the softkey SAVE UNDER are used to save the edited program on the hard disk.

This function is described in the Section Saving programs.



If the edited part program already exists in the target workpiece, this program is overwritten on the hard disk. If you want to keep the version in the workpiece, it is recommended that you copy the whole workpiece (e.g. WELLE1) and insert it on the hard disk under a different name (e.g. WELLE10) (using function Copy to clipboard/Insert from clipboard in area PROGRAM).



#### Note

You can instead store the program only (e.g. MPF100) under a different name (e.g. MPF101) but must then activate the SELECT command in the job list (see Editing/Syntax description of job lists).

You can also change the name of the part program with the function "Rename" before saving it in directory NCK/MPF or NCK/SPF.

Tool path simulation
-------------------------

Select this softkey to call the graphic tool path simulation function (see Section "Graphic Tool Path Simulation").

### 5.1.3 Editing a new NC program

Set  
block number

A presetting can be made before editing the program.

Softkey SET BLOCK NUMBER is used to preset block number generation.

Now press this softkey:

Fig. 5.5 Block number setting

The values are entered via the keyboard with INPUT.

The presetting is "5"; i.e. the first block number is N5, the next N10 etc.

Example: N5  
N10  
N15  
...

Numerical values up to 4 digits in length can be entered.

*Note*

The block settings are also effective in TEACH IN mode.

In SW version 6.3 and later, the block number settings remain valid after POWER OFF/ON.

Edit

This calls the program.

Generate  
block number

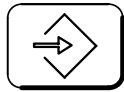
Press this softkey to generate a block number. The block number is inserted at this point according to the presetting.

...

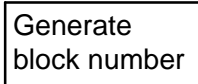
Enter the program blocks on the alphanumeric keyboard.

**Note**

You can enter spaces between the words (G, H, M... commands) for clarity. These spaces are not required. The NCK processes the program blocks even if no spaces are entered.

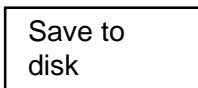


Press the INPUT key to terminate the program block. An end of block character (LF) is automatically created.



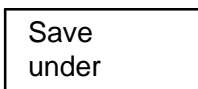
If you press the GENERATE BLOCK NUMBER softkey, the system adds the block number difference to the block number and inserts the result at the cursor position.

You must save the part program after you have completed editing.



The softkey SAVE TO DISK and

(in the softkey bar extension (with the ETC key))



the softkey SAVE UNDER are used to save the edited program on the hard disk.

This function is described in more detail in the Section Saving programs.

**Notes**

- The program entered must be completed with end of program (M30, M17, M02).
- Programming is described in more detail in the Programming Guide.
- In the PROGRAMM./EDIT NC area you can create a program
  - via the keyboard and/or
  - with the help of the operator support function (you can also work alternately with keyboard or operator support).

	<p><b>Caution:</b> the part program memory has no battery backup. If an NC program is edited in the part program memory and is to be retained in the control after it has been switched off, this program must be transferred onto the hard disk.</p>	
--	---	--

	<p>The workpiece STANDARD is setup in the workpiece directory LOCAL by the operating system. It is automatically loaded into the NCK memory on powering up the control. In STANDARD you can store global part programs or NCK data.</p>	
--	---	--

	<p>The machine tool manufacturer can configure via machine data whether the workpiece last active is to be loaded again. Precondition therefore is that a workpiece was loaded before switching off the control.</p>	
--	--	--

## 5.1.4 Graphic tool path simulation

### Tool path simulation

With the graphic tool path simulation, you can carry out a fast and easy visual check of the tool paths.

Call the function by selecting softkey TOOL PATH SIMULATION

- in the AUTOMATIC operating mode under the PROGRAM CORRECTION function or
- in the PROGRAMMING area under the NCK editor.

### Notes

- Depending on the channel, the axes to be simulated can be entered in the parameter display.
- The NCK editor selected in the part program simulates the tool path with a polymarker.
- The point at which the polymarker starts corresponds to the reference point from the machine data.
- Different colours are used to display the traversing paths travelled in rapid traverse (broken line) and path feed **or** the traversing path colors can also be set for each tool (PARAMETER SIMULATION/COLORS).
- Traversing paths can be maintained after editing or changing the channel, settable under parameter display.
- The machining time of the workpiece is calculated as a function of the feedrate from the traversing paths and can be supplemented with the tool change time and the M/H functions, settable under PARAMETER SIMULATION/TIMES.
- When an edited program is called, existing subroutines, R parameters and data such as NC data, cycle, machine and setting data are loaded from the NCK. At the same time, a workpiece TOOL PATH is set up in the GLOBAL directory under PROGRAMMING. You should not use this workpiece name anywhere else (e.g. under LOCAL).
- The graphic tool path simulation function does not load any zero point offset.
- No tool offsets are needed to execute the graphic tool path simulation. If there are any tool offsets in the program, then only the tool radius is taken into account and displayed. A polymarker is displayed with turning, the milling cutter diameter with milling.
- Alarms are acknowledged with OK and Abort. Follow-up messages appear with OK; with Abort, they are suppressed.
- The axis/spindle converter function is implemented in conjunction with Graphic Tool Path Simulation in SW version 6.1.
- The mirroring function is implemented in conjunction with Graphic Tool Path Simulation in SW version 6.2.
- Those functions can be skipped in the part program by means of the special bit @371 "Read simulation".

*Display elements*

- Axis addresses, setpoint values, tool offsets (D number), spindle numbers, spindle speed, simulation channel, program name, NC block, tool and machining time are displayed.
- If a \* is displayed in front of the current block, it is an insert block (e.g. of TRC, contour definition).
- Cycle blocks are displayed depending on cycle disable.

*Display elements*

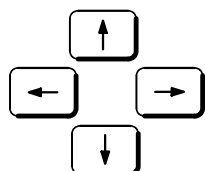
The colours in the **graphic field** represent the following (only if the color allocation to tools is active):

- **blue** = Paths travelled in rapid traverse (broken line)
- **red** = Path travelled in path feed, polymarker
- **green** = Path travelled with TRC (if cutting edge radius > 0)
- **yellow** = Threading
- **white** = Origin

*Vertical softkeys*

One of the functions of the VERTICAL SOFTKEYS is to alter the screen display.

Softkey	Description
SELECT WINDOW	If you have activated two windows by selecting the VIEWS softkey, use the SELECT WINDOW function to choose the window you want. All the functions of the vertical softkeys are available.
MAX. DISPLAY	When you select this softkey, a window-sized display of all traversing paths is output in the selected window. If there are no traversing paths, the blank is displayed in the window with the dimensions set in display PARAMETER SIMULATION.
VIEWS	The views selected under PARAMETER SIMULATION/VIEWS are displayed in succession.
ZOOM + ZOOM -	To increase or decrease the display size by 10 %.
DELETE T PATHS	The tool paths that have already been generated and stored are deleted.
CLOSE SIMULATION	When you exit the graphic tool path simulation function by selecting the softkey CLOSE SIMULATION or pressing RECALL, all the values set under search and program control remain stored.



You can shift the window contents in the direction you want by using the CURSOR KEYS.

*Horizontal softkeys* You can call the following softkeys before activating START:

Softkey	Description
START	Select this softkey to start machining at the position you stopped at after STOP, after a RESET (or after the end of a program M30, M17, M02) at the beginning of a program or after an active alarm.
RESET	The tool paths are deleted. The program is reset to the beginning.
START SINGLE BLOCK	Starts the simulation in single blocks.
SEARCH	<p>The following search methods referred to</p> <ul style="list-style-type: none"> <li>• a block in a program</li> <li>• a tool change and</li> <li>• a main block</li> </ul> <p>are provided.</p> <p>Searches in the simulation function are implemented through calculation of the traversing paths. The graphic display of the paths is suppressed until the search criterion has been found.</p>
PROGRAM MODIFIC.	<p>Program modification:</p> <p>You can set the skip levels here.</p>
R PARAMETERS	Display of current R parameter values.
PARAMETER SIMULATION	<p>This softkey calls a screen in which you can set the simulation parameters. For example, you can parameterize:</p> <ul style="list-style-type: none"> <li>• Transmit instead of 2D/3D transformation: The simulation replaces a 2D or 3D transformation with Transmit.</li> <li>• Keep traversing paths after editing: The traversing paths remain unchanged after you edit the program again or select a new channel.</li> <li>• Keep R parameters after editing: The R parameters remain unchanged after you edit the program again.</li> </ul>



Softkey	Description
PARAMETER SIMULATION	<p>The following functions are assigned to the softkeys:</p> <ul style="list-style-type: none"> <li>• CHANNEL+ or CHANNEL–: Select channel to be simulated.</li> <li>• COLORS: Set colors for tool paths according to tool (turning technology).</li> <li>• TIMES: Set time period for tool changes and M/H functions for workpiece program runtime.</li> <li>• VIEWS: Choose between side view, end face view and view of peripheral surface for turning applications. A plan view, side view or perspective view of traversing paths can be selected for milling applications. These view settings can be selected via the vertical softkey VIEWS in the basic display.</li> <li>• M/H FUNCTIONS: Call a screen in which PLC functions can be parameterized. The following parameters can be set: <ul style="list-style-type: none"> <li>– Tool change identifier This setting is required for the SEARCH function and tool change time.</li> <li>– Main spindle active Sets the main spindle by means of an M/H function to which the feedrate is referred (not on power tools).</li> <li>– Counterspindle active: Sets the counterspindle by means of an M/H function to which the feedrate is referred, e.g. when machining operation is transferred from main spindle to counterspindle (counter-clamping) (not on power tools).</li> <li>– Switch spindle actual value: A spindle can be designated as a leading spindle by means of a configurable PLC function. The PLC function is configured for spindle S1. Further leading spindles are declared implicitly through incrementation of the address number or address extension. Example: M1=26 is configured: Spindle S1 is declared as leading spindle  When M3=26 is set in the NC program, spindle 3 is declared as the leading spindle for simulation runs.</li> <li>– Mirroring: The simulation function supports axis control in mirror-image mode, the parameterization method is analogous to the M/H function for switching the spindle actual value.</li> </ul> </li> <li>• TECHNOLOGY: Choose between simulation of a milling or turning application. The simulation data parameter settings are loaded from the SIEMENS path each time the alternative technology is selected.</li> </ul>

*Horizontal softkeys* You can call the following softkeys once you have activated START:

Softkey	Description
STOP	Stops the current part program.
→ N	Suppresses the screen displays until the next block number is reached.
→ TOOL	Suppresses the screen display until the next tool change is carried out.
→ :	Suppresses the screen display until the next main block is reached.
OVERRIDE + OVERRIDE –	Increases/reduces the simulation speed.
OVERRIDE 100%	The simulation runs at “normal speed”.

## 5.1.5 Program input with operator support

### Operating sequence

Edit

Select a program and press the softkey EDIT.

Support

Then press the softkey SUPPORT and enter the program with the operator support function.

This means:

Entering an opened program with menu guidance where frequently recurring input functions are presented as a menu.

Choose operator support by calling up the SUPPORT function. After pressing the associated softkey, the following softkey functions are displayed:

Contour	Turning	Drilling and Milling	Plane			
---------	---------	-------------------------	-------	--	--	--

Contour

After pressing the CONTOUR softkey, the following display appears in which you can select contour elements.

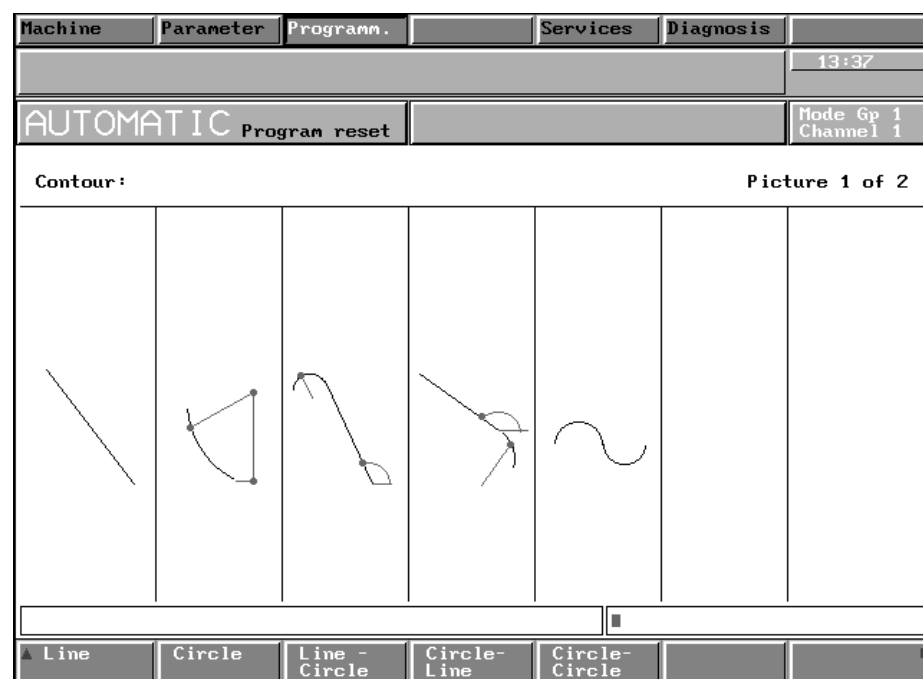
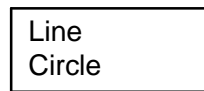


Fig. 5.6 Contour definition programming in the PROGRAMMING area

### Selecting the contour definition



You press the LINE – CIRCLE softkey, for example.



Enter the geometric values in all input fields of the contour one after the other. Each value is concluded by pressing the INPUT key.

The selected contour definition appears on the CRT with an input field for geometric values.

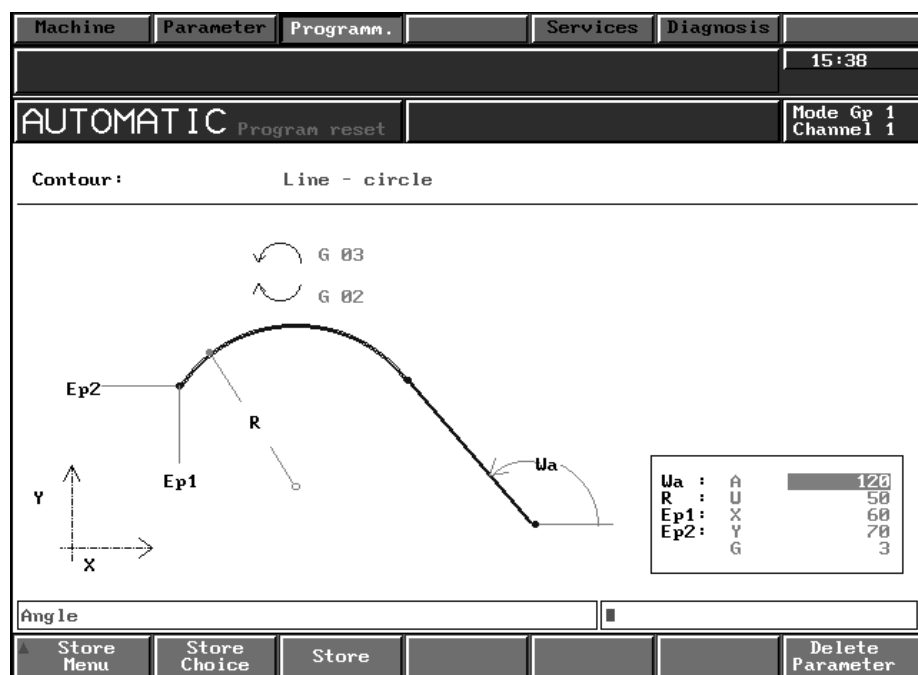
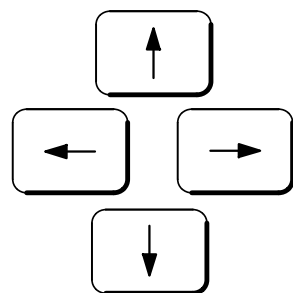
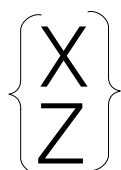


Fig. 5.7 Line – circle geometric contour programming



Using the cursor keys, you can choose any input field, for example if you wish to subsequently modify geometric values for the selected contour.

#### Note



Watch out for input fields in which several values are marked in reverse video in curly brackets on the screen ("X" and "Z" in the example).

In this case you can only enter one value (i.e. only the X value or only the Z value in the example).

If you enter more than one value, then once the STORE softkey has been pressed, the message "TOO MANY VALUES ENTERED" appears in the dialog text line.

### 5.1.6 Machining cycles

Turning

Press the softkey for the cycle required (e.g. turning).

Drilling and  
milling

*Turning cycles*

Stock r.	Thread	Drill	Groove	Undercut		
----------	--------	-------	--------	----------	--	--

*Drilling/Milling cycles*

Drilling cycles	Drilling patterns	Milling cycles	Milling patterns			
--------------------	----------------------	-------------------	---------------------	--	--	--

The machining cycles are described in detail in a separate document

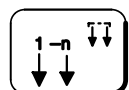
**SINUMERIK 840 C Cycles**

**Programming Guide.**

### 5.1.7 Plane

Plane

Press the PLANE softkey.



Switch over to the desired channel.

G18  
plane

Press the softkey for the desired plane, e.g. G18.

Store  
plane

Press the STORE PLANE softkey.

After entering the channel number via the keyboard and selecting the plane G17, G18 and G19, via a softkey, the abscissa, ordinate and applicant fields and the two interpolation parameters are filled in automatically by the control.

You can also define a 4th axis. You can select the leading sign for the 3rd and 4th axes using the toggle fields.

## 5.1.7 Plane



Use the selection key to select the options in the toggle fields. You can use the leading sign settings for the 3rd and 4th axes to decide on the use of the angle head mill. For example the following blocks are generated in the part program:

Output of a 4th axis:                   G16 X Z Y+ X-  
No 4th axis defined:               G16 X Z Y

By pressing the STORE PLANE softkey, you transfer the selected plane into the part program. In addition, the axes are correctly determined in the input display of the CONTOUR DEFINITION and CYCLES function.

On selecting the G16 plane, you enter the values marked in the input display via the keyboard.



By pressing the STORE PLANE softkey, you transfer the G16 plane into the part program. You then additionally enter the axis names in the part program.

## 5.1.8 Program management

### 5.1.8.1 Changing the access rights

The following access rights can be defined in the overview displays:

- r – Program is read only
- w – Program is read/write
- x – Program is executable
- c – Program is copy-protected (cycle protection)

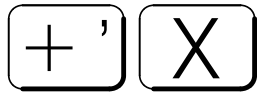
	<p>The cycle protection (access right c) only functions in combination with the "Cycle disable" interface signal, i.e. this function must be configured by the machine manufacturer. Please refer to the manufacturers documentation.</p>	
---	---	---

*Precondition*

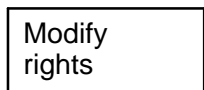
The PROGRAMMING/EDIT NC area is selected. The main program or subroutine overview display has been called up.

*Operating sequence*

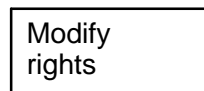
Enter the name of the program whose access rights you want to change (or select the program in directory NCK with the cursor) and



insert an access right with +X (executable program), or



remove an access right with -X (executable program).

*Example*

A program has the following access rights:

MPF1 -rw-

I.e., MPF1 is read/write but cannot be processed.

Use the +X operator action and the MODIFY RIGHTS softkey to define MPF1 as follows:

MPF1 -rwx.

I.e., MPF1 can be selected for processing.

➔	<p>The access rights are <b>not</b> saved on the hard disk with the SAVE function, i.e. the selected access rights are deleted when the system is switched off. Whenever a program is loaded from the hard disk it automatically receives the rights -rwx (read, write, executable), regardless of the access rights the program had when it was saved to the hard disk.</p>	➔
---	--	---





Machine	Parameter	Programm.		Services	Diagnosis	
						13:44
<b>AUTOMATIC</b>		Program reset				Mode Gp 1 Channel 1
Programming USER/NCK						
Copy from:		USER/NCK/MPF/MPF1				
Mark: EXT = external program    c = cycle protect.    x = executable r = readable              w = writable						
Copy to:		USER/NCK/MPF/MPF1				
Name	Type	Length	Mark	Comment in the 1st block/date		
MDA	parp					
MPF	parp					
SPF	parp					
				MPF11■		
		Copy				

Fig. 5.9 Copy MPF/SPF 02

Copy

Now press the softkey COPY – this copies program MPF2 to MPF11.

The basic display for editing automatically reappears. MPF11 is now listed in the program overview.

*Note*

A main program can also be copied to a subroutine (and vice versa). The different possible program ends (i.e. M17/M30/M02) should be checked if this is done.

MDA programs can be copied to main programs and subroutines – but not vice versa (i.e. an MPF/SPF to an MDA program).

The copy function described above is executed in the part program memory, i.e. a copied program is **not** backed up when it is copied.

The program must be backed up using the softkey functions **SAVE TO DISK** and **SAVE UNDER**.

### 5.1.8.3 Rename program

*Operating sequence:* Select a part program (MPF/SPF) with the cursor.

Rename

Now enter the new program number (e.g. MPF 2222) and press the softkey RENAME. The program is copied.

If you press the softkey **RENAME** before entering the program number, the message “Please enter new name” appears.

Do this and then press the softkey **RENAME** again.



## Deleting by entering the program number

*Operating sequence*

The program with the number entered is deleted by the control.

or

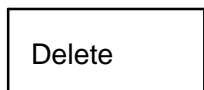
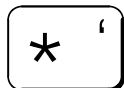


all of the programs with the identifiers before the “\*” are deleted

**e.g. MPF1\***

MPF1, MPF10, MPF12, MPF100, etc. are deleted.

If you enter MPF\* or SPF\* all of the main programs or subroutines are deleted



or



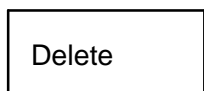
you can use the question mark to represent an individual character in the program name.

**e.g. MPF?**

All of the main programs with a single-digit program number are deleted, i.e. MPF0 to MPF9

**MPF2??**

The programs MPF 200 to 299 are deleted.

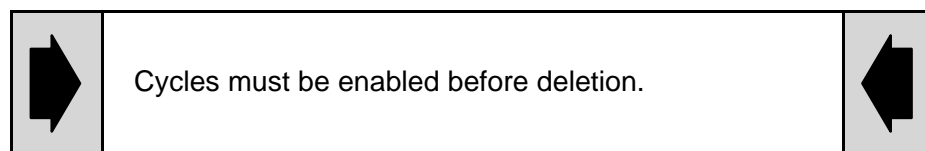


You can also delete areas of a program by entering the start and end of the area and pressing the delete softkey. For example, if you enter

% 700 □ %710 or



%700=%710

all of the main programs between %700 and %710 are deleted.

*Note*

As long as a part program has been selected in the editor (EDIT NC), it cannot be deleted. The program can be released for deletion by selecting another program.

### 5.1.8.5 Move cycles

	Cycles can be prevented from unauthorized read-out by the PLC. This provides an active protection of the expertise of the machine manufacturer or the user. It is then no longer possible for the operator to disable and enable at will (see machine tool manufacturer's documentation).	
---	---	---

<i>Load cycles</i>	The SIEMENS cycles and the user cycles are stored on the hard disk. You can load the cycles into the memory using the LOADCYC job list command (see the Section "Syntax description for job lists").
<i>Move cycles</i>	If the cycle disable (see the machine manufacturer's documentation) is configured, you can cancel a cycle protection by modifying the access rights (see the Section "Changing the access rights"). You can now rename, copy and edit the cycle just like a normal subroutine. You can assign the cycle as a subroutine to a workpiece.
<i>Move subroutine to cycle</i>	A subroutine can be loaded as a cycle in a local or global workpiece. This type of subroutine or cycle cannot be protected by a cycle disable. If you save subroutines with the access right "Cycle protection" (c) on the hard disk, the access rights are <b>not</b> saved (see the Section "Changing the access rights").  The cycle disable must be configured by the machine manufacturer, i.e. it can only be created by the machine manufacturer. A cycle protection can be assigned to cycles. These user cycles are not stored in the LOCAL or GLOBAL directory, but are stored under password protection on the hard disk.
<i>Note</i>	You will find the description of how to include the machine manufacturer cycles in "Installation Instructions SINUMERIK 840C Software Versions 1 to 5".

### 5.1.9 FANUC program converter (from SW 4.9)

The FANUC converter allows you to convert FANUC 0T programs to the DIN code of the SINUMERIK 840C.

#### Note

The FANUC program must be syntactically correct for successful conversion. Owing to the very wide variety of programming options, however, we cannot guarantee that manual corrections will not be necessary in some programs after they have been converted with the FANUC converter.

<i>Creation</i>	<p>Creating FANUC programs on the SINUMERIK 840C: Like other part programs, FANUC 0T programs are stored in workpieces. They have the identifier EPF (External Program File). A part program with a FANUC 0T code can be integrated in the control in the following way:</p>
<i>Generation</i>	<p>Generating a program with the FANUC 0T code in the editor: Create a new workpiece, create a new program with input Exxx or EPFxxx, edit and store. A JOB LIST with entry "LOAD" is automatically generated when you create a new workpiece.</p>
<i>Import</i>	<p>Importing FANUC programs via the V.24 (RS232C) interface: FANUC programs are imported into the 840C control in punchtape format via the V.24 interface. You can specify an existing or new workpiece as the destination. On import, the FANUC program number "Onnnn" or the program identifier ":nnnn" is converted to the MMC program number "EPFnnnn". The program is not otherwise changed during import.</p>
<i>Starting the converter</i>	<p>You can initiate the conversion of a type EPF part program by calling the converter in the PROGRAM\WORKPIECES area using softkey "Start converter". The converter is started automatically when you load an EPF or a workpiece that contains an EPF. When you start the converter manually, the program is <b>always</b> converted. When it is started by "Load workpiece", the conversion is carried out only if the EPF is more recent than the DIN programs generated by the conversion. This would be the case, for example, if the EPF has never been converted or has been modified with the Editor after conversion. Whether the converter is started explicitly or implicitly through loading of a workpiece, a message is displayed and an error log stored in the current workpiece if a conversion error occurs.</p>

<i>Call</i>	<p>You can call the program converter by making the following inputs: <b>In the PROGRAMWORKPIECES area, select</b> a workpiece, or a program in a workpiece which contains a file of type EPF, and press softkey "START CONVERTER". The converter creates an MPF...parp file from the EPF...epf file as well as the SPF...parp subprograms belonging to the MPF file. With the inputs described above, the EPF program is always converted and the existing programs of type PARP thus overwritten. The part programs generated by the conversion are stored in the workpiece.</p>
<i>Loading in AUTOMATIC</i>	<p><b>Loading a part program in AUTOMATIC mode</b> Press softkey "Select program" in AUTOMATIC mode and select a workpiece which contains a file of type EPF. When you press the "Select program" softkey again, the conversion operation commences, provided that the external part program (EPF) is more recent than the part programs (MPF or SPF) to be generated from it. When the conversion is successful, the generated files of type PARP are stored in the workpiece and loaded to the working memory for execution. If you select the same workpiece again, the converter will start only if the program code in the EPF file has changed. If a compilation error occurs during conversion, the generated PARP files will be stored in the workpiece, but will not be loaded to the working memory.</p>
<i>Loading Services</i>	<p><b>Loading programs in Services/NC</b> In the Services/NC area, select a workpiece, or a program in a workpiece which contains a file of type EPF and press softkey "Start load". The conversion run then commences. After a successful conversion, the generated files of type PARP are stored in the workpiece and loaded to the working memory for execution. If you select the same workpiece again, the converter will start only if the program code in the EPF file has changed. If a compilation error occurs during conversion, the generated PARP files will be stored in the workpiece, but will not be loaded to the working memory.</p>
<i>Display</i>	<p><b>Displaying conversion errors/conversion warnings</b> Errors that have occurred during conversion are recorded in an error log. If an error log has been generated, the message "Conversion error in workpiece &lt;wkst&gt;" is displayed. You can display the error log using the double editor in the PROGRAM\WORKPIECES\Converter (SK "Error log"). The program to be converted is displayed in the upper half of the screen and the error log in the lower half. The error log is stored in the workpiece which contains the EPF part program and is updated with every conversion. To display the log, therefore, the relevant workpiece, or an object it contains, must be selected with the data selector. To locate the errored program code in the EPF file quicker, the individual error message can be selected in the error log with the cursor key and displayed by pressing SK "Error location".</p>

A setting can be made in the BEDCONF configuration file in directory Diagnosis/Start-up/PC Data/Operation/Basic Settings to suppress the output of a message in response to a conversion warning.

Entry I 1 2 16002=1 Message "Conversion warnings in work-piece

<wkst>"

Entry I 1 2 16002=0 No message on conversion warnings

#### Converter settings

To allow adjustment of the program converter to the actual machine data (e.g. reference point values of axes for G28 function), defaults must be set when the converter is started up in the SINUMERIK 840C. File KONVFO.INI can be copied from path Standard\Operation\GCODE by SK "Preset" to path User\Operation\GCODE in the Diagnosis/Start-up/PC Data area. Here, it can be modified with the Editor and thus adapted to the applicable machine data. Modifications to file KONVFO.INI are activated immediately after they are saved!

#### Note

For parameters required by FANUC, please refer to the FANUC documentation.

#### File KONVFO.INI

Parameter	Description
;The file KONVFO.ini is divided into three individual sections. ;1st: [OPTIONS] ;2nd: [CCYCLE] ;3rd: [MTABLE] ;Attention! ;The positions of the different parameters included in the sections can be varied, but never the less the section including the parameter cannot be changed.	The file KONVFO.ini is divided into three individual sections. 1. [OPTIONS] 2. [CCYCLE] 3. [MTABLE] Attention! The positions of the different parameters included in the sections can be varied, but never the less the section including the parameter cannot be changed.
;OPTIONS, including parameters ; ;GCode, CAxisExist, InputWithPoint, MD5002_4, Inch, Diameter, Semicolon, Language, MinUN_V, MaxUN_V, MinRP_V, MaxRP_V, XRefP_V, ZRefP_V, N735, N736, N780, N781, N784, N785	Options: Existing parameters
;Selection of the FANUC G Code System ;0 is the equivalent of G code system A ;1 is the equivalent of G code system B ;2 is the equivalent of G code system C GCode=0	See FANUC 0T, with what type of program code was the FANUC program created.
;0 No C axis exists ;1 C axis exists CAxisExist=0	C axis present on machine?

Parameter	Description															
<p>;0 Pocket cal. type dec. point input 1000. -&gt; 1000/1000 -&gt; 1000 ;1 Former type decimal point input 1000 .-&gt; 1000/1000 -&gt; InputWithPoint=1</p>	See input format in FANUC program															
<p>;Selection of Inputsystem SIN 840C (Default) ;0 SIN 840C inputsystem Metric ;1 SIN 840C inputsystem Inch MD5002_4=0</p>	<p>On the SIN 840C, the unit of measurement of feedrate F and cutting rate S are determined by the preset system of measurement, irrespective of program code G70 (inch) or G71 (metric) in the part program. To match the FANUC 0T and SIN 840C programs, these units of measurement are adjusted by parameter MD 5002.4.</p> <p>Calculation of setting for parameter MD 5002.4:</p> <p>Edit the following program sequence in MDA mode and then execute:</p> <p>@302 R0 K5002 K4</p> <p>Now read out the value of R parameter R0 and enter it in parameter MD 5002.4.</p>															
<p>;Selection of Inputsystem FANUC (Default) ;0 FANUC inputsystem Metric ;1 FANUC inputsystem Inch Inch=0</p>																
<p>;Adjustment of SIN 840C Face Axis ;0.5 Radius programming and read face axis position in diam. ;1 Radius programming and read face axis position in radius ;1 Diameter programming and read face axis position in diam. ;2 Diameter programming and read face axis position in radius Diameter=1</p>	<p>The SINUMERIK 840C offers various setting options for programming and reading positions of the facing axis. To ensure that the converter can take these different settings into account, it must be adjusted in this parameter.</p> <p>Calculation of value for "Diameter" parameter:</p> <p>Edit the following program instruction in MDA mode and then execute:</p> <p>@302 R0 K5011 K7 @302 R1 K5011 K5</p> <p>Now read out the values of R parameters R0 and R1 and calculate the entry for "Diameter":</p> <table><tr><td>R0</td><td>R1</td><td>"Diameter"</td></tr><tr><td>0</td><td>0</td><td>1</td></tr><tr><td>0</td><td>1</td><td>2</td></tr><tr><td>1</td><td>0</td><td>0.5</td></tr><tr><td>1</td><td>1</td><td>1</td></tr></table>	R0	R1	"Diameter"	0	0	1	0	1	2	1	0	0.5	1	1	1
R0	R1	"Diameter"														
0	0	1														
0	1	2														
1	0	0.5														
1	1	1														



Parameter	Description
;Display Source Code as Comment due to Semikolon ;0 No ;1 Yes Semikolon=0	The converter provides the option of transferring the source code of the FANUC program as a comment into the converted SINUMERIK 840C program. These comments are not displayed in the alarm display text field during part program processing. <b>Example:</b> Parameter semicolon= 0 FANUC            SINUMERIK 840C G50 S1250       G92 S1250 <b>Example:</b> Parameter semicolon= 1 FANUC            SINUMERIK 840C G50 S1250       ;G50 S1250 G92 S1250 Note: The source code is not stored in the subprograms for contour definition implemented by FANUC 0T codes G70 to G73. It is stored in the main program after the call for machining cycle L95.
;Languages ;0 Deutsch ;1 English ;2 Español ;3 Français ;4 Italiano Language=1	Converter messages are output in the selected language.
Minimum subprogram number MinUN_V=101	Lowest subprogram number for contour definition implemented by FANUC 0T codes G70 to G73. Note: Setting must not clash with user subprograms
;Maximum subprogram number MaxUN_V=200	Highest subprogram number for contour definition implemented by FANUC 0T codes G70 to G73. Note: Setting must not clash with user subprograms
Minimum R-Parameter number MinRP_V=200	Lowest R parameter number for supplying Siemens machining cycles and backing up program data.
Maximum R-Parameter number MaxRP_V=299	Highest R parameter number for supplying Siemens machining cycles and backing up program data. Note: R parameters must be < 300
Reference point X axis XRefP_V=1000	Reference point value of X axis.
Reference point Z axis ZRefP_V=2000	Reference point value of Z axis.

Parameter	Description
;G30 - 2nd reference position X axis (incrm.) N735=0 ;G30 - 2nd reference position Z axis (incrm.) N736=0 ;G30 P3 - 3rd reference pos. X axis (incrm.) N780=0 ;G30 P3 - 3rd reference pos. Z axis (incrm.) N781=0 ;G30 P4 - 4th reference pos. X axis (incrm.) N784=0 ;G30 P4 - 4th reference pos. Z axis (incrm.) N785=0	Distances between axes and reference point positions of G28 function for G30, G30 P2, G30 P3 and G30 P4 applications.
;Parameter of FANUC 0T ;No. Parameter ;N717 U(delta d)/W(delta d) ;N718 R(e) ;N719 U(delta i) ;N720 W(delta k) ;N721 R(d) ;N722 R(e) ;N723 P(m) ;N109 P(r) ;N724 P(a) ;N725 Q(delta d min) ;N726 R(d) ;N13 GOFU2 ;N14 T2D ;N19 XRC [CCYCLE] N717=0 N718=0 N719=0 N720=0 N721=0 N722=0 N723=01 N109=10 N724=60 N725=0 N726=0 N13=0 N14=0 N19=0	See FANUC 0T description Function Canned cycle G71/G72 Canned cycle G71/G72 Canned cycle G73 Canned cycle G73 Canned cycle G73 Canned cycle G74/G75 Canned cycle G76 Canned cycle G76 Canned cycle G76 Canned cycle G76 Canned cycle G76 Canned cycle G76 T code for tool offset T code for tool offset Radius designated for X axis
. ; ;Example for the 3rd M function : M4 has to convert to M6: ;From_3=4 ;To_3=6 ; ;	

Parameter	Description
[MTABLE] From_1=8 To_1=8 From_2=9 To_2=9 ;	Adapt M functions of FANUC machine tool to the machine tool operated by SINUMERIK 840C If no FANUC machine tool is installed, then create a 1:1 list. M0, M01, M02, M03, M04, M05, M30, M98 and M99 are automatically compiled by the converter and need not be entered in the table.
End of file	

#### *Supplementary conditions*

Depending on the memory configuration of the MMC CPU (8 MB RAM), the following applications cannot be started while the converter is running: Programming system (WOP), simulation (SIM), tool path simulation (WKZ); machine data interactive mode (MDD) and start-up. These applications are automatically disabled as long as the converter is running. A memory configuration of 16 MB is recommended.

# **FANUC G code conversion list**

Page	Function	FANUC 0T G CODE A / B / C G functions listed singly below are valid for all 3 axes.	Only these words allow- ed in block	SINUMERIK 840C	G group, modal, basic set- ting	Comment
	Tape start/end	%	yes	( empty )		
	Program number	Oxxxx and M02 or M30	yes	%MPFxxxx		A colon (:) can be used instead of a O in the ISO code
	Subprogram num- ber	Oxxxx and M99	yes	%SPFxxxx		
	Skip block	/x		/x		x: 0–9 for Fanuc, 0–8 for SINUMERIK 840C
	Block number	Nxxxx		Nxxxx		
	Travel in rapid tra- verse	G00		G00	1,M,G	
	Traverse at feed- rate	G01		G01	1,M	
	Circular interpola- tion	G02		G02	1,M	
	Circular interpola- tion	G03		G03	1,M	
	Dwell time in sec- onds	G04 Xxxxx G04 Uxxxx G04 Pxxxx.xx	yes	G04 Xxxxx or G04 Fxxxx	0	The delay time is specified in seconds 99999.999 The delay time is specified in seconds 99.999 No decimal point, value x 0.001 seconds 0.1 to 99.9
15–58	Change in tool off- set value	G10P X Y Z R Q (abso- lute) G10P U V W C Q (incr.)		@420 K1 D No. P No. value @423 K1 D No. P No. value	0	P<100: Write to geometry memory D No. = P P No. = 2 (X,U); 3 (Z,W); 4 (R,C); 1 (Q); (Y,V error) P>100: Write to wear memory D No. = P–100 P No. = 5 (X,U); 6 (Z,W); 7 (R,C); 1 (Q); (Y,V error)
						e.g.: G10 P11 X1.2 Z2.4 R0.1 Q3 @420 K1 K11 K2 K1.2 (D11,P2: 1.2 length 1 geometry) @420 K1 K11 K3 K2.4 (D11,P3: 2.4 length 2 geometry) @420 K1 K11 K4 K0.1 (D11,P4: 0.1 radius geometry) @420 K1 K11 K1 K3 (D11, P1:3 type)
						e.g.: G10 P120 U1.2 W2.4 C0.1 Q2 @423 K1 K20 K5 K1.2 (D20,P2: 1.2 length 1 wear) @423 K1 K20 K6 K2.4 (D20,P3: 2.4 length 2 wear) @423 K1 K20 K7 K0.1 (D20,P4: 0.1 radius wear) @423 K1 K20 K1 K2 (D20, P1:2 type)
	Plane selection	G17		G17	16,M	
	Plane selection	G18		G18	16,M	
	Plane selection	G19		G19	16,M	
	Inch selection	G20		G70	6,M	
	mm selection	G21		G71	6,M	G20/G21 of FANUC is not identical to G70/G71 of SIEMENS!
	Stored stroke limit ON	G22			9,M,G	
	Stored stroke limit OFF	G23			9,M	
	Spindle speed monit. OFF	G25		No corresponding function	8,M,G	Check spindle speed deviation
	Spindle speed monit. ON	G26		No corresponding function	8,M	Check spindle speed deviation
7–2	Reference point approach check	G27 <IP>	yes	G27 G0 Xxx Zzz L <sub>F</sub> @714 L <sub>F</sub> @361 R200 K @361 R201 K2 L <sub>F</sub> @171 R200 Xref1 Kn @121 R201 Zref1 K1 @100 Kn+1 L <sub>F</sub> N1100 (G27: reference point not reached) L <sub>F</sub> @100 K-N Kn+1	0	A random point is approached. The approach position is then checked to see whether it corresponds to the G28 reference position.

## 5.1.9 FANUC program converter

Page	Function	FANUC 0T G CODE A / B / C G functions listed singly below are valid for all 3 axes.	Only these words allow- ed in block	SINUMERIK 840C	G group, modal, basic set- ting	Comment
7-1	Travel to ref. point via interm. point	G28 <IP>		G0 <IP> @706 G0 G90 Xref1 Zref1	0	A random point is approached first, followed by the 1st reference point. If no intermediate point is programmed, then the last programmed intermediate position is applied. Define point in settings (X, Y, Z, C)!!!
7-3	Approach 2nd, 3rd or 4th reference point	G30 P2 (3,4) <IP>		G0 <IP> @706 G0 G90 Xref1+Δref2 (3,4)	0	
16-1	End cutting (delete distance to go)	G31			0	
	Thread cutting long- itudinal, face, co- nical	G33 Wxxxx Ffff G33 Uxxxx Ffff G33 Xxxx Zxxx Ffff		G33 Zxxxx Kfff G33 Xxxxx Lfff G33 Xxxx Zxxx Kfff	1,M	If no thread pitch (P) is programmed in Fanuc, the last programmed pitch is applied. When conical threads are machined with Fanuc, the pitch applies to the axis in which the longest distance is traversed (with Siemens progr. of I or K)
	Thread cutting with variable pitch	G34 Xxx Zxx Fff Kkk G34 Xxx Zxx Fff K-kk		G34 Xxxx Zxxx Kff Fkk G35 Xxxx Zxxx Kff Fkk	1,M	Increasing pitch Decreasing pitch
16-3	Autom. tool com- pensation in X	G36		No corresponding function	0	
16-3	Autom. tool com- pensation in Z	G37		No corresponding function	0	
	Tool radius com- pensation OFF	G40		G40	7,M,G	
	Tool radius com- pensation on left of contour	G41		G41	7,M	
	Tool radius com- pensation on right of contour	G42		G42	7,M	
	Set coordinate system	G92 Xxxxx Zzzzz G92 Uuuuu Wwww	yes	G58 Xxxxx Zzzzz G58 G91 Uuuuu Wwww	0	Mixed programming of G90/G91 in one block permissible
	Max. const. cutting rate	G92 Sxxxx		G92 Sxxxx		Max. speed at constant cutting rate
	Call customer macro	G65		Not implemented	0	
	Call (modal) customer macro	G66			12	
	Delete mod. call	G67			12	
	Mirroring double revolver ON	G68		No corresponding function	4,M	
	Mirroring double revolver OFF	G69		No corresponding function	4,M,G	
	Final machining cycle	G70 P(ns) Q(nf)		L subprogram number	0	Finishing without prior G7x roughing operation
	Final machining cycle	G70 P(ns) Q(nf)		R20=subprogram number R21=X value from 1st line of subprogram R22=Z value from 1st line of subprogram R24=u R25=w R26=d	0	Subprogram number for contour Starting point of contour in X (absolute)  Starting point of contour in Z (absolute) Finishing allowance in X (incremental) Finishing allowance in Z (incremental) Rough cut depth in X or Z (R29 depend. on machining)
				R27=41/42 R28=f R29=21/22 R30=1 fixed L95		Cutter radius compensation (G41, G42) Feed velocity (applies only to L95) Type definition for roughing and finishing Feed factor for infeed velocity for undercutting

## 5.1.9 FANUC program converter

Page	Function	FANUC 0T G CODE A / B / C G functions listed singly below are valid for all 3 axes.	Only these words allow- ed in block	SINUMERIK 840C	G group, modal, basic set- ting	Comment
	Chip removal in turning operation	G71 U(d) R(e) G71 P(ns) Q(nf) U(u) W(w) F(f) S(s) T(t)		@714 @360 Rglob K1 @360 Rglob+1 K2 S(s) T(t) D(t) R20=new subprogram num- ber R21=X value from 1st line of subprogram R22=Z value from Stop sub- program R24=u R25=w R26=d	0	Subprogram number for contour  Starting point of contour in X (absolute)  Starting point of contour in Z (absolute)  Finishing allowance in X (incremental) Finishing allowance in Z (incremental) Rough cut depth in X or Z (R29 depend. on machining)
				R27=41/42 R28=f R29=31/33 R30=1 fixed L95		Cutter radius compensation (G41, G42) Feed velocity (applies only to L95) Type definition for roughing and finishing Feed factor for infeed velocity for undercutting
	Chip removal in surface machining operation	G72 W(d) R(e) G72 P(ns) Q(nf) U(u) W(w) F(f) S(s) T(t)		@714 @360 Rglob K1 @360 Rglob+1 K2 S(s) T(t) D(t) R20=new subprogram num- ber R21=X value from Stop sub- program R22=Z value from 1st line of Stop subprogram R24=u R25=w R26=d	0	Subprogram number for contour  Starting point of contour in X (absolute)  Starting point of contour in Z (absolute)  Finishing allowance in X (incremental) Finishing allowance in Z (incremental) Rough cut depth in X or Z (R29 depend. on machining)
				R27=41/42 R28=f R29=32/34 R30=1 fixed L95		Cutter radius compensation (G41, G42) Feed velocity (applies only to L95) Type definition for roughing and finishing Feed factor for infeed velocity for undercutting
	Pattern repeat	G73 U(i) W(k) R(d) G73 P(ns) Q(nf) U(u) W(w) F(f) S(s) T(t)		R20,R21,R22,R27,R28,R30 as for G71 R26=0 R29=21 to 24 R24=u+i R25=w+k Rxy=d  Nxy L95 Rxy=Rxy-1 R24=R24-i(1-Rxy/d) R25=R25-k(1-Rxy/d) @126 Rxy K1 K-xy	0	Rough cut depth Finishing in parallel with contour Finishing allowance in X Finishing allowance in Z Number of divisions in free R parameters  Reduce finishing allowance Jump to Nxy to d=1
	Interrupted drilling in Z axis	G74 R(e) G74 X(U) Z(W) P(di) Q(dk) R(dd) F(f)		R11=0 R22=Z(W) R24=Q(dk) R25=Q(dk) R26=Z(W) R27=0 R28=0 Nxxx X=R 0 L98	0	Chipbreaking Starting point in (Z) Cutting depth 1st cutting depth Final drilling depth Dwell time at starting point Dwell time at drilling depth
	Grooving in Z axis	G74 R(e) G74 X(U) Z(W) P(di) Q(dk) R(dd) F(f)		R11=0 R22=Z(W) R24=Q(dk) R25=Q(dk) R26=Z(W) R27=0 R28=0 Nxxx X=R_0 L98		Chipbreaking Starting point in (Z) Cutting depth 1st cutting depth Final drilling depth Dwell time at starting point Dwell time at drilling depth
	Interrupted drilling in X axis	G75 R(e) G75 X(U) Z(W) P(di) Q(dk) R(dd) F(f)		R11=0 R22=Z(W) R24=Q(dk) R25=Q(dk) R26=Z(W) R27=0 R28=0 Nxxx X=R_0 L92		Chipbreaking Starting point in (X) Cutting depth 1st cutting depth Final drilling depth Dwell time at starting point Dwell time at drilling depth

Page	Function	FANUC 0T G CODE A / B / C G functions listed singly below are valid for all 3 axes.	Only these words allow- ed in block	SINUMERIK 840C	G group, modal, basic set- ting	Comment
	Grooving in X axis	G75 R(e) G75 X(U) Z(W) P(di) Q(dk) R(dd) F(f)		R11=0 R22=Z(W) R24=Q(dk) R25=Q(dk) R26=Z(W) R27=0 R28=0 Nxxx X=R_0 L92	0	Chipbreaking Starting point in (X) Cutting depth 1st cutting depth Final drilling depth Dwell time at starting point Dwell time at drilling depth
	Thread cutting cycle with cut segmentation	G76 P(m r a) Q(dmin) R(d) G76 X(U) Z(W) R(i) P(k) Q(dd) F(l)		R20=F(l) R21=X(U) R22=Z(W) R23=P(m) R24=P(k) R25=R(d) R26=0 R27=0 R28=roughing cuts R29=P(a)/2 R31=end point X R32=end point Z L97	0	Thread pitch Starting point in (X) Starting point in (Z) Number of repeat roughing cuts Thread depth First cutting depth Run-in path Run-out path Roughing cuts Infeed angle End point in X End point in Z
		G80			10,M,G	
		G83			10,M	
		G84			10,M	
		G87			10,M	
		G88			10,M	
		G89			10,M	
	Fours cycle: Cylinder and taper turning	G77 Uuu Www Rrr Fff		G0 G91 X=uu+rr G1 Zww X-rr Fff X-uu G0 Z-ww	1,M	
14-1		G77 Xxx Www Rrr Fff		@714 @360 Rvar1 K1 G0 G90 X=xx+rr G1 G91 Zww X-rr Fff G90 X=Rvar1 G0 G91 Z-ww		
		G77 Uuu Zzz Rrr Fff		@714 @360 Rvar1 K2 G0 G91 X=uu+rr G1 X-rr G90 Zzz Fff G91 X-uu G0 G90 Z=Rvar1		
		G77 Xxx Zzz Rrr Fff		@714 @360 Rvar1 K1 @360 Rvar2 K2 G0 G90 X=xx+rr G1 Zzz G91 X-rr Fff G90 X=Rvar1 G0 Z=Rvar2		
	Fours cycle: Longitudinal and taper thread	G78 Uuu Www Rrr Fff		G0 G91 X=uu+rr G33 Zww X-rr Kff G0 X-uu Z-ww	1,M	
14-3		G78 Xxx Www Rrr Fff G78 Uuu Zzz Rrr Fff G78 Xxx Zzz Rrr Fff		See G77 implementation		
	Fours cycle: Face and taper turning	G79 Uuu Www Rrr Fff		G0 G91 Z=ww+rr G1 Xuu Z-rr Fff Z-ww G0 X-uu	1,M	
14-5		G79 Xxx Www Rrr Fff G79 Uuu Zzz Rrr Fff G79 Xxx Zzz Rrr Fff		See G77 implementation		
	Const. cutting rate on	G96 Sxxxx		G96 Sxxxx	2,M	Cutting rate in m/min
	Const. cutting rate off	G97 G97 Sxxxx		G97 G95 Sxxxx	2,M,G	
	Feed mm/min	G94		G94	5,M	

## 5.1.9 FANUC program converter

Page	Function	FANUC 0T G CODE A / B / C G functions listed singly below are valid for all 3 axes.	Only these words allow- ed in block	SINUMERIK 840C	G group, modal, basic set- ting	Comment
	Feed mm/rev.	G95		G95	5,M,G	
	Absolute dimen- sions	G90		G90	3,M,G	
	Incremental dimen- sions	G91		G91	3,M,G	
	Return to start plane	G98			11	
	Return to retraction plane	– / G99 / G99			11	
	Cylinder interpola- tion	G107				
	Polar coordinates Ipo ON	G112			25	
	Polar coordinates Ipo OFF	G113			25	
	Orient. spindle stop, incremental via M functions	M(a+0) to M(a+5)		The positioning range for M19 S.. is 0.1 to 359.9 de- grees. The spindle is positio- ned from standstill via the shortest possible route.		a = Parameterizable numerical value of first M function b = Parameterizable indexing angle Parameterizable direction of rotation e.g.: spindle position=20°, a=50, b=30, positive direction M52 (20°+3°30°=110°) M55 (20°+6°30°=200°)
	Orient. spindle stop, absolute and incremental via C or H	G code System A: C+---xxx.x H+---xxx.x  G code System B, C: C+---xxx.x		The positioning range for M19 S.. is 0.1 to 359.9 de- grees. The spindle is positio- ned from standstill via the shortest possible route.		G code System A (without G90 and G91): C+---xxx.x absolutely to +---xxx.x H+---xxx.x increm. by +---xxx.x  G code System B or C (with G90 and G91): (G90 active) C+---xxx.x absolutely to +---xxx.x (G91 active) C+---xxx.x increm. by +---xxx.x
	X axis with linear interpolation	Xxxxx		Xxxxx		
	Y axis with linear interpolation	Yxxxx		Yxxxx		
	Z axis with linear in- terpolation	Zxxxx		Zxxxx		
	C axis with linear interpolation	Cxxxx		Cxxxx		
	Incremental X	Uxxxx		G91 Xxxx	Grp3=91	1st axis basic setting group 3 = G30
	Incremental Y	Vxxxx		G91 Yxxx	Grp3=91	4th axis
	Incremental Z	Wxxxx		G91 Zxxx	Grp3=91	2nd axis
	Incremental C	Hxxxx		G91 Cxxx	Grp3=91	3rd axis
	Interpolation parameter I	Ixxx		Ixxx		
	Interpolation parameter K	Kxxx		Kxxx		
	Radius	Rxxxx		Bxxxx		
	Angle	Axxxx		Axxxx		
	Feedrate F	Fxxx		Fxxx		
	Spindle speed cut- ting rate	Sxxxx		Sxxxxx		
	Tool selection	Txyy Txy		Txx Dyy Tx Dy		xx tool no. yy offset no. x tool no. y offset no.
	M functions	M00		M00		Program stop
		M01		M01		Optional stop
		M02		M02		End of program
		M30		M30		End of program
	Subprogram call	Xxx Zxx M98 PpppIIII	no	Xxx Zxx LIII Pppp		IIII=subprogram number; pppp=number of runs
	Branch to address	M99 Pxxxx		@100 Kxxxx or @100 K---xxxx		@100 Kxxxx or @100 K---xxxx if the target lies towards the program beginning. M99 without specified target in the main program means a branch to the program beginning.
	End of subprogram	Xxxx Zyyyy M99		Xxxx Zyyyy M17		If programmed alone in block > end of subprogram

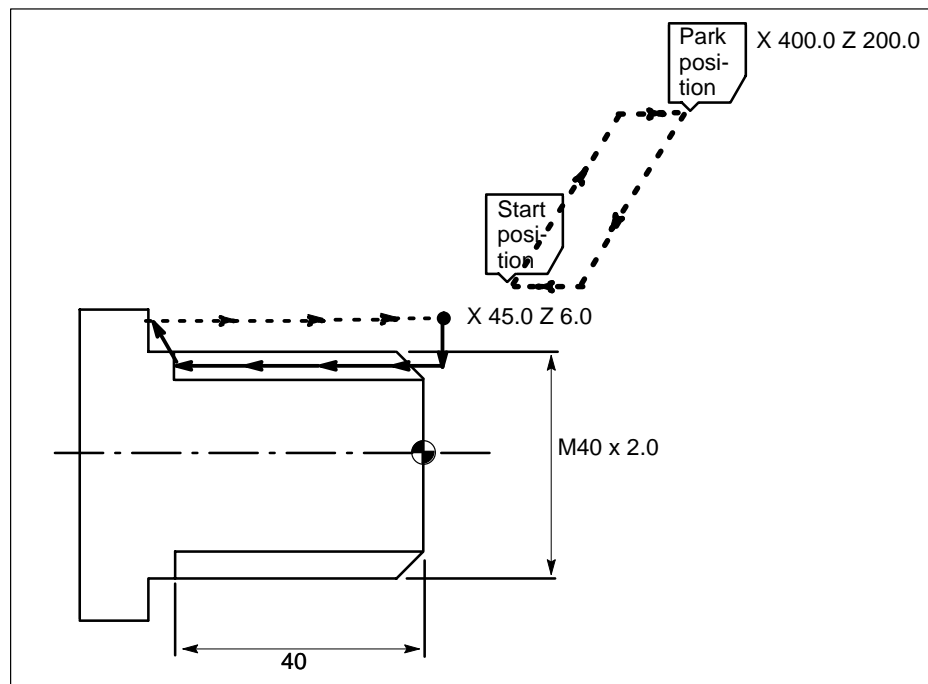


## 5.1.9 FANUC program converter

Page	Function	FANUC 0T G CODE A / B / C G functions listed singly below are valid for all 3 axes.	Only these words allow- ed in block	SINUMERIK 840C	G group, modal, basic set- ting	Comment
	Subprogram with destination	M99 Pxxxx				Conversion not possible because Siemens system only allows branching within the same program
	Auxiliary functions	Bxxxxxxxx		Hxxxxxxxx		Value range for Fanuc: -99999999 to +99999999
	Comment	(Comment)		(Comment)		Comment text in round parenthesis
	Insert chamfer Z→X	G1 Zxxxx lyyyy		G1 Zxxxx B-yyy		Sign of yyyy specifies direction with Fanuc
	Insert chamfer X→Z	G1 Xxxxx Kyyyy		G1 Xxxxx B-yyy		Direction is determined by next block with Siemens
	Insert radius Z→X	G1 Zxxxx Ryyyy		G1 Zxxxx Byyyy		Siemens: B- is chamfer: B is radius
	Insert radius X→Z	G1 Xxxxx Ryyyy		G1 Xxxxx Byyyy		In systems without a 3rd axis (C axis) (parameter no. 29 ADRC=0), chamfer is specified via C and not via I,K.
	2-point definition	X2 A		A X2		
		Z2 A		A Z2		
	3-point definition	A1; X3 Z3 A2		A1 A2 X3 Z3		
	3-point definition + radius	X2 Z2 ,R1; X3 Z3		X2 Z2 B1; X3 Z3		In systems without a 3rd axis (C axis) (parameter no. 29 ADRC=0), the comma before C and R can be omitted.
		A1 ,R1; X3 Z3 A2		A1 A2 X3 Z3 B1		
	3-point definition + chamfer	X2 Z2 ,C1; X3 Z3		X2 Z2 B-1; X3 Z3		
		A1 ,C1; X3 Z3 A2		A1 A2 X3 Z3 B-1		
	3-point definition + radius + radius	X2 Z2 ,R1; X3 Z3 ,R2; X4 Z4		X2 Z2 B1; X3 Z3 B2; X4 Z4		
		A1 ,R1; X3 Z3 A2 ,R2; X4 Z4		A1 A2 X3 Z3 B1 B2; X4 Z4		
	3-point definition + chamfer + chamfer	X2 Z2 ,C1; X3 Z3 ,C2; X4 Z4		X2 Z2 B-1; X3 Z3 B-2; X4 Z4		
		A1 ,C1; X3 Z3 A2 ,C2; X4 Z4		A1 A2 X3 Z3 B-1 B-2; X4 Z4		
	3-point definition + radius + chamfer	X2 Z2 ,R1; X3 Z3 ,C2; X4 Z4		X2 Z2 B1; X3 Z3 B-2; X4 Z4		
		A1 ,R1; X3 Z3 A2 ,C2; X4 Z4		A1 A2 X3 Z3 B1 B-2; X4 Z4		
	3-point definition + chamfer + radius	X2 Z2 ,C1; X3 Z3 ,R2; X4 Z4		X2 Z2 B-1; X3 Z3 B2; X4 Z4		
		A1 ,C1; X3 Z3 A2 ,R2; X4 Z4		A1 A2 X3 Z3 B-1 B2; X4 Z4		
	Global variables 100 – 149 are not remanent 500 – 531 are re- manent	#xxx	no	Rxxx		
	Indirect addressing #100 =234 #234= 55 #9100 = 55	#9xxx	no	Not supported		
17–10	Macro instruction (G65)	G65 Hm P#i Q#j R#k				#i: Target variable #j: Variable or constant #k: Variable or constant
		G65 H01 P#i Q#j		Ri = Rj		Definition and substitution of variables #i = #j
		G65 H02 P#i Q#j R#k		Ri = Rj + Rk		Addition #i = #j + #k
		G65 H03 P#i Q#j R#k		Ri = Rj – Rk		Subtraction #i = #j – #k
		G65 H04 P#i Q#j R#k		Ri = Rj * Rk		Multiplication #i = #j * #k

## 5.1.9 FANUC program converter

Page	Function	FANUC 0T G CODE A / B / C G functions listed singly below are valid for all 3 axes.	Only these words allow- ed in block	SINUMERIK 840C	G group, modal, basic set- ting	Comment
		G65 H05 P#i Q#j R#k		Ri = Rj / Rk		Division #i = #j / #k
		G65 H11 P#i Q#j R#k		@655 Ri Rj Rk		Logic sum #i = #j .OR. #k
		G65 H12 P#i Q#j R#k		@657 Ri Rj Rk		Logic product #i = #j .AND. #k
		G65 H13 P#i Q#j R#k		@656 Ri Rj Rk		Exclusive OR #i = #j .XOR. #k
		G65 H21 P#i Q#j		@613 Ri Rj		Square root #i = square root (#j)
		G65 H22 P#i Q#j		@610 Ri Rj		Absolute value #i =  #j
		G65 H23 P#i Q#j R#k		Ri=Rj/Rk @622 Ri Ri=-Ri*Rk+Rj		Division remainder #i = #j MOD #k
		G65 H24 P#i Q#j		No corresponding function		Conversion of BCD to binary #i = BIN(#j)
		G65 H25 P#i Q#j		No corresponding function		Conversion of binary to BCD #i = BCD(#j)
		G65 H26 P#i Q#j R#k		Ri = Ri * Rj / Rk		Combined multiplication / division #i = (#i * #j) : #k
		G65 H27 P#i Q#j R#k		@614 Ri Rj Rk		Combined square root 1 #i = sqrt (sqr(#j) + sqr(#k))
		G65 H28 P#i Q#j R#k		No corresponding function		Combined square root 2 #i = sqrt (sqr(#j) - sqr(#k))
		G65 H31 P#i Q#j R#k		@630 Ri Rk Ri = Ri * Rj		Sine #i = #j * SIN (#k)
		G65 H32 P#i Q#j R#k		@631 Ri Rk Ri = Ri * Rj		Cosine #i = #j * COS (#k)
		G65 H33 P#i Q#j R#k		@632 Ri Rk Ri = Ri * Rj		Tangent #i = #j * TAN (#k)
		G65 H34 P#i Q#j R#k		@637 Ri Rj Rk		Arc tangent #i = ATAN (#j / #k)
		G65 H80 Pn		@100 Kn		Unconditional branch
		G65 H81 Pn Q#j R#k		@122 Rj Rk Kn		Conditional branch #j .EQ. #k => Pn
		G65 H82 Pn Q#j R#k		@121 Rj Rk Kn		Conditional branch #j .NE. #k => Pn
		G65 H83 Pn Q#j R#k		@126 Rj Rk Kn		Conditional branch #j .GT. #k => Pn
		G65 H84 Pn Q#j R#k		@124 Rj Rk Kn		Conditional branch #j .LT. #k => Pn
		G65 H85 Pn Q#j R#k		@125 Rj Rk Kn		Conditional branch #j .GE. #k => Pn
		G65 H86 Pn Q#j R#k		@123 Rj Rk Kn		Conditional branch #j .LE. #k => Pn
	The following addresses can be typed with a decimal point: X Y Z C U V W H I J K R A F The mode of input can be parameterized (page 9-3).					

*Example*

```

N10 G21
N20 M42
N30 G92 S4000
N40 G00 G95 G97 G40 X400. Z200. T0100 S1275 M03
N50 X45. Z6.0 T0100 M08
N60 G76 P021060 Q50 R0.025
N70 G76 X37.54 Z-40. R0. P1230 Q500 F2.
N80 G00 X400.Z200. T0100 M09
N90 M30

```

**Corresponding data (MD, SD)***Machine data*

Machine data 5007, bit 5

The mode of interpretation for programmed values, i.e. absolute (G90) or incremental (G91), is always defined for each separate block (up to LF) in a part program.

The bit 5=1 setting offers the option of defining the mode of interpretation, i.e. absolute (G90) or incremental (G91), before every programmed value of an axis (for interpolation parameters and angles, see NC–MD 5003, bits 4 and 5). Values are also calculated either absolutely or additively with respect to programmed zero offsets G58/G59 as a function of bit 5.

**Note**

If the MD bit is not set, the last function programmed in the block is applied.

## 5.2 Programming with the ASCII editor in the hard disk data management

Unlike modifying part programs in the NCK memory, you can perform the following with the aid of the softkey functions NEW, JOB LIST ...

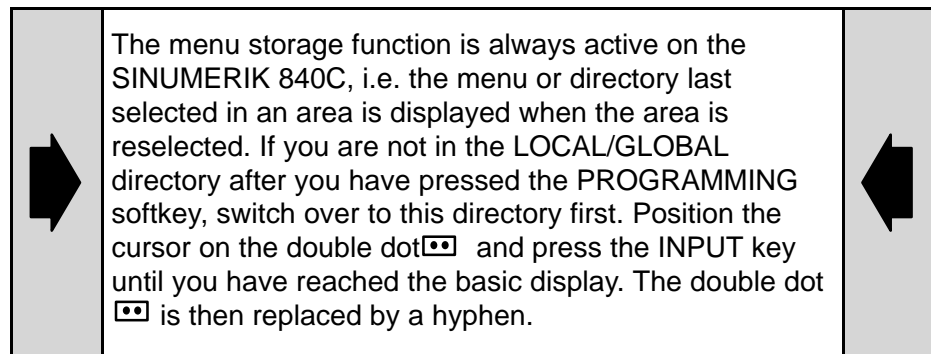
- Create files and workpieces and
- Modify the files assigned to the workpieces.

The workpieces or files created or modified here are stored directly on the hard disk.

## 5.2.1 Creating a workpiece

### Precondition

The PROGRAMMING area has been selected in the menu bar and you are in the LOCAL/GLOBAL directory.



A workpiece is a directory. All files required for machining of a workpiece can be stored under a workpiece:

- Main programs MPF ...
- Subroutines SPF ...
- R parameters RPA ...
- Setting data SEA ...
- Cycle setting data SEA4 ...
- Zero offsets ZOA ...
- Tool offsets TOA ...
- Work schedule (with WOP or simulation only)
- Tool list (with WOP or simulation only)
- IKA number IKA1 (creation makes little sense here)
- Curve pointer IKA2 (creation makes little sense here)
- Curve points IKA3 (creation makes little sense here)

You can store your workpiece either in the

- GLOBAL or
- LOCAL

directory. This classification is merely a suggestion for the management of your workpiece programs, but you need not follow this. You can, for example, organize your main programs and subroutines as described below:

### GLOBAL

General main programs and subroutines (e.g. cycles which you have generated).

### LOCAL

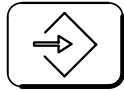
Main programs and subroutines which apply specially for your workpiece and which you can edit at any time, i.e. are not of wider significance.

### Operating sequence



Select the desired directory, LOCAL or GLOBAL with the cursor keys.

## 5.2.1 Creating a workpiece



Press the INPUT key.



Press the NEW softkey. You are requested to enter a name. Enter the name for your workpiece using the alphanumeric keyboard.

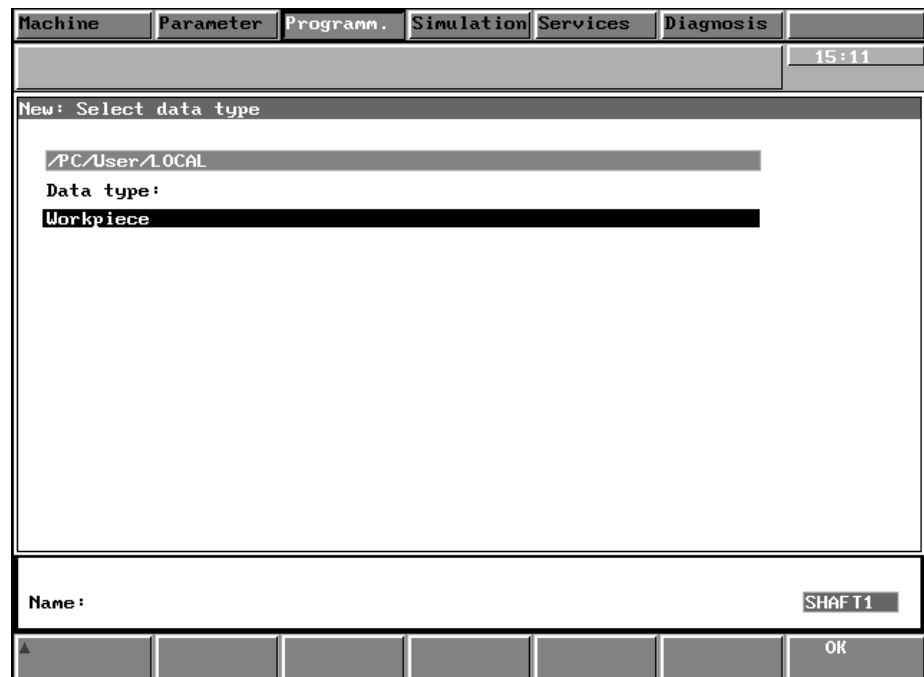
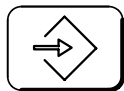
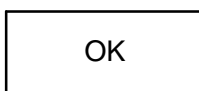


Fig. 5.11 Creating a workpiece



and conclude with the INPUT key.



Press the OK softkey to transfer the name into the directory.

*Note*

By entering the workpiece name (e.g. SHAFT1) you have merely created a directory for the type workpiece. Now you can assign job lists, part programs, R parameters, setting data, zero offsets and tool offsets to the workpiece (shaft1) and you can edit correspondingly. Furthermore you can add a comments file to the workpiece. With the help of the ASCII editor you can enter information to the workpiece in plain text and store it.

## 5.2.2 Creating a part program file

After you have opened a workpiece with the NEW softkey function, the following image is displayed:

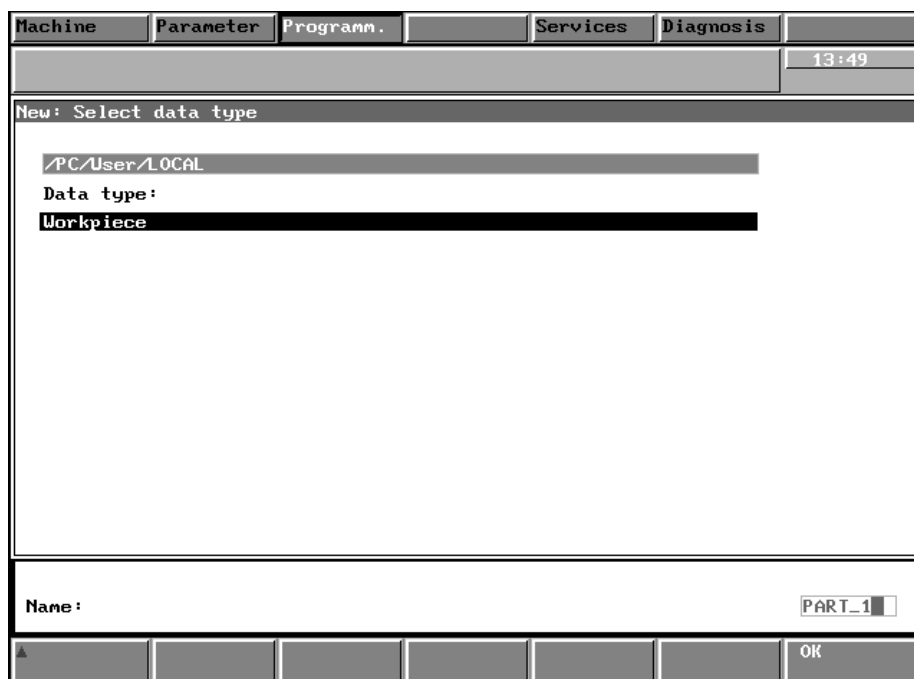
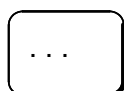
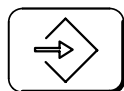


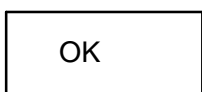
Fig. 5.12 Selecting the file types for editing in the PROGRAMMING area



The data type “part program” is preselected; enter the file name with the alphanumeric keyboard in the input screenform and



acknowledge with the INPUT key.



Store the file in the directory by pressing the OK softkey. The ASCII editor is displayed on the CRT. The part program can be edited immediately (see Section “Description of the ASCII editor”).

If a part program is to be created and edited in an already existing workpiece you

- Select the directory of the files and the workpiece using the cursor keys and the INPUT key and
- press the NEW softkey.

Perform the other operations as described above.

## 5.2.2 Creating a part program file

**Note**

When you enter the filename, the operating system automatically creates the type identifiers (see Section 1.4):

Main programs	mpf ...
Subroutines, cycles	sfp ...
R parameters	rpa ...
Setting data	sea ..., sea4 ...
Zero offsets	zoa...
Tool offsets	toa ...
IKA number	ika1
Curve pointer	ika2
Curve points	ika3

You must use the name MPF or SPF for main programs and subroutines respectively. You can enter any name in all other cases.

The worksheet and tool list files are not loaded into memory when the workpiece is loaded.

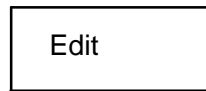
This data is required for a

- Simulation and
- Worksheet display

Select a program that already exists as follows:

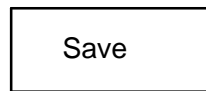


Position the cursor on the required program (e.g. MPF3) and



press the EDIT PC softkey.

A working field is displayed in which you can enter your part program via the keyboard. Operation of the ASCII editor is described in more detail in the "Description of the ASCII editor" section.



After the program has been entered press the SAVE softkey and the edited program is stored on the hard disk.



If you do **not** save and press the RECALL key instead, a warning message is output: "File has been edited. Lose the changes?".



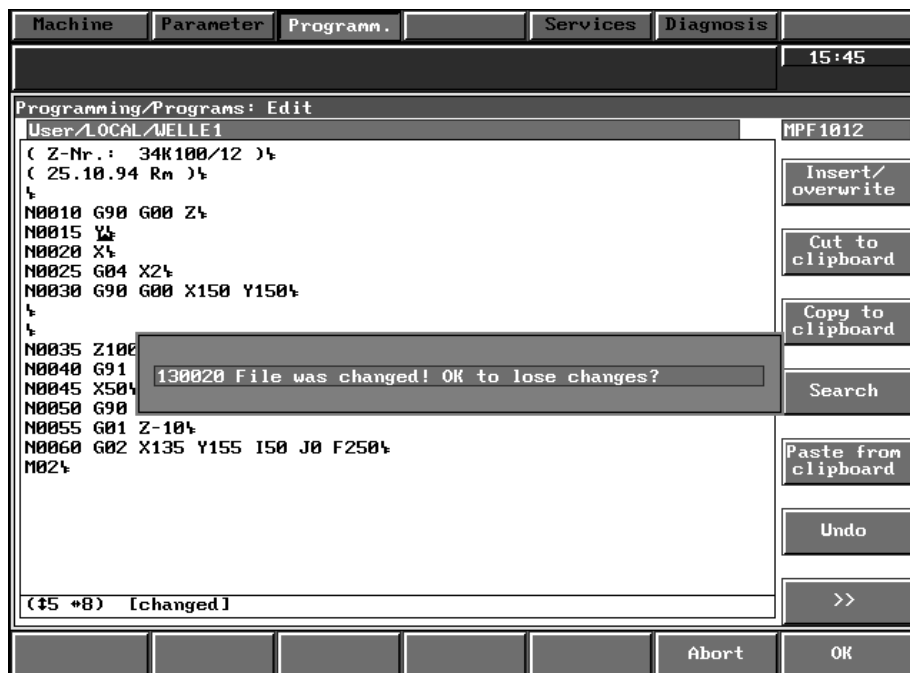
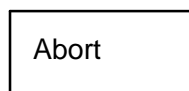


Fig. 5.13 Edit MMC, abort with RECALL

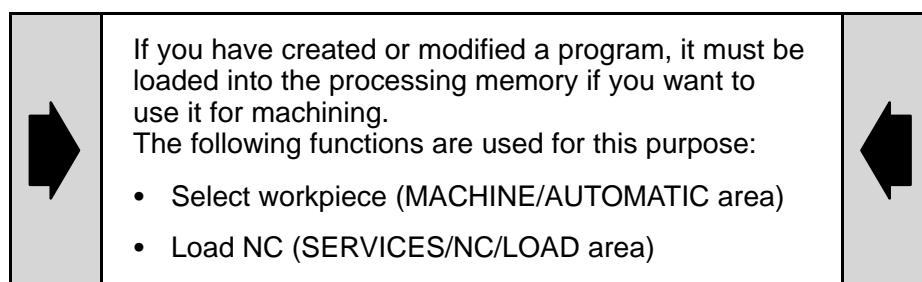
You can now select one of the two softkeys ABORT or OK.



Press the softkey OK to exit the edit mode and the changes you have made are **lost**.



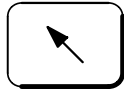
If you press the ABORT softkey, you remain in the Edit mode. You can carry out further modifications or store the file.



Press the RECALL key to return to the directory.

**Note**

Proceed as follows when editing R parameters, setting data, tool offsets and zero offsets:



After pressing the NEW softkey switch to the “Data type” window.



Select the desired data type.



Switch to the input screenform and enter the name as described. After pressing the INPUT and OK softkeys the ASCII editor is displayed. The remaining operating sequences are as described above.

**IKA data**



The abbreviation IKA stands for interpolation and compensation with tables. On the SINUMERIK 840C this function can be used in relation to machines or workpieces.



1. IKA data **defining a machine**

The IKA function is used for compensation of machine-dependent error curves caused by mechanical interference, temperature etc. The data are entered by the machine manufacturer and are stored in the diagnosis area under password protection.

2. IKA data **defining the workpiece**

This function is used to define the contours of a workpiece from the IKA data. Up to 65535 geometry interpolation points can be defined in a table. Up to 32 IKA relations (IKA data blocks) are supported. You can use this function to create complicated contours from a burst of geometry points.

	The IKA function must be completed by the machine manufacturer. The control also features a flexible memory configuration (the number of geometrical interpolation points is variable) that must be configured. Please refer to the machine manufacturer's documentation.	
---	---	---

	IKA data defining machine and workpieces are managed in the same memory area, i.e. if you use IKA data defining a workpiece, the machine-related IKA data in the same memory are overwritten. Please refer to the machine-specific documentation to determine which areas are used for error compensation and which are used for the workpiece functions.	
---	---	---

## 5.2.3 Creating job lists

### 5.2.3.1 General

A job list (load list) can be created for each workpiece to be machined. A load list contains instructions for initializing execution of part programs. A load list contains not only the main programs and subroutines required for machining of the workpiece, but also assignment of all relevant data, such as tool offset (ZOA), R parameters (RPA) and setting data (SEA, SEA4). The job list is executed if the workpiece has been selected in the machine area (AUTOMATIC basic display). If a workpiece is to be loaded via the MACHINE area, the job list must be generated in a workpiece in the LOCAL directory.

If a new workpiece is created, the control generates a standard job list in the form of a file in this workpiece. This standard job list contains the job list syntax (as a comment) as well as a `LOAD.*` command which loads all files of this workpiece. The job list syntax is described in more detail in the "Syntax description for job lists" section.

### 5.2.3.2 Creating the JOB LIST file

If no job list exists for the workpiece, you can create one following the operating sequences described below.

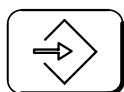
#### *Preconditions*

- The PROGRAMMING area has been selected.
- The workpiece exists to which the job list is to be assigned.

#### *Operating sequence*



In the workpiece directory, select the workpiece with the cursor keys and



press the INPUT key.



Press the NEW softkey; the following display appears:

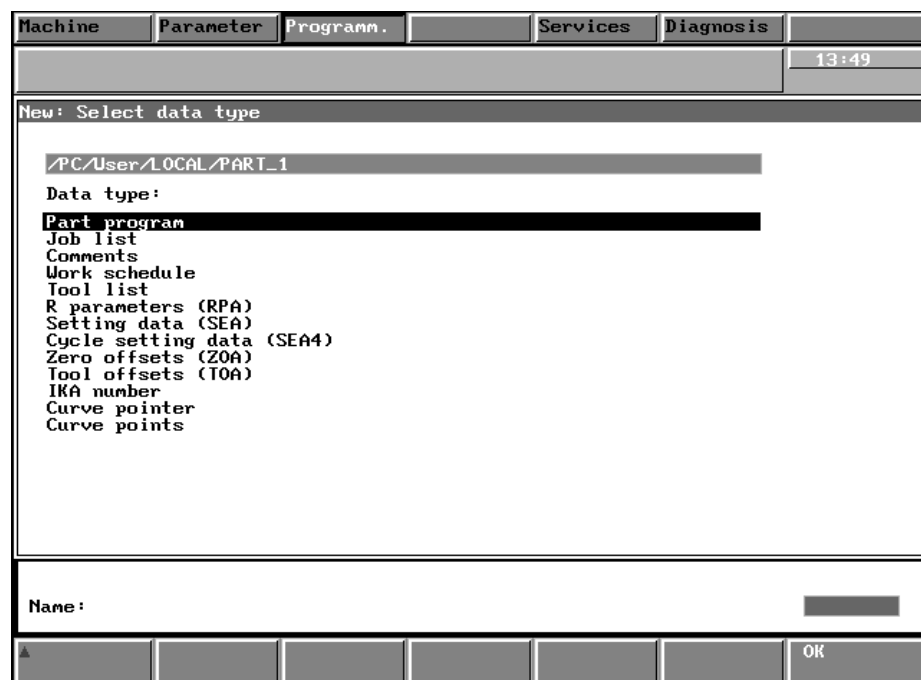


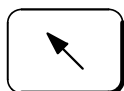
Fig. 5.14 Selecting the file types for editing in the PROGRAMMING area



Press the window switchover key to switch to the “Data type” window.  
The active window is marked (yellow frame).



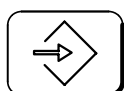
Select the data type “Job list” with the cursor keys.



Switch to the input screenform and



enter the name with the alphanumeric keyboard.



Accept the name by pressing the INPUT key and the





OK softkey.

The standard job list is displayed in the ASCII editor.

**Note**

A standard job list is created containing the permissible command set as a comment. The standard job list includes the command LOAD. \* which causes all data for the workpiece to be loaded in such a way as though there were no job list.

It can then be adapted by means of the EDIT PC and JOB LIST functions.

	The STANDARD job list can be configured by the machine tool manufacturer. In this case this description does not apply. Read the machine tool manufacturer's documentation.	
---	---	---


Machine	Parameter	Programm.	Services	Diagnosis	
					15:46
<b>Programming</b>					
User/LOCAL/WELLE1					HD memory 60KB
Name	Type	Length	Date		
..					
JOB	jobl	1898	09-22-1995	11:04:38	
MPF1	parp	294	09-22-1995	11:04:40	
MPF1012	parp	237	11-03-1995	14:20:54	
MPF11	parp	334	09-22-1995	11:04:38	
MPF22	parp	180	09-22-1995	11:04:38	
MPF23	parp	110	09-22-1995	11:04:38	
MPF4711	parp	103	09-22-1995	11:04:40	
RPA1	rpa	115	09-22-1995	11:04:40	
SEA	sea	13411	09-22-1995	11:04:40	
TOA1	toa	738	09-22-1995	11:04:40	
<div> <div>Edit PC double</div> <div>Edit</div> <div>New</div> <div>Copy to clipboard</div> <div>Paste from clipboard</div> <div>Delete</div> <div>Program. WOP</div> </div>					

Fig. 5.15 Directory of the files of a workpiece with job list

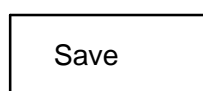
There are two possibilities for editing the job list:

1. By pressing the EDIT softkey
2. By pressing the JOB LIST softkey.

**Note**

To return from a directory to the directory previously selected, position the cursor on the double dot  and press the INPUT key.

If you do not wish to edit the job list press the RECALL key to leave the ASCII editor.



Press the SAVE softkey to store the edited job list on the hard disk.

### 5.2.3.3 Editing job lists

#### Operating sequence



In the workpiece directory in which you want to generate a job list, you position the cursor on the previously opened file of the JOB LIST type and



press the EDIT softkey.

The screen now displays the entire command set that you can use for creating a job list.

A load list comprises:

1. Clear files for NCK memory (CLEAR ...)
2. Load instructions (LOAD ...)
3. Start preparation (SELECT ...).

This sequence is mandatory.

Every area begins with comments (...).

These are followed by the actual command to which the operator must add path and program names. For reasons of clarity, you should only edit in these command lines.

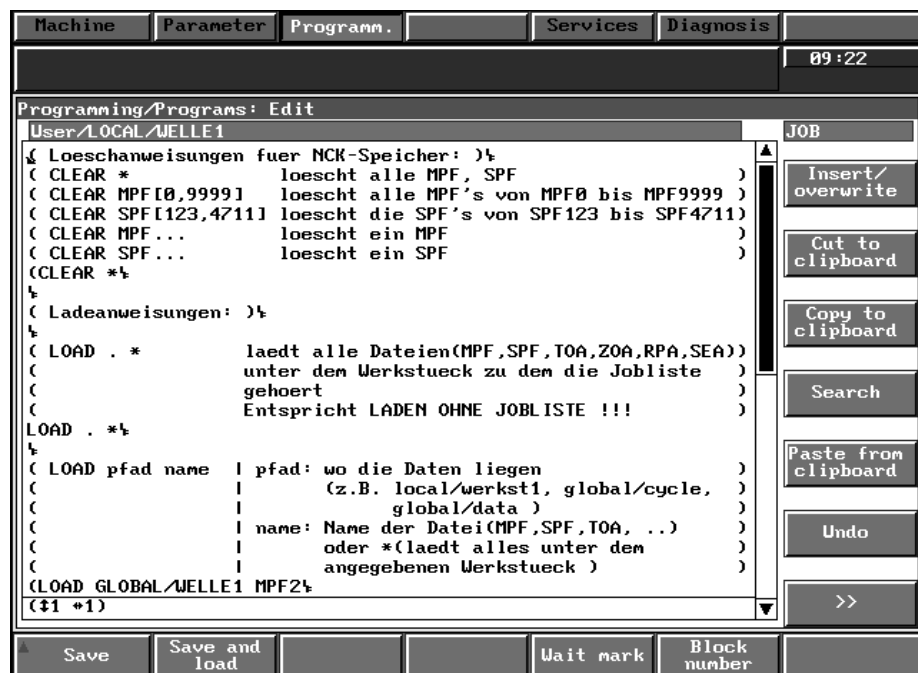


Fig. 5.16 ASCII editor for the "Creation of job lists" function via the EDIT softkey

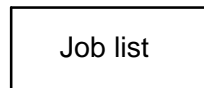
### 5.2.3.4 Creating job lists

**Precondition** The file directory of a workpiece has been selected.

**Operating sequence**



Press the ETC key to extend the softkey list in the basic display of the PROGRAMMING area.



Press this softkey and the following display appears:

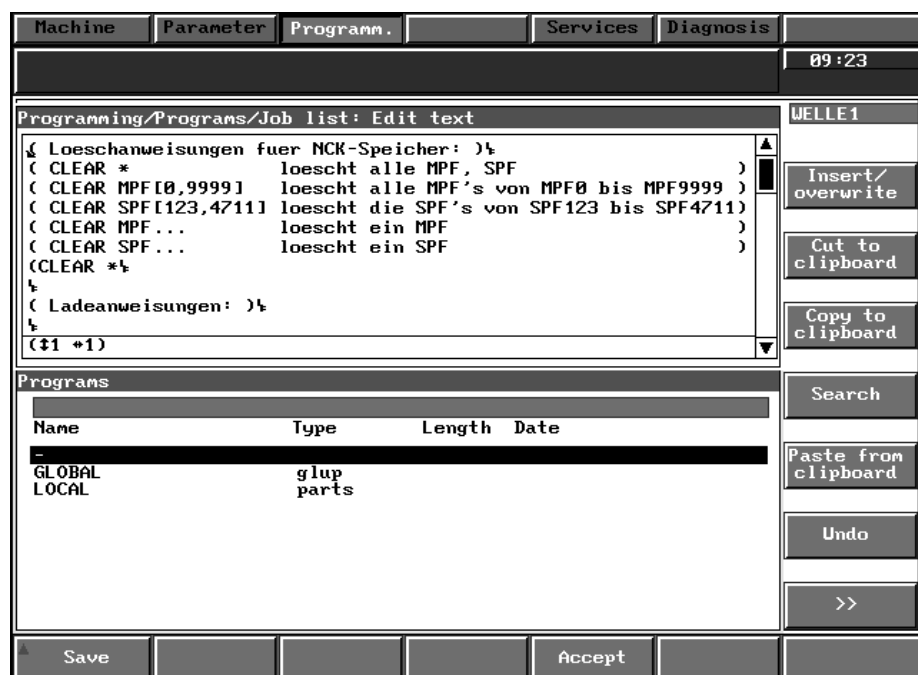
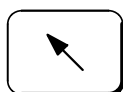


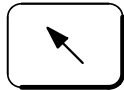
Fig. 5.17 Job list function via job list softkey



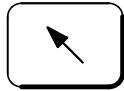
With the window switchover key you can switch between the two windows. The active window is always marked.

- In the top window, the standard job list from the EDIT area is displayed. You can page in this window with the paging keys.
- Please refer to the section “Syntax description for the job list” for syntax.
- The lower window is a display window. Here you can only read the local and global program lists but you cannot edit.
- The editor is described in the section “Description of the ASCII editor”.

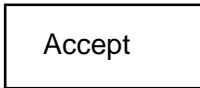
With the softkey ACCEPT you can enter paths from the lower display window directly in the job list.



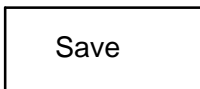
Switch to the upper window, if required. Move the cursor to the position in the file where the name is to be entered, e.g.  
LOAD ▲ ....



Switch to the lower window and page with the cursor keys and the INPUT key through the directories until you have reached the desired part program file.



Press the ACCEPT softkey. In the job list the current path name is entered at the cursor position, e.g.  
LOAD ▲ LOCAL/WELLE 1 MPF1



Press the SAVE softkey after having completed editing and the file is transferred to the hard disk.

### 5.2.3.5 Syntax description for the job lists

The syntax of a job list comprises three instructions:

- Clear file CLEAR
- Load instruction LOAD
- Select instruction SELECT

This sequence must be kept to.

#### *Comment*

All terms in brackets are comments and are not taken into account when processing the job list.

#### *Clear file*

With the CLEAR command, part programs are deleted from the NCK part program memory.

#### *Command syntax*

CLEAR □ name L<sub>F</sub>

Use the following command set for "name":

MPFn (one main program)

SPFn (one subroutine)

MPF[n,m] (part programs MPFn...MPFm)

SPF[n,m] (subroutines SPFn...SPFm)

\* means that all part programs are deleted from the part program memory (in the NCK).

#### *Examples*

Deletion of the MPF7 part program:

CLEAR □ MPF7

Deleting the SPF 1 to 999:

CLEAR □ SPF[1, 999]

Deletion of all MPF,SPF:

CLEAR □ \*

**Caution!** The syntax is mandatory. Observe the position of the blanks □.



<i>Load instructions</i>	By means of the LOAD instruction one or several files from the local or global workpieces are loaded into the part program memory of the NCK.
<i>Command syntax</i>	<p>LOAD _pfad _name L<sub>F</sub></p> <p>“path” comprises:</p> <ul style="list-style-type: none"> <li>• Path name and</li> <li>• Workpiece name.</li> </ul> <p>Use the following command set for “path”:</p> <ul style="list-style-type: none"> <li>• LOCAL/workpiece name</li> <li>• GLOBAL/workpiece name</li> </ul> <p>. (full stop) means that the data of the current workpiece from the job list selected for processing is transferred.</p> <p>Use the following command set for “name”:</p> <ul style="list-style-type: none"> <li>• MPFn (one main program)</li> <li>• SPFn (one subroutine)</li> <li>• MPF[n,m] (part programs MPFn...MPFm)</li> <li>• SPF[n,m] (subroutines SPFn...SPFm)</li> </ul> <p>* means that all part programs of a workpiece are loaded into the part program memory (NCK).</p> <p>Only files designed for use in the NCK can be loaded with a load list: MPF, SPF, TOA, ZOA, RPA, SEA, SEA4. Identifiers MPF and SPF are mandatory. The channel number and the area number must be specified if this can not be concluded from the last character of the file name (TOA1).</p>
<i>Examples</i>	<p>Load all files of workpiece SHAFT1 from the LOCAL directory:</p> <p>LOAD _LOCAL/WELLE1 _ *</p> <p>Load file WKZK (type toa) of workpiece WELLE1:</p> <p>LOAD _LOCAL/WELLE1 _ WKZK _ CH=1</p> <p>Load files SPF7 to SPF19 of workpiece WKST1 from the LOCAL directory:</p> <p>LOAD _LOCAL/WKST1 _ SPF[7,19]</p> <p>Load file SPF1 of workpiece ZYC1 from the global directory:</p> <p>LOAD _GLOBAL/ZYC1 _ SPF1</p> <p>Load all files of the current workpiece:</p> <p>LOAD _._ *</p> <p><b>Caution!</b> The syntax is mandatory. Observe the position of the blanks _.</p>

➔	<p>Avoid the following combinations:</p> <p>LOAD GLOBAL / &lt;dir.&gt; MPFx or LOAD GLOBAL / &lt;dir.&gt; SPFx, programs MPFx or SPFx in directory &lt;dir.&gt; under GLOBAL also exist in another (different) workpiece under LOCAL. See also the explanation below.</p>	➔
---	---	---

*Explanation*

The only real use of command LOAD GLOBAL/... is for loading global programs, e.g. GLOBAL/Loader/MPF100. No workpiece under LOCAL should then contain another MPF100 (i.e. not, for example, LOCAL/Welle1/MPF100), as otherwise one program will overwrite the other (depending on the order in the job list).

The only real use of programs with the same name is for backing up workpieces under GLOBAL, e.g. GLOBAL/Welle1. In this case, command LOAD GLOBAL/Welle1/MPF100, for example, should be avoided because after CORRECT PROGRAM and SAVE TO DISK any changes are stored under LOCAL but the old version is called up again the next time the workpiece is loaded.

*Remedy*

Use LOAD.\* for workpiece programs, only use LOAD GLOBAL/... for global programs (of a different name!).

*Loading instruction for cycles*

You can use the LOADCYC instruction to load the SIEMENS and user cycles with the name SPF. The instruction only takes effect in the job list of a standard workpiece. The instruction is ignored in other workpieces.

*Command syntax*

LOADCYC    pfad    name L<sub>F</sub>

path:           USER or STANDARD

name:           Cycle name

                  SPF[n, m]       (cycles SPF<sub>n</sub>...SPF<sub>m</sub>)

                  SPF n           (one cycle)

*Examples*

Load all Siemens cycles:  
LOADCYC    STANDARD    SPF[0, 9999] or  
LOADCYC    STANDARD   

Load all user cycles:  
LOADCYC    USER    SPF[0, 9999] or  
LOADCYC    USER    \*

Load user cycle SPF5:  
LOADCYC    USER    SPF5

**Caution!** You must observe the exact syntax. Please note the position of the space character   .

*Selection instruction* With the SELECT instruction a part program with the identifier MPF is selected for execution in a specific channel in the NCK.

*Command syntax* SELECT \_name\_ CH=nr (DISK) L\_F

name: Part program name

CH=nr: Channel number

DISK: Execution from hard disk (need be specified only if function required)

Use the following command set for part program names:

MPFn (one main program).

The part program must first be transferred to the part program memory of the NCK by means of a load instruction.

*Examples* Select part program MPF7 in channel 1:

SELECT \_MPF7\_ CH = 1 L\_F

Select function "Execution of a part program" from the hard disk:

SELECT \_name\_ CH=nr DISK L\_F

Select function "Execution of part program MPF123" from the hard disk:

SELECT \_MPF123\_ CH=1 \_DISK L\_F

*Notes*

The program selected by means of the function "Execution from hard disk" must **not** be loaded with LOAD and must be stored under the workpiece belonging to the job list.

**Caution!** The syntax is mandatory. Observe the position of the blanks \_.

## 5.2.4 Shopfloor sheet

The shopfloor sheet is an overview of program data of a workpiece generated graphically (WOP). All technological information on the workpiece, which otherwise is to be found in the workpiece program, job list, etc. is displayed here.

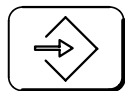
### Preconditions

- The PROGRAMMING area has been selected.
- You are in the LOCAL/GLOBAL selection display.

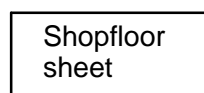
### Operating sequence



Page with the cursor keys and the input key to a workpiece directory.



Select a workpiece.

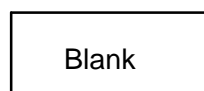


Press the SHOPFLOOR SHEET softkey; an overview of all existing part programs under the selected workpiece is displayed.

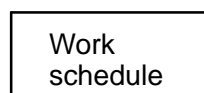
### Note

In the "Type" field, either "DIN" or "Graphic" will appear. This indicates whether the part programs in the workpiece directory have been created in DIN code or graphically.

The channel shown for MPFs, SPFs is taken from the job list.



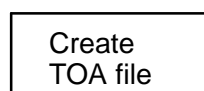
Press the following softkeys BLANK or



WORK SCHEDULE or



COMMENTS EDITOR or



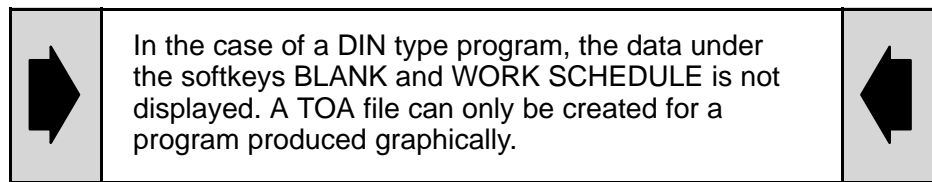
CREATE TOA FILE.

If you press the BLANK softkey, the data of the selected workpiece blank is displayed.

Pressing the WORK SCHEDULE softkey offers technological information on the selected workpiece.

Technological steps (such as solid milling, drilling) are displayed on the screen. You can select a machining step with the arrow keys and with

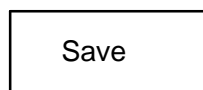
the MACHINING OPERATION softkey you can display the technology data (as for example tool, zero offset etc.).



If you press the COMMENTS EDITOR softkey, you are provided with the ASCII editor with which you can create a comments file to the workpiece displayed.

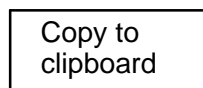
You can use the CREATE TOA FILE function to include tool data created with the graphic programming system (WOP) in a tool offset (TOA) file. This function applies exclusively to programs developed with the graphic programming system.

For a description of the editor functions please refer to the "Description of the ASCII editor" section.

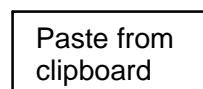


After you have completed work with the editor, press the SAVE softkey and the text is transferred to the hard disk.

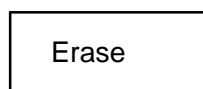
### 5.2.5 Copy, paste and delete files in the data management area of the hard disk



By means of these softkey functions you can copy workpieces and files from the hard disk data management and paste it from another position.



Use the INPUT key to page to the source file. If you press the COPY TO CLIPBOARD softkey, the selected file/directory is entered in a buffer. Then select a target directory. If you press the PASTE FROM CLIPBOARD softkey an input screenform is displayed in which you can enter a new name for the workpiece or the file. Conclude with the INPUT or the OK key. If the name is not to be changed simply press the OK key and the file/directory is pasted to the selected target directory.



With the cursor keys or the INPUT key select the workpiece or the file to be deleted. After you have pressed the softkey you are asked by an interactive form whether you really wish to delete. Press the OK softkey to confirm or the RECALL key to abandon.

## 5.2.6 Description of the ASCII editor

Edit

The following display is provided after you have called up the ASCII editor by pressing the EDIT softkey in directories LOCAL and GLOBAL:

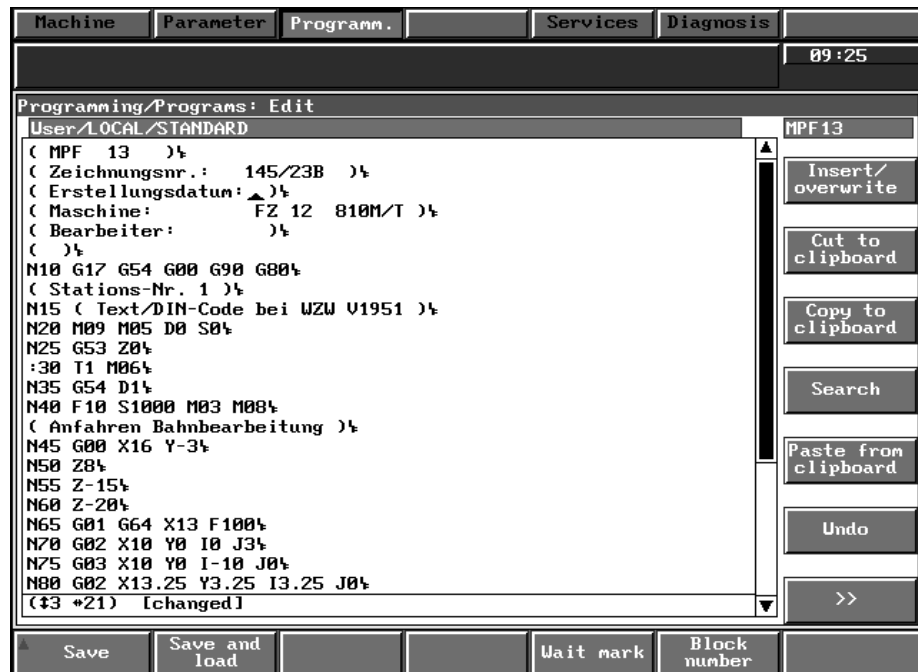
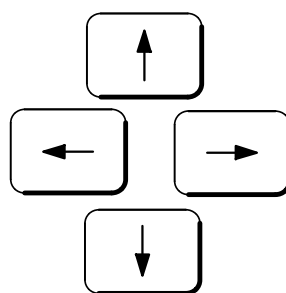


Fig. 5.18 ASCII editor in the programming area

**Note**

You can edit files up to 8 MB in size with the editor.

### 5.2.6.1 Editing with the ASCII editor



With the cursor keys you move the cursor in the text.

The current line *z* and column *s* are displayed using symbols  $\updownarrow z \leftrightarrow s$  below the text window (e.g.  $\updownarrow 10 \leftrightarrow 5$  = line 10, column 5).

This lower line is called the status line.

Status line  
[Changed]  
Read xx%  
Saved xx%  
Replace xx%  
Load

Other information displayed in the status line:

Displayed if the file has been changed.

These messages show the status for reading, saving and replacing a file in percent

File (MPF.. or SPF..) transferred to NCK (after softkey: SAVE AND LOAD)

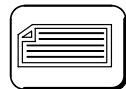


Move the cursor to the beginning/end of the line.

or



With these keys you page the screen display.

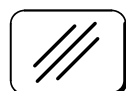


Move the cursor to the beginning/end of the file.

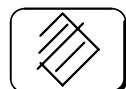
or



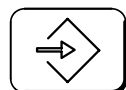
With this key you switch the selection mode on or off (see “Cut”, “Copy”, “Paste” functions).



You move the cursor with the cursor keys to the right of the term to be deleted. This key is used to delete character by character from right to left.



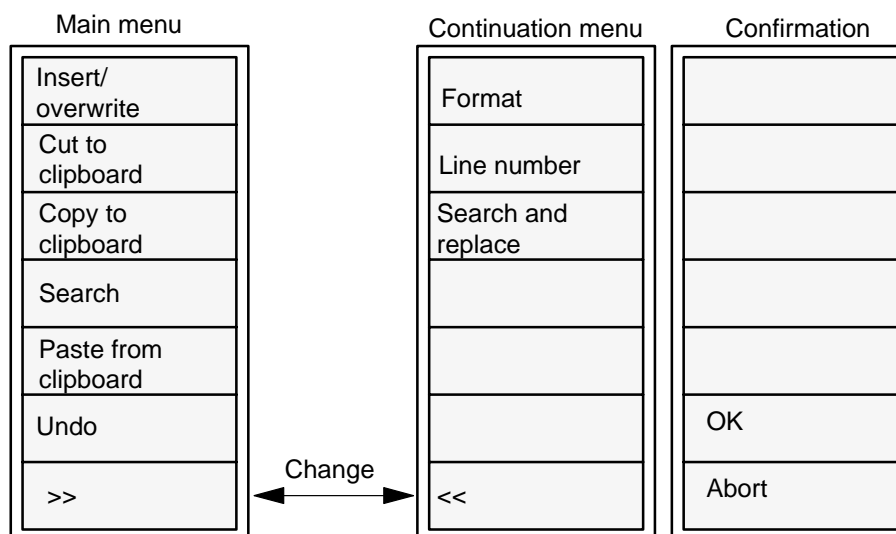
Insert mode: The character to the right of the cursor is deleted.  
Overwrite mode: The character under the cursor is deleted.



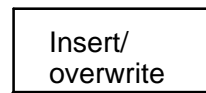
The INPUT key is used to conclude. LF is generated.



Use the vertical softkeys to select functions for text editing.

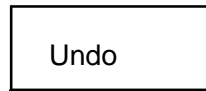
There is a main menu and a continuation menu.



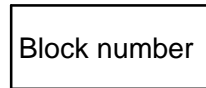
## 5.2.6 Description of the ASCII editor



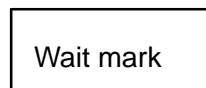
switches over between insert and overwrite mode  
(Cursor:  or )



Cancels the last function performed on the text



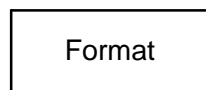
Inserts a block number at the cursor position



If you press the WAIT MARK softkey, the following screenform is displayed:

Type	<input type="text" value="m"/>		Number	<input type="text" value="1"/>
Chan.	<input type="text" value="Slide 1"/>	<input type="text" value="Slide 2"/>	<input type="text" value="Slide 3"/>	<input type="text" value="Loader"/>
	<input type="text" value="-"/>	<input type="text" value="-"/>		

The "Wait mark" function is described in the Programming Guide SIN 840C "Program coordination" section. The toggle fields for channel specifications are configurable by the machine tool manufacturer (entry in the file Operation/PROGRAM/BEDCONF).



Displays the form for format parameter specification

Format:			
Font:	<input type="text" value="small"/>	Block numbers:	<input type="text" value="no"/>
Foreground color:	<input type="text" value="red"/>	Steps:	<input type="text" value="5"/>
Background color:	<input type="text" value="white"/>	Start value:	<input type="text" value="5"/>

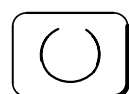
Font: medium/large/small

Foreground color: red/green/blue/yellow/violet/cyan/white/black

Background color: red/green/blue/yellow/violet/cyan/white/black

Block numbers: yes/no.

When you press the FORMAT softkey, a form is displayed offering various selection and input fields.



In the selection fields Font, Foreground color, Background color and Block numbers, you can page through the possibilities specified by the operating system.



You switch back the toggle field by pressing the key combination SHIFT + Selection key.



In the Steps input field, you can enter the intervals between the block numbers. In the Start value input field you enter the 1st required block number.

#### Example

Block number: yes  
 Steps: 5  
 Start value: 10  
 Execution: N10  
           N15  
           N20.

Conclude your inputs with the INPUT key. The entire form is stored and becomes effective when the OK softkey is pressed.

The softkey calls up the search key over form.

Search

Find

Enter the term to be found and conclude with INPUT. Press the OK softkey. The cursor is positioned on the required character string.

Displays form for defining the old and new character string.

Search and  
replace

Find:

Replace by:

First only

Enter the required character string in the input fields "Find" and "Replace by" and conclude with INPUT.

In the selection field you can select between:

- First only  
(the first required character encountered behind the cursor position is replaced),
- Cursor-file\_end  
(all occurrences of the required character string are replaced from the the current cursor position to the end of the file),
- File\_beg-cursor  
(all occurrences of the required character string are replaced up to the current cursor position),
- File\_beg-file\_end  
(all occurrences of the required character string are replaced from beginning of file to end of file).

You thus specify the range in which the term is to be replaced.

Replacement occurs after pressing the OK softkey.

From SW 6, another softkey bar is displayed providing another selection menu.

## 5.2.6 Description of the ASCII editor

*Notes*

- Blanks are considered as normal characters when finding and replacing.
- **Caution** when replacing: The ASCII editor is a pure word editor, i.e. no validity check is carried out:

*Example*

Find N10 → Replace by N11

Program before replacing / after replacing (to end of file)

N5	N5
N10	N11
•	•
•	•
•	•
N105	N115
•	•
•	•
•	•
M30	M30

You can insert a blank in order to restrict replacement, e.g.:

Find N10 (blank) Replace by N11.

Only the character string N10 is replaced.

Copies selected text into buffer and deletes it subsequently in the file.

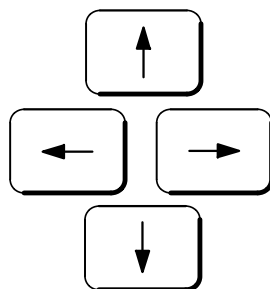
Cut to  
clipboard

Copy to  
clipboard

Paste from  
clipboard



You select the 1st character of the character string you want to cut or copy.



Now you select the required character string with the cursor keys. The selected text is highlighted.

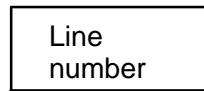
Press the CUT or COPY softkey, move the cursor to the location at which this character string is to be inserted and press the INSERT softkey.

For the CUT function you can use all known cursor functions (Shift+Cursor, Shift+Page keys). It is thus possible to select an entire text or a complete screen display for cutting with few operations.

**Note**

The CUT and COPY functions operate area-independent, i.e. you can copy a text from every ASCII editor field of an area into another area. The data is stored in a buffer.

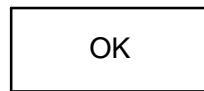
Press this key to cancel the selection of a text area.



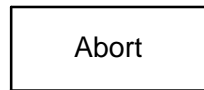
Switch line number on or off.

**Note**

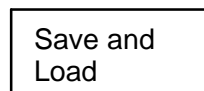
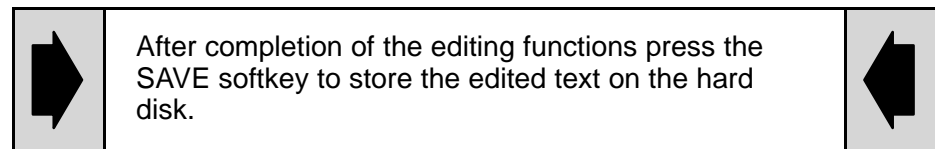
The line number is **not** identical to the block number. The line number is merely an editing aid and is not processed when cutting, copying etc.



Stores the input parameters and executes the function



Aborts the function.



If you press softkey SAVE AND LOAD instead you also load the current program in the NCK memory (RAM) which can then be tested immediately.

### 5.2.6.2 Input fields

The characters in the input fields are always inserted to the left of the cursor.

The color changes when selecting to the color "Edit cursor".

The following character set applies to the input fields:

Letters, digits, underscore.



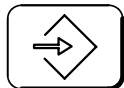
Cursor keys for positioning the cursor.



With the cursor keys, you move the cursor to the right of the character you wish to delete and press the CLEAR key. The character is deleted.



This key resets the input in the active input field.



The input is stored in the directory by pressing the INPUT key or

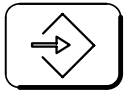


the cursor keys and the next control element is selected. If the part program has already been loaded in the NCK, data transfer must be enabled with OK after the message "130021 MPF exists" "Overwrite 2". With Abort, the part program is not transferred.

### 5.2.6.3 Selection fields (toggle fields)

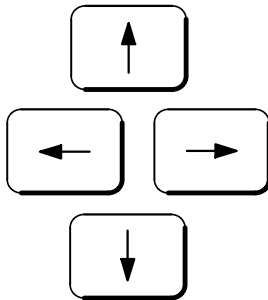
The text color of a selection field not selected corresponds to the color "Application window". The color changes on selection.

With the selection key you switch through the possible values.



Press the INPUT key to select the next field or

use the arrow keys.



### 5.2.6.4 Double ASCII editor

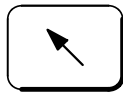
Edit  
PC double

The EDIT PC DOUBLE softkey is provided. If you press this softkey, the following image is displayed:

Machine	Parameter	Programm.	Simulation	Services	Diagnosis	
						15:40
File_1						
User/LOCAL/SHAFT1						
Name	Type	Length	Date			
SHAFT	comm	25	01-13-1994	15:16:54		
JOB	job1	1866	01-13-1994	15:11:48		
MPF13	parp	596	01-13-1994	15:13:50		
MPF14	parp	468	01-13-1994	15:14:06		
SPF103	parp	472	01-13-1994	15:14:22		
MPF4711	parp	352	01-13-1994	15:40:30		
WERKSTA	fainf	1998	01-13-1994	15:14:58		
File_2						
User/LOCAL/PART						
Name	Type	Length	Date			
KOMMENT	comm	160	12-09-1993	15:49:32		
JOB	job1	1505	12-09-1993	15:49:32		
MPF13	parp	596	12-09-1993	15:49:32		
MPF14	parp	412	01-13-1994	15:40:24		
SPF103	parp	472	12-09-1993	15:49:32		
MPF12	parp	4	01-27-1992	20:45:24		
Edit						

Fig. 5.19 Directories for the double ASCII editor function

The LOCAL/GLOBAL user directory is displayed twice.



Edit

You can switch between the two files with the window switchover key. The active window is marked (yellow frame). You select a file in **both** directories by pressing the cursor keys or the INPUT key.

Press the EDIT softkey. The double ASCII editor is displayed.

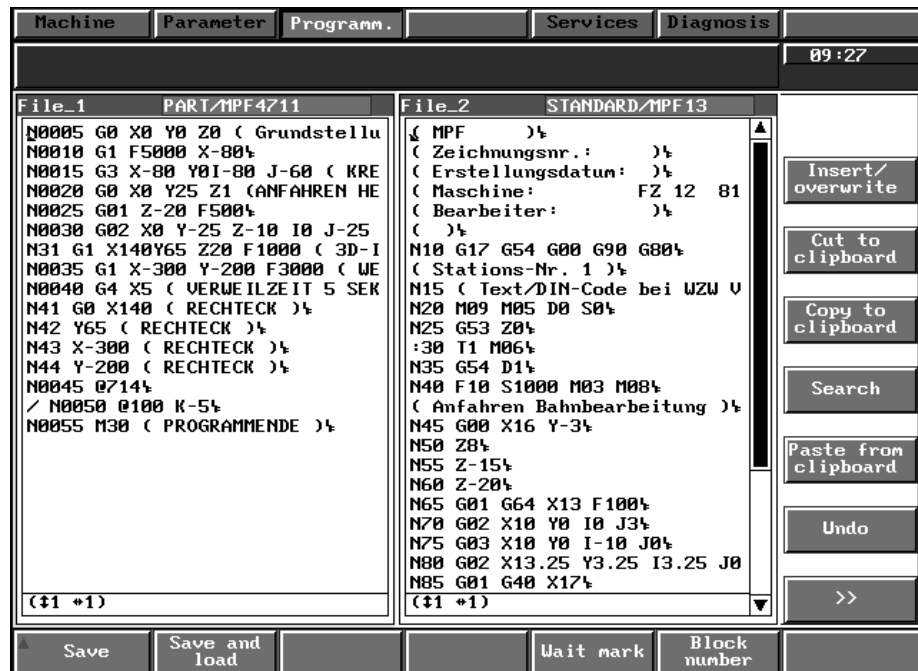


Fig. 5.20 Double ASCII editor



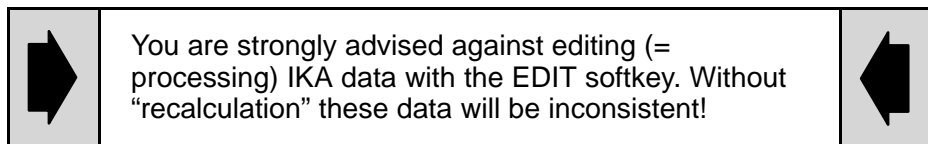
With this key you can switch between the two editors. The active window is marked. In both editors you can edit two files in parallel. This application is relevant for double-slide machines.

## 5.2.7 Viewing IKA data



View IKA data

IKA data can be viewed in the PROGRAMMING area. To do this, press softkey VIEW IKA DATA after you have selected file of "type" IKA..3 with the data selector.



*IKA data*

The abbreviation IKA stands for Interpolation and Compensation with Tables. This function can be used with reference to the machine or the workpiece.



*IKA data describing the machine*



This function is used to compensate for errors caused by mechanical factors.

- Temperature (expansion due to heat)
- Mechanical errors caused by axes themselves (e.g. sag)
- Leadscrew error (spindle machining tolerances)

These errors can be compensated for in the control by means of temperature compensation and compensation with tables (part of IKA). The parameters of the machine and the measuring system are measured by laser during installation and stored in the control to be used for calculating the compensation at a later date. The calculated compensation values are included in the position control loop so that the axes approach the desired machine setpoint position.

A means of compensation already exists for compensating leadscrew errors (see the Section entitled "Leadscrew error compensation") but which can only be used to a limited degree because of the rigid grid spacing and compensation value.

	The IKA function must be configured by the machine-tool manufacturer. Please consult the documentation supplied by the machine-tool manufacturer.	
---	---	---

	This function is available with Software Version 4.1 and higher.	
---	--	---

*Precondition*

The following is required for viewing IKA data under PROGRAM

- The IKA data are stored under one workpiece

*Limitation*

- No SEARCH function
- It is not possible to edit.

*Creating any contours*

With this IKA function it is possible to create complicated contours using many geometrical interpolation points. To do this, the values of the output variables are assigned to the input variables for a number of intermediate points. These value pairs are then stored as a table in the NC. A maximum of 65535 intermediate points (depending on the memory configuration; please consult the machine-tool manufacturer's documentation) are possible. As soon as the input variable reaches an intermediate point during part program processing, the corresponding output variable is output added to it. Input and output variables can be axis positions. The output variable (positional offset) is added to the programmed axis position in question as an offset.

The axis for the output variable must not be a fictitious axis.

*General*

The following options can be active simultaneously:

- Temperature compensation to compensate for heat expansion
- Interpolation and compensation with tables (IKA), e.g. for leadscrew error compensation, beam sag compensation for telescopic axes, creating curve profiles etc.

Both options are fully independent of each other. The individual results are added together axis-specifically to give the absolute traversing path of an axis. If several interpolations or compensations have to be calculated, they all take a common value as the input variable.

These functions are active in all modes after reference point approach of the axes involved. If an axis that has already been referenced is rereferenced, these functions are disabled when the cam is reached and until reference point approach has been completed.

The resulting traversing path for the functions involved is displayed in the service display in position control resolution.

The resulting path to be traversed and compensation value (sum of temperature compensation and all IKAs) is calculated as an absolute positional error and is included in the actual value path.

END OF SECTION





## 6 Services

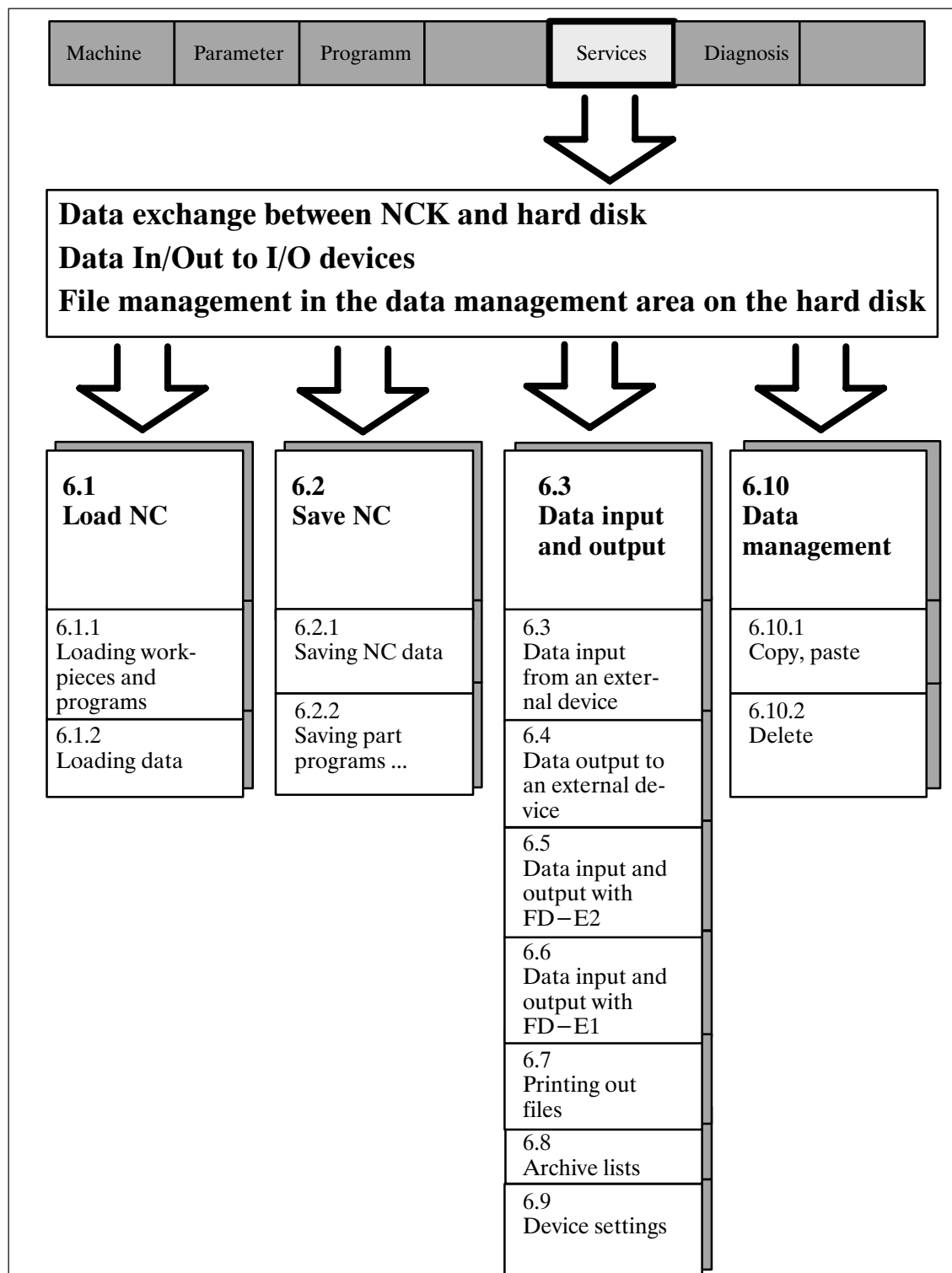


Fig. 6.1 Services

Services

Press this softkey in the area menu bar to select the SERVICES area.

Machine	Parameter	Programm.	Services	Diagnosis	14:14	
Service						
NC	Transfer data from/to NC Display information from the shopfloor sheet					
Computer link	Transfer data from/to host computer transfer					
Devices	Device settings					
Print serial	Print PC data					
Data input	Read-in PC data					
Data output	Output PC data Process archive lists					
Data management	Copy and delete PC data					
Interface	Device	Status				
Serial 1		inactiv				
NC	Computer link	Devices	Print serial	Data input	Data output	Data management

Fig. 6.2 Basic display "Services"

In the SERVICES area you can store NCK data (MPF, SPF, TOA, ZOA, RPA, SEA) on the hard disk and transfer it from the hard disk into the NCK memory. Furthermore, you can organize and start data exchange with the peripheral devices.

You can use the data management functions to copy, insert and delete files on the hard disk.

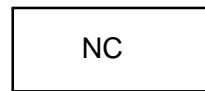
## 6.1 LOAD NC

In the SERVICES/LOAD NC area, you can transfer NCK data from the hard disk into the NCK memory.

### Precondition

You are in the SERVICES area. The basic display is active.

### Operating sequence



Press the NC softkey and the following display appears:



Fig. 6.3 Save/load NC



Press the HOME key to switch to the bottom window (NC: User). The active window is highlighted by a yellow frame.

Move the cursor keys and press the INPUT key in the selected directory to select the subdirectory containing the data you wish to load in the NCK memory.

Directory NC contains the data that are loaded into the working memory when the control is started up. This directory should only contain IKA and GIA data. Other data, such as SEA, RPA, etc. should be assigned to the workpieces.

The workpieces and the associated data are contained in the LOCAL and GLOBAL directories (see PROGRAMMING section). You can load individual workpieces or files in the NCK memory from the local or global directories.

**Note**

Only workpieces or NC files can be loaded. If you try to load a directory (e.g. LOCAL) a dialog box containing the following message is displayed:

110109 Only workpieces/NCK data can be loaded

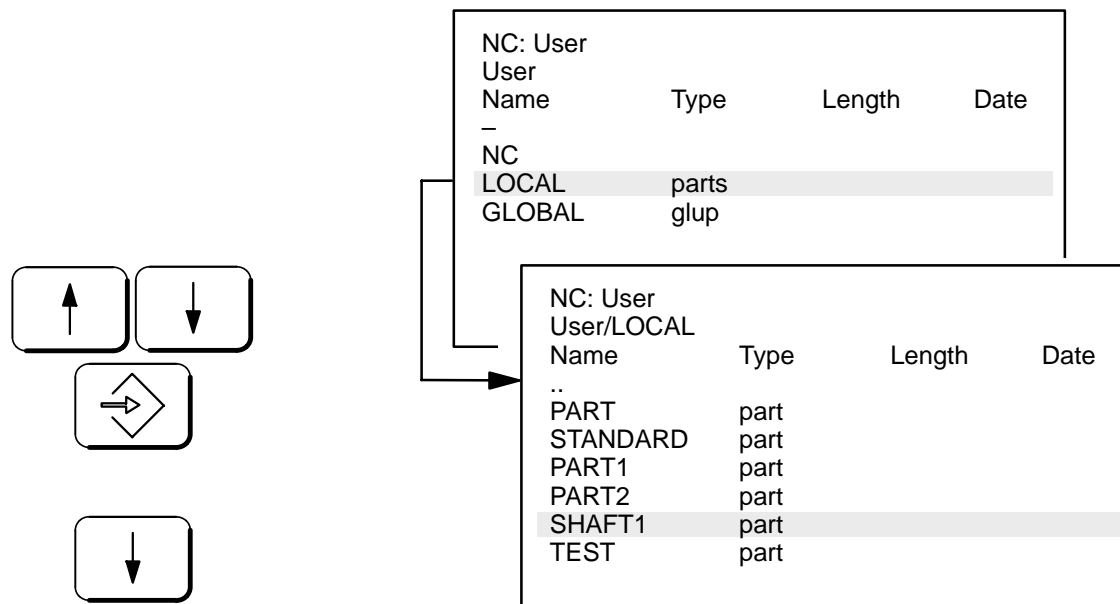
Acknowledge the message with the OK softkey.

## 6.1.1 Loading workpieces and programs

**Preconditions**

- You have selected the SERVICES/NC area.
- You have activated the user window with the HOME key.

**Operating sequence** Move through the user directories as follows and select the workpiece to be loaded into the processing memory (e.g. SHAFT1).



Press the LOAD START softkey and the following message appears briefly:

!!! NC directory being read !!!

If a part program already exists in the NCK memory, the following message appears:

Part program already exists. Overwrite ?

MPF2

Yes

No

Use the two softkeys YES and NO to decide whether the existing part program in the NCK memory is to be overwritten by the part program with the same number on the hard disk.

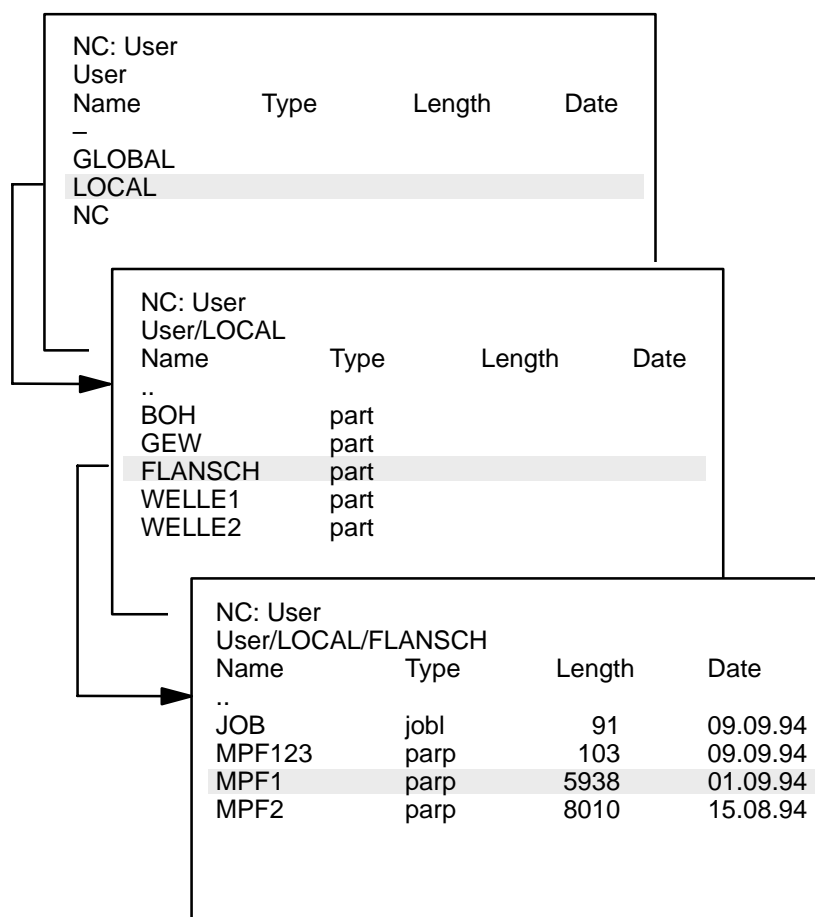
The file transfer is indicated by the following message:

Transfer from PC to NC in progress !!!	MPF1
--	------

Abort

Pressing the ABORT softkey during data transfer causes the transfer to be canceled.

You can also load a part program from any workpiece. The procedure is the same.





## 6.1.1 Loading workpieces and programs



*Notes*

Workpieces in the MACHINE/AUTOMATIC area can be loaded into the NCK memory by means of the SELECT WORKPIECE function **or** by means of the LOAD NC function in the SERVICES area. If the workpiece has no job list, all workpiece data stored are loaded into the working memory by means of the LOAD NC function, but no main program is selected for processing as in the case of the PROG./SELECT JOB function. If the workpiece has a job list, it is also interpreted with LOAD NC, i.e. only that data is loaded which is identified with LOAD instructions in the job list. If the workpiece is transmitted with LOAD NC, it is **not** selected for processing.

The program lengths displayed in the NCK and the MMC do not correspond.

If you load a program/workpiece and the part program on the MMC is, for example, 596 characters long, a length of only 564 characters is displayed in the NCK.

	Up to Software Version 2, the message output when an attempt is made to load a part program that is being processed is "Prog. being processed or cycle disable", with Software Version 4 and higher the message "No read/write rights" is output.	
---	---	---

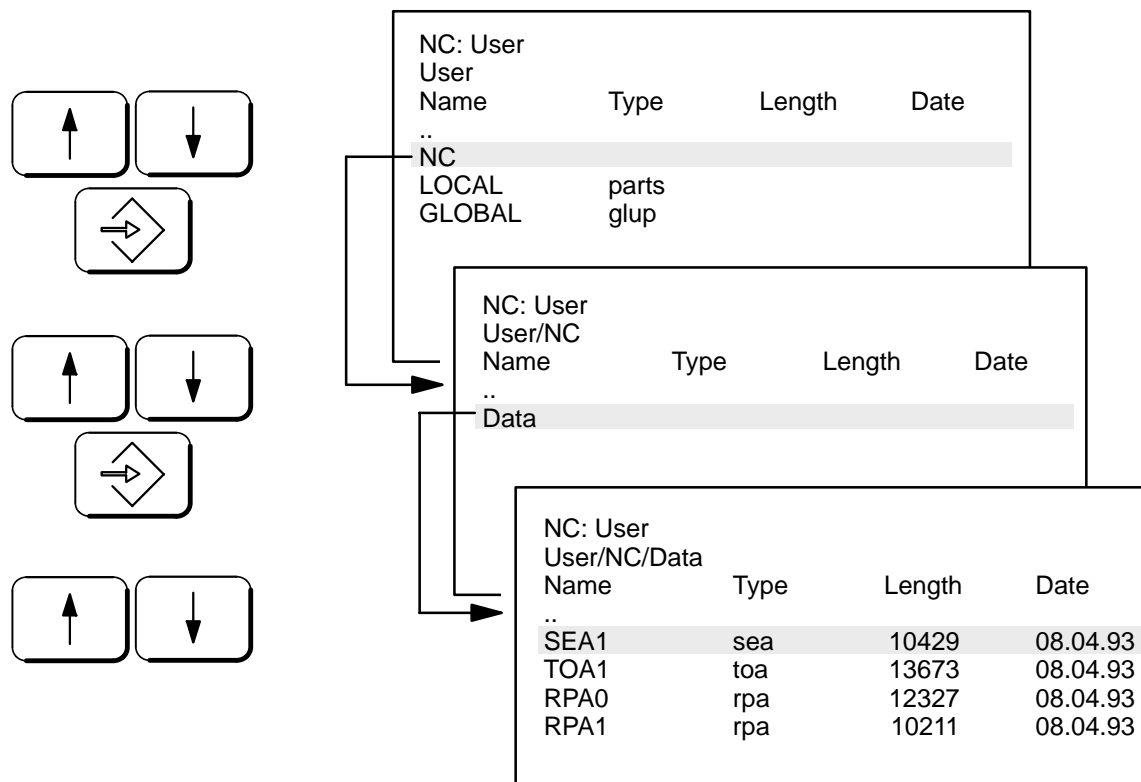
	<p>When loading workpieces the following priorities for channel interrogation are scanned by the control (also with automatic loading of the STANDARD workpiece)</p> <ol style="list-style-type: none"> <li>1. Channel specification in job list</li> <li>2. Channel specification in file name (last character of the file name)</li> <li>3. Current NC channel.</li> </ol>	
---	--	---

## 6.1.2 Loading data

### Preconditions

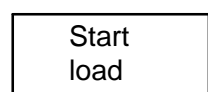
- The SERVICES area has been selected.
- In the user branch the NC, LOCAL or GLOBAL directory has been selected.

**Operating sequence** Page through the directories as shown below:

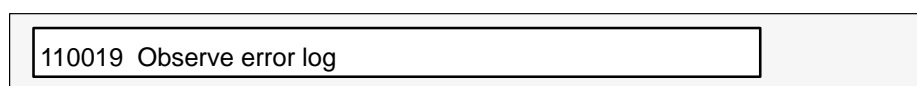


Use the arrow keys to select the file to be loaded into the NCK memory and start data transfer by

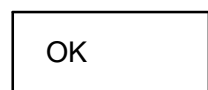
pressing the START LOAD softkey (see also “Loading workpieces”).

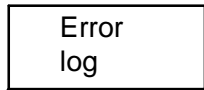


If an error occurs during data transfer, the following message appears in a dialog box:



Press the OK softkey to acknowledge the message.

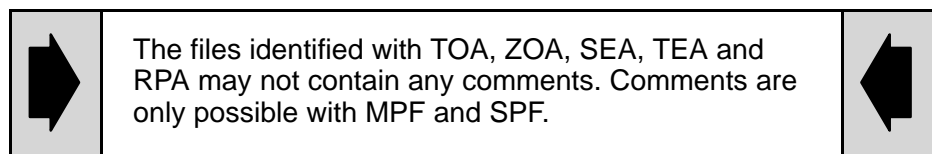
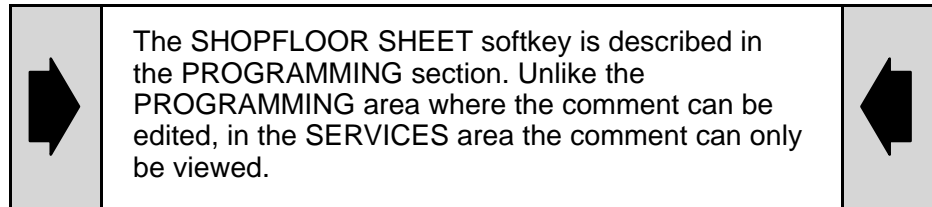




If you press this softkey, the faulty directory path and the faulty file are displayed in the error log.



Press RECALL to exit from the error log



Press RECALL to switch back to the basic display of the SERVICES area.



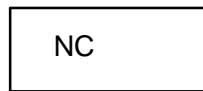
## 6.2 SAVE NC

In the SERVICES/NC area, you can transfer NCK data from the NCK memory onto the hard disk with the SAVE START function.

### Precondition

You are in the SERVICES area, the basic display is active.

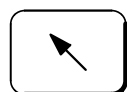
### Operating sequence



Press the NC softkey. The following selection display appears on the screen:

Machine	Parameter	Programm.	Services	Diagnosis
				15:43
NC: Standard				
Data				
Name	Type	Length	Date	
-				
NC: User				
User				
Name	Type	Length	Date	
-				
NC				
LOCAL	parts			
GLOBAL	glup			
Shop floor sheet	Save Start	Load Start	Error log	

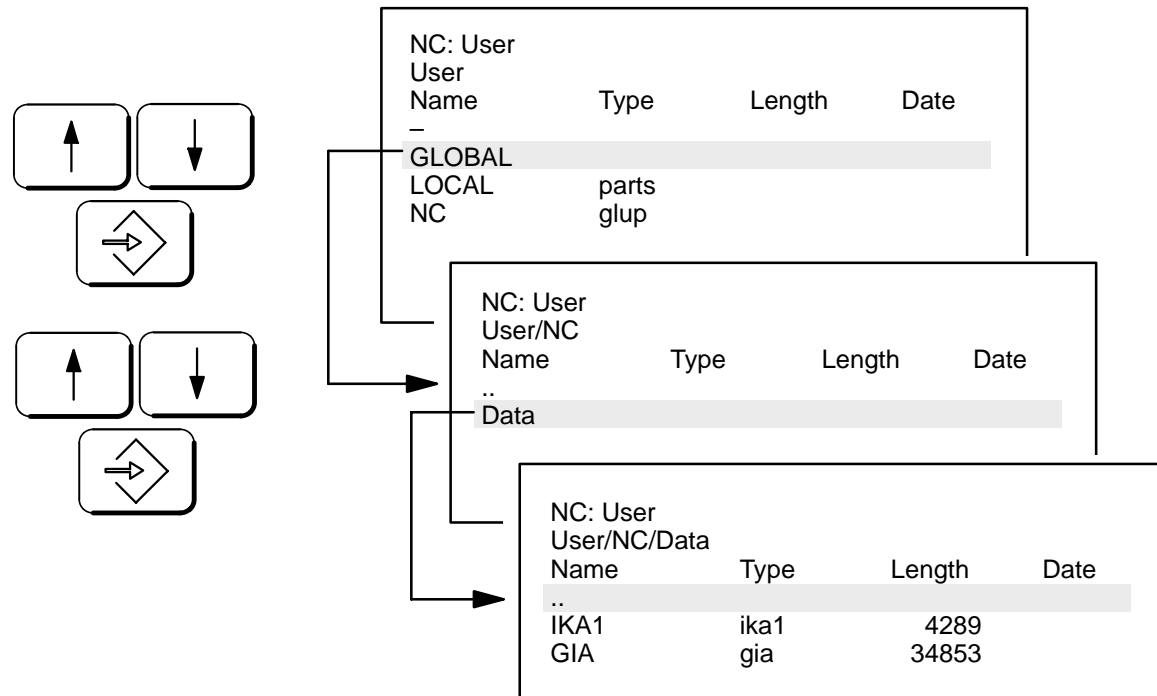
Fig. 6.4 NC save/load



Press the HOME key to switch to the bottom window (NC: User). The active window is highlighted by a yellow frame.

## 6.2.1 Saving NC data

*Operating sequence* Page through the directories in the user branch as follows:



Save  
Start

When you have reached the NC files directory, press the SAVE START softkey.

The following display appears:

Machine	Parameter	Programm.	Services	Diagnosis	10:13
NC: User					
User/NC/Data					
Name	Type	Length	Date		
NC source IKA configuration					
from/to	0	100			
Channel no./T0 area	0				
Name PC data	IKA3				
<p>Note :</p> <p>For data requiring a channel no. or T0 area, the channel no. or the T0 area is automatically added to the file name that has been entered.</p> <p>Example: T0 area = 3, name = T0A -&gt; File name: T0A3</p> <p>Channel no. 0 applies for data relating to all channels and the channel nos. greater than 0 apply for channel-specific data.</p> <p>Values greater than 0 are possible for the T0 area.</p>					
					OK

Fig. 6.5 Interactive screenform for the SAVE NC function



In the NC SOURCE selection field (toggle field) you can select the following data for saving using the selection key:

Gearbox interpolation data (GIA)

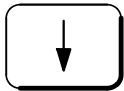
IKA configuration (IKA1)

IKA curve pointer (IKA2)

IKA points (IKA3)

	<p>If you try to select other data types offered (TOA, SEA, SEA4, ZOA, RPA, MPF, SPF), the error message "... saving of &lt;...&gt; not allowed here".</p> <p>These data can only be stored in a workpiece in directory GLOBAL or LOCAL.</p>	
--	--	--

	<p>It is advisable to save IKA data in directory NC/Data or in workpiece STANDARD. If these IKA data are not machine compensation data (e.g. beam sag compensation) but interpolation data (e.g. for camshaft grinding), they should be backed up in the corresponding workpiece. Please refer to the machine manufacturer's documentation.</p>	
--	---	--



After the data have been selected, press this cursor key to switch to the next input window. Enter the range of selected data to be saved (e.g. IKA3: 0 ... 100) in the FROM/TO input window and conclude with the INPUT key.

Enter the data identifier without channel number in the NAME PC data input field. The control adopts and stores this information from the CHANNEL input field. You can store different versions of a file by adding a digit to the identifier. The control then adds the channel number.

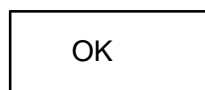
#### Note

You can choose any name when entering the PC data in the input box. The identifiers RPA, TOA, etc. do not have to be used. The type identifier (see section 1.4) provides the unique assignment. This assignment is automatically created by the operating system. The length of the name in the input field must not exceed 7 characters.

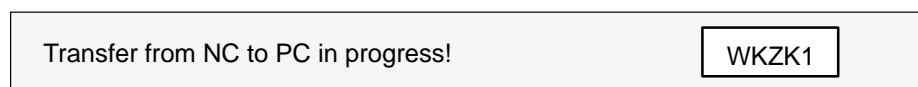
### Examples

NC source	Channel number	Entry in PC data name	Stored data	Type
TOA tool offsets	1	TOA	TOA1	toa
TOA tool offsets	1	WKZK1	WKZK11	toa
ZOA zero offsets	2	NPV	NPV2	zoa
SEA setting data	independent of channel	SETEIL1	SETEIL1	sea
GIA gearbox interpolation data	independent of channel	GIA	GIA	gia
SEA cycle setting data	3	SEA4	SEA43	sea4
SEA cycle setting data	3	SEA	SEA3	sea4
SEA setting data	independent of channel	SEA3	SEA3	sea

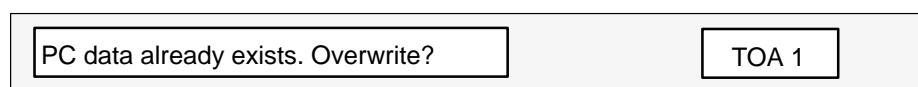
After you have filled in all input fields  
press the OK softkey.



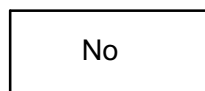
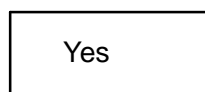
The following dialog text is displayed:



If the data has been stored already in the PC area under the same name, you are asked:

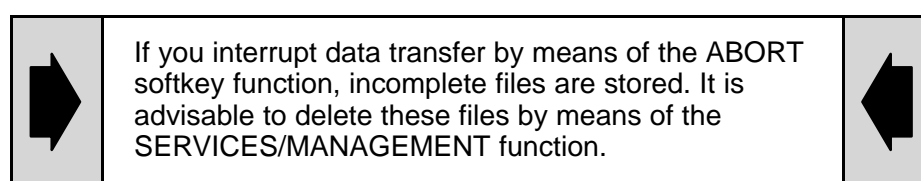
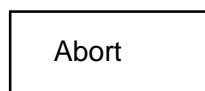


With these softkeys you decide whether existing data on the hard disk is to be overwritten.



### Note

For saving NC data in the NC/Data directory, the password must be set.  
If you press the ABORT softkey during transfer, the transfer is aborted.



## 6.2.2 Saving part programs or NCK data in a workpiece in the local or global directory

If an error occurs during the data transfer, the path and file which caused the error are indicated in the error log.



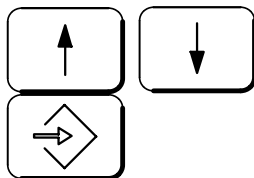
Press the RECALL key to return to the next-higher menu.

## 6.2.2 Saving part programs or NCK data in a workpiece in the local or global directory

### Preconditions

- The SERVICES/NC area has been selected.
- The user data (bottom window) has been selected with the HOME key and the NC/LOCAL/GLOBAL directory is displayed on the screen.
- In the directory, there is a workpiece in which you can store data.

### Operating sequence



With the cursor keys, position the cursor on LOCAL or GLOBAL and press the INPUT key.

The directory of all local or global workpieces is displayed.

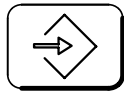
Machine	Parameter	Programm.	Services	Diagnosis	
					10:42
NC: User					
User/LOCAL					
Name	Type	Length	Date		
...					
ACR	part		11-03-1995 12:53:52		
ELG_LIN	part		11-03-1995 12:57:22		
EXMP_T	part		11-03-1995 12:59:28		
GIA_LIN	part		11-03-1995 12:58:02		
IKA_BSP	part		11-03-1995 12:54:40		
IKA_BSPG	part		11-03-1995 12:56:42		
IKA_KOMP	part		11-03-1995 13:02:30		
IKA_SIN	part		11-03-1995 13:02:44		
KGI_KREI	part		11-03-1995 12:58:18		
KGI_LISS	part		11-03-1995 12:58:50		
MPF1	part		11-03-1995 15:42:46		
N15	part		11-03-1995 13:56:26		
NCKTMP	part		11-02-1995 08:58:44		
PART	part		10-23-1995 18:02:22		
RECHTECK	part		11-03-1995 13:03:14		
SIM_EXMP	part		11-03-1995 12:57:36		
STANDARD	part		10-23-1995 18:02:24		
SUCH1	part		11-03-1995 13:03:22		
TB07081	part		11-03-1995 13:03:46		
TB07082	part		11-03-1995 13:03:58		
TEIL1	part		11-03-1995 12:56:30		
TEIL2	part		11-03-1995 12:56:38		

Fig. 6.6 Directory of local workpieces in the SERVICES/SAVE NC area

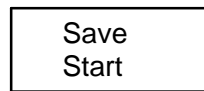
## 6.2.2 Saving part programs or NCK data in a workpiece in the local or global directory



Using the cursor keys, select the workpiece in which you wish to store the data and



confirm with the INPUT key.



Press the SAVE START softkey. The following interactive screenform is displayed:

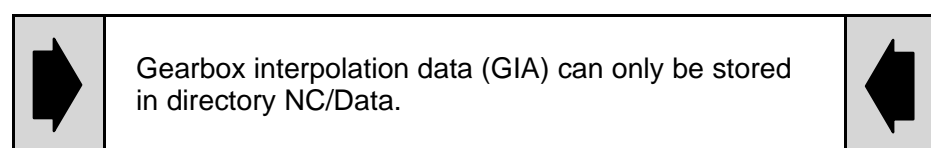
Machine	Parameter	Programm.	Services	Diagnosis	10:44
NC: User					
User/LOCAL/TB07081					
Name	Type	Length	Date		
NC source <b>TOA tool offsets</b> from/to <input type="text" value="0"/> <input type="text" value="100"/> Channel no./TO area <input type="text" value="1"/> Name PC data <input type="text" value="TOA"/>  <b>Note :</b> For data requiring a channel no. or TO area, the channel no. or the TO area is automatically added to the file name that has been entered.  Example: TO area = 3, name = TOA -> File name: TOA3 Channel no. 0 applies for data relating to all channels and the channel nos. greater than 0 apply for channel-specific data.  Values greater than 0 are possible for the TO area.					
					OK

Fig. 6.7 Interactive screenform for the SAVE part programs function

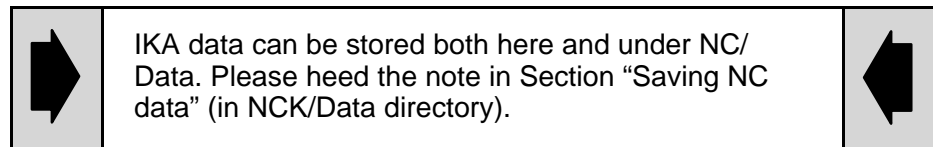


In the NC SOURCE selection field, you can select the following data for saving:

- Tool offsets (TOA)
- Setting data (SEA)
- Cycle setting data (SEA4)
- Zero offsets (ZOA)
- R parameters (RPA)
- IKA configuration (IKA1)
- IKA curve pointer (IKA2)
- IKA points (IKA3)
- Main programs (MPF)
- Subroutines (SPF)



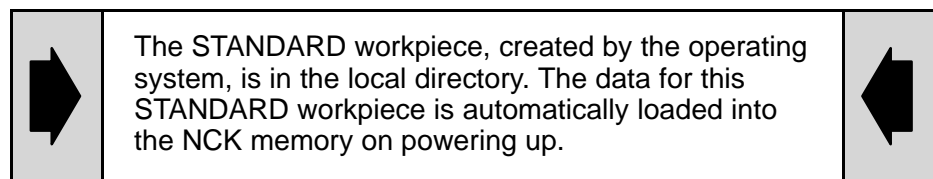
## 6.2.2 Saving part programs or NCK data in a workpiece in the local or global directory



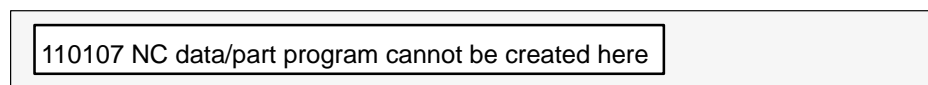
Several part programs can be transferred at the same time.

Presetting is performed in the FROM/TO input window (e.g. B. MPF 1... MPF 100, all programs between MPF 1 and MPF 100 are transferred). When transferring MPFs and SPFs, no entries need be made in the input fields NAME, PC-DATA and channel No. since the data identifier is adopted from the NCK.

The other operations are identical with those in the "Saving NC data" section.

*Note*

NCK data and part programs can only be stored into the file directories provided for this purpose. If the SAVE NC function is started in another directory level a dialog form with the following message appears on the screen:



Acknowledge the form with the OK softkey.

Press the RECALL key to return to the basic SERVICES display.

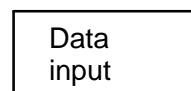


## 6.3 Data input from an external device

### Preconditions

- You have selected the SERVICE area.
- The interface has been correctly set for your data device (see the “Device settings” section).

### Operating sequence



Press the DATA INPUT softkey and the following interactive screenform is displayed:

Machine	Parameter	Programm.	Services	Diagnosis
				14:23
<b>Data input</b>				
Interface	Serial 1	Archive name	ARCHFILE	
Device		Existing PC data overwrite	Yes	
<b>Note:</b> Data in punch tape format that cannot be stored under the selected target path are placed in the buffer.				
<b>Data input: Target path</b>				
PC				
Name	Type	Length	Date	
-				
Buffer				
User				
Into new workpiece	Archive contents	Floppy	Input Start	Buffer Error log

Fig. 6.8 “Data input” interactive screenform

The display for the DATA INPUT function is divided in two windows.

You can switch between the windows with the window switchover key. The active window is marked (yellow frame).



The screenform in the upper window shows the currently parameterized interface and provides a toggle field (selection field).

Use the selection key to select whether data available in PC or punched tape format on the hard disk is to be overwritten.



### Overwriting existing PC data

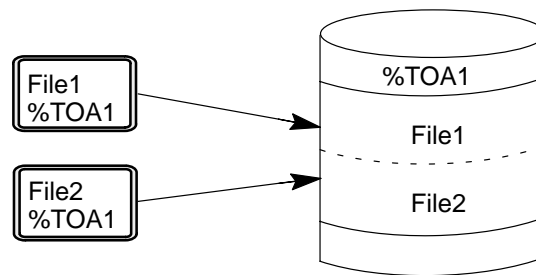
All data with the identifier %RPA, %TOA, %ZOA, %SEA, %IKA, and which are transmitted more than once in punch tape format are linked in the buffer but not overwritten if this is “no”.

The data are overwritten if the setting is “yes”.



*Example*

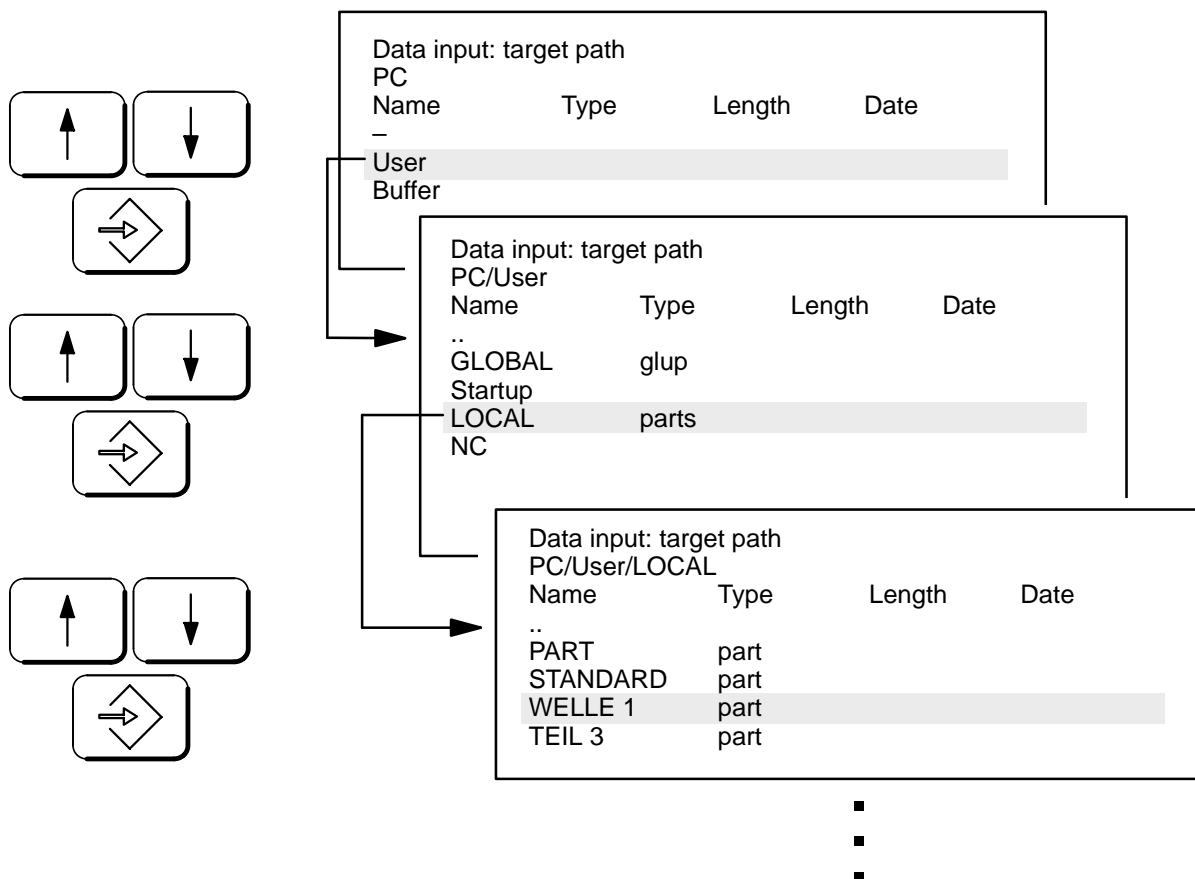
Reading-in two files with the identifier TOA1:

*Notes*

Data in PC format can be generated and edited externally with PCIN V3.1 and higher, i.e. you can only read-in data in PC format if it has been output from the SINUMERIK 840C. When the data is output, information stating from which directory and the type of data is also output and recognized when reading-in again.

The entry in the "Archive name" input field is only used for data communication with an external floppy drive. The name entered here is created on diskette as an archive. Data read out from the system are stored in this archive. When the control reads in the data from the floppy drive, it accesses the data in the archive file whose name is entered in the "Archive name" input field.

In the lower window you can specify the directory in which the read-in files or directories are to be stored. If the window TARGET PATH is active, page through the directories as follows until you have reached the target directory:



## 6.3 Data input from an external device

Into new  
workpiece

By pressing this softkey you create a new workpiece in which you can then read-in files. If you press the INTO NEW WORKPIECE softkey, you are requested to enter the desired workpiece name:

Name of the new workpiece

PART 1

Enter a new name via the alphanumeric keyboard and acknowledge with the INPUT key.

Input  
Start

Press the INPUT START softkey and data transmission begins. The following interactive screenform appears:

Data transmission PC/external

PC format

MPF13

1536

It displays:

- the format of the transmitted file (punched tape or PC format)
- the file identifier and
- the number of characters transmitted.



If the read-in files cannot be assigned to the selected destination directory, they are stored in the clipboard. Data in the PC format contain information on destination path and destination directory. If none of these exist they are created during the read-in process.



Files which are stored in the buffer can be assigned to a directory by means of the PASTE FROM CLIPBOARD function.

If an error occurs during data transmission the following interactive screenform is displayed:

110019 Observe error log

Acknowledge this form by pressing the OK softkey.

Error  
log

If you press the ERROR LOG softkey, the faulty file and the directory as well as the error message are displayed:

e.g. /PC/User/LOCAL/PART 1/MPF4711  
<110002> Name < MPF4711 > already exists

When transmitting files with the default "overwrite existing PC data" = no, all files which already exist on the hard disk are entered in the error log.



The serial interfaces on the MMC must not be operated simultaneously.



**Note**

When reading in programs in tape format problems arise if the subroutine call Lxx00 LF is programmed separately (System 3 format), e.g. L3500 LF. In this case, the rest of the program is interpreted as subroutine L35 and is therefore stored in the clipboard or the workpiece.

From SW 6, System 3 format is deselected.

**Reason**

SINUMERIK 840C can read System 3 LS format.

**Remedy**

Use the block number, e.g. N77 L3500 LF.

### 6.3.1 Insert data from the clipboard

**Operating sequence** When data transmission has been completed press RECALL to return to the menu.

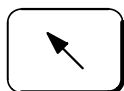
Data input	Data output	Print	Archive list	Devices	Format	Clipboard
------------	-------------	-------	--------------	---------	--------	-----------

Clipboard

Press the CLIPBOARD softkey and the following display appears:

Machine	Parameter	Programm.	Services	Diagnosis	
					11:44
Data input/Clipboard					
Buffer					
Name	Type	Length	Date		
-					
MPF1	parp	295	11-06-1995	11:23:32	
MPF1012	parp	231	11-06-1995	11:24:10	
MPF11	parp	335	11-06-1995	11:23:32	
MPF22	parp	181	11-06-1995	11:23:32	
MPF23	parp	111	11-06-1995	11:23:32	
MPF4711	parp	104	11-06-1995	11:23:34	
RPA1	rpa	115	11-06-1995	11:23:34	
SEA1	sea	13411	11-06-1995	11:24:10	
Data input/Clipboard/Insert: Target directory					
User/LOCAL/WELLE2					
Name	Type	Length	Date		
JOB	job1	1898	09-22-1995	11:04:52	
MPF1	parp	294	09-22-1995	11:04:52	
MPF1012	parp	236	09-22-1995	11:04:52	
MPF11	parp	334	09-22-1995	11:04:52	
MPF22	parp	181	09-22-1995	11:04:52	
Del buffers all    Del buffers singly    Insert in target					

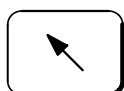
Fig. 6.9 Buffer in the SERVICES area



Press the HOME key to switch top the upper CLIPBOARD window. The active window is marked (yellow frame).



Using the cursor keys, select a file which you wish to move in the user area.



Press the HOME key to switch to the lower window CLIPBOARD/INSERT.

## 6.3.1 Insert data from the clipboard

Using the cursor keys and the input key page through the directories until you have reached one into which you wish to move the file selected.

Insert in  
target

Press the INSERT IN TARGET softkey; the following dialog screenform is displayed:

Please enter name

MPF 123

OK

If you wish to retain the file name from the buffer, press the OK softkey.

If you wish to rename the file, enter the new file name via the alphanumeric keyboard and conclude with the INPUT key. Press the OK softkey to transfer the file into the directory.



When files are moved from the buffer they **cannot** be overwritten. Already existing files must be copied and renamed or deleted, if necessary in the SERVICES/MANAGEMENT or PROGRAMMING area.



In the case of part programs with the identifiers MPF or SPF, only the four-digit program number can be changed but not the identifier since the latter is required for loading into the NCK memory.



Del buffers  
singly

Using the DEL BUFFERS SINGLY softkey in the SERVICES/DATA-INPUT area you can delete data only from the buffer. For this purpose, select the files to be deleted and press the softkey.

The following screenform is displayed:

Do you want to delete the data?

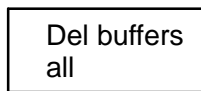
MPF 123

OK

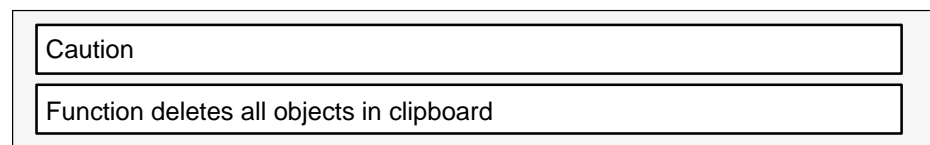
Press the OK softkey to delete the selected data in the buffer.



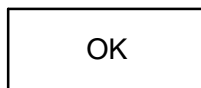
Press the RECALL key if you do not wish to delete.



Using this softkey you delete all files from the buffer. If you press this key, the following dialog text is displayed:



Press the RECALL key to leave the menu if you do not wish to delete the files.



The function is executed if you press the OK softkey.

### 6.3.2 Continuous mode for data input via RS232C (V24) (from SW 5.6)

Continuous mode for reading in of part programs via an RS232C (V24) interface is possible for SINUMERIK 840C from SW 5.6. Continuous mode means that the control remains ready to receive as long as desired when an "Input Start" for a serial interface has been given.

This operating mode can be permanently specified for each individual interface of the 3 possible RS232C (V24) interfaces of the MMC. The corresponding entries must be made in the file BEDCONF (line 191) in the directory Operation/SERVICES.

The following points must be observed for continuous mode:

- "Stop at end character" must be set when programming the interface.
- Only data in tape format may be read in.
- During data transmission (between %... and M02/M17/M30), transmission pauses caused by the external device must not exceed 5 seconds. Longer pauses or reading in of another %... cause the currently read-in file to be terminated on the hard disk. M02/M17/M30 do not have any effect here.
- The RS232C (V24) interface is closed when the end character (ETX) is received, by the "Abort" softkey or when errors occur.

## 6.4 Data output to an external device

With the SINUMERIK 840C you can output user data to an external device. It is advisable to backup all user data on an external device, because if a defect occurs on the MMC-CPU this data would be irretrievably lost. The standard operating software additionally includes the ARCHIVE LIST function. By means of this function you can open an archive with any amount of user data and output it as a block.

The external device can be a

- Programmer
- PC
- Floppy drive FD-E2 or a
- Tape reader.

When outputting to an external device, the output formats and device settings must be considered. Please bear in mind that your data saving device must be capable of receiving and storing the data sent.

*Precondition*

You have selected the basic display of the SERVICES area.

*Operating sequence*

Data  
output

Press the DATA OUTPUT softkey and the following display appears:

Machine	Parameter	Programm.		Services	Diagnosis																																																																	
						15:00																																																																
Data output: Data																																																																						
Interface		Serial 1		Archive name		ARCHFILE																																																																
Device		STANDARD		Format		PC format																																																																
<table border="1"> <thead> <tr> <th colspan="4">User</th> </tr> <tr> <th>Name</th> <th>Type</th> <th>Length</th> <th>Date</th> </tr> </thead> <tbody> <tr> <td colspan="4">-</td> </tr> <tr> <td colspan="4">Operation</td> </tr> <tr> <td>Data_management</td> <td></td> <td></td> <td></td> </tr> <tr> <td>GLOBAL</td> <td>glup</td> <td></td> <td></td> </tr> <tr> <td>Start-up</td> <td></td> <td></td> <td></td> </tr> <tr> <td>List_module</td> <td></td> <td></td> <td></td> </tr> <tr> <td>LOCAL</td> <td>parts</td> <td></td> <td></td> </tr> <tr> <td>NC</td> <td></td> <td></td> <td></td> </tr> <tr> <td>PLC</td> <td></td> <td></td> <td></td> </tr> <tr> <td>Master_control</td> <td></td> <td></td> <td></td> </tr> <tr> <td>Services</td> <td></td> <td></td> <td></td> </tr> <tr> <td>Drives</td> <td></td> <td></td> <td></td> </tr> <tr> <td>Simulation</td> <td></td> <td></td> <td></td> </tr> <tr> <td>WOP</td> <td></td> <td></td> <td></td> </tr> </tbody> </table>							User				Name	Type	Length	Date	-				Operation				Data_management				GLOBAL	glup			Start-up				List_module				LOCAL	parts			NC				PLC				Master_control				Services				Drives				Simulation				WOP			
User																																																																						
Name	Type	Length	Date																																																																			
-																																																																						
Operation																																																																						
Data_management																																																																						
GLOBAL	glup																																																																					
Start-up																																																																						
List_module																																																																						
LOCAL	parts																																																																					
NC																																																																						
PLC																																																																						
Master_control																																																																						
Services																																																																						
Drives																																																																						
Simulation																																																																						
WOP																																																																						
Work-pieces	Archive list		Floppy		Output Start	Error log																																																																

Fig. 6.10 Data output screenform

The screen for the data output function is divided into two windows.



You can switch between the two windows by pressing the window switchover key. The active window is highlighted by a yellow frame. The top window displays the current interface parameter, an input field and a toggle field (selection field).



Press the selection key in the toggle field to select whether the data are to be output in PC format or punch tape format.

The format is preset via the DEVICE SETTINGS function. If you change this setting in the toggle field provided here, the newly set format is effective until you leave the DATA OUTPUT menu.

When reselecting the DATA OUTPUT function, the transmission format is taken from the device setting of the currently parameterized interface.

The entry in the "Archive name" input field only applies to data communication with an external floppy drive. The name entered here is created as an archive on the diskette. The data read-out from the system are stored in this archive.

All the directories/files which can be read-out are displayed in the bottom "User" window.

You can use the arrow keys and the INPUT key to browse through the directories until you find the file or directory that you want to read-out (see the section "Data output").

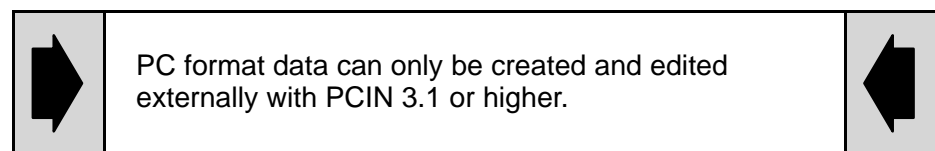
## 6.4.1 Output formats

### 6.4.1.1 Punch tape format

Only NC data, i.e. data with the % identifier (%TOA, %TEA1, %TEA2, etc.) can be transferred in punch tape format (for description of the punch tape format see Programming Guide SINUMERIK 840C). Files and directories are output without reference to the directory level or the path. Data in punch tape format can also be edited externally.

### 6.4.1.2 PC format

All existing data can be output in PC format. Data in PC format contains all information regarding the path or the directory from which data has been read-out. When reading-in this data is loaded in a similar manner which means that it does not have to be temporarily stored in the buffer.

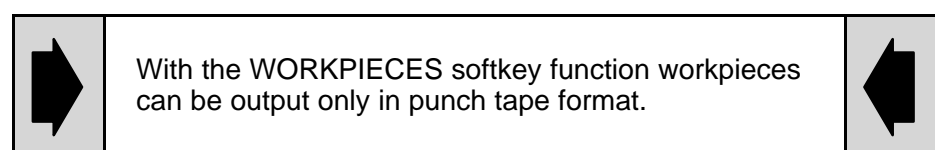
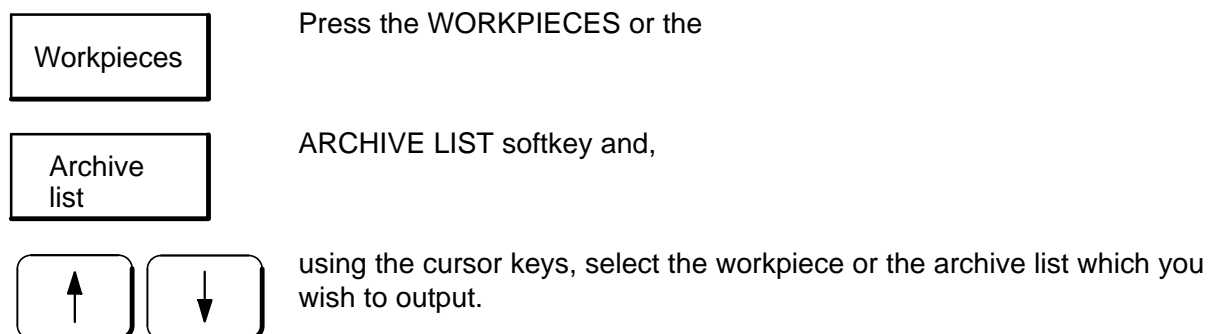


Data in PC format is written on the external device as byte stream complying with the standardized "tar Format". The PC format standard is described in:

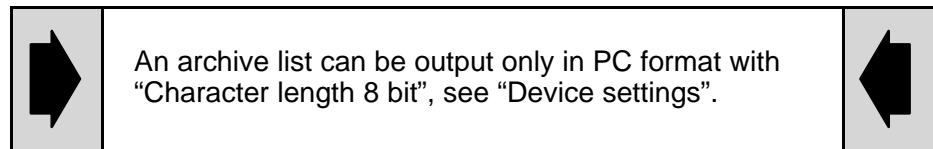
- X/OPEN Company Ltd.: Portability Guide, XSI Supplementary Definitions. Prentice Hall, Englewoods Cliffs (NJ), USA, 1989,
- IEEE: Portable Operating System for Computer Environments (POSIX), IEEE Std 1003.1–88. Institute of Electrical and Electronics Engineers Inc., New York (NY), USA, 1986.

## 6.4.2 Outputting workpieces and archives

### *Operating sequence*

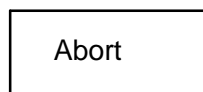
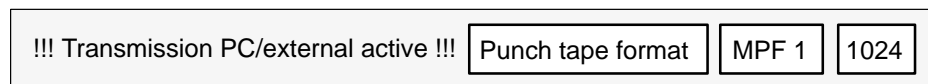






Press the START softkey to start transmission.

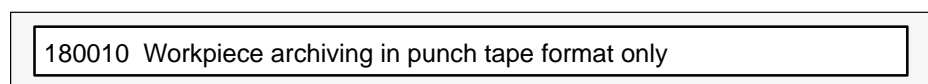
The following dialog screenform is displayed:



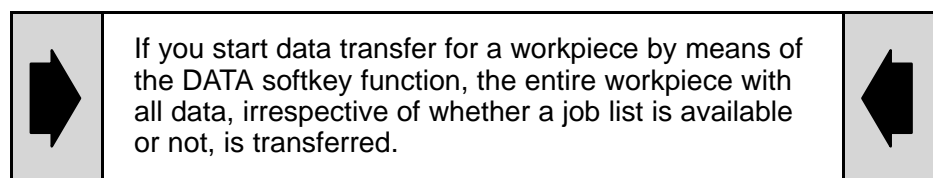
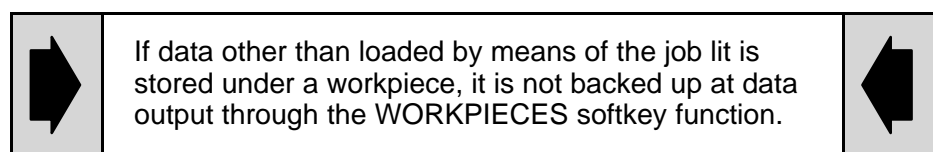
Press the ABORT softkey to interrupt data output.

#### Notes

- You can only output workpieces from the LOCAL directory with the WORKPIECES softkey function.
- If you output a workpiece via the RS232C(V.24) interface by means of the WORKPIECES function, the job list (if available) is interpreted at the time of data output, i.e. only that data that is output which has been loaded with the LOAD instruction.
- If the workpiece has **no** job list, only that data in punch tape format belonging to the workpiece is output.
- A job list **cannot** be output in punch tape format.
- If you start data output with the WORKPIECES softkey and if you have selected PC format as transmission format, then a dialog box with the following text is displayed:



Acknowledge the dialog box by pressing the OK softkey.

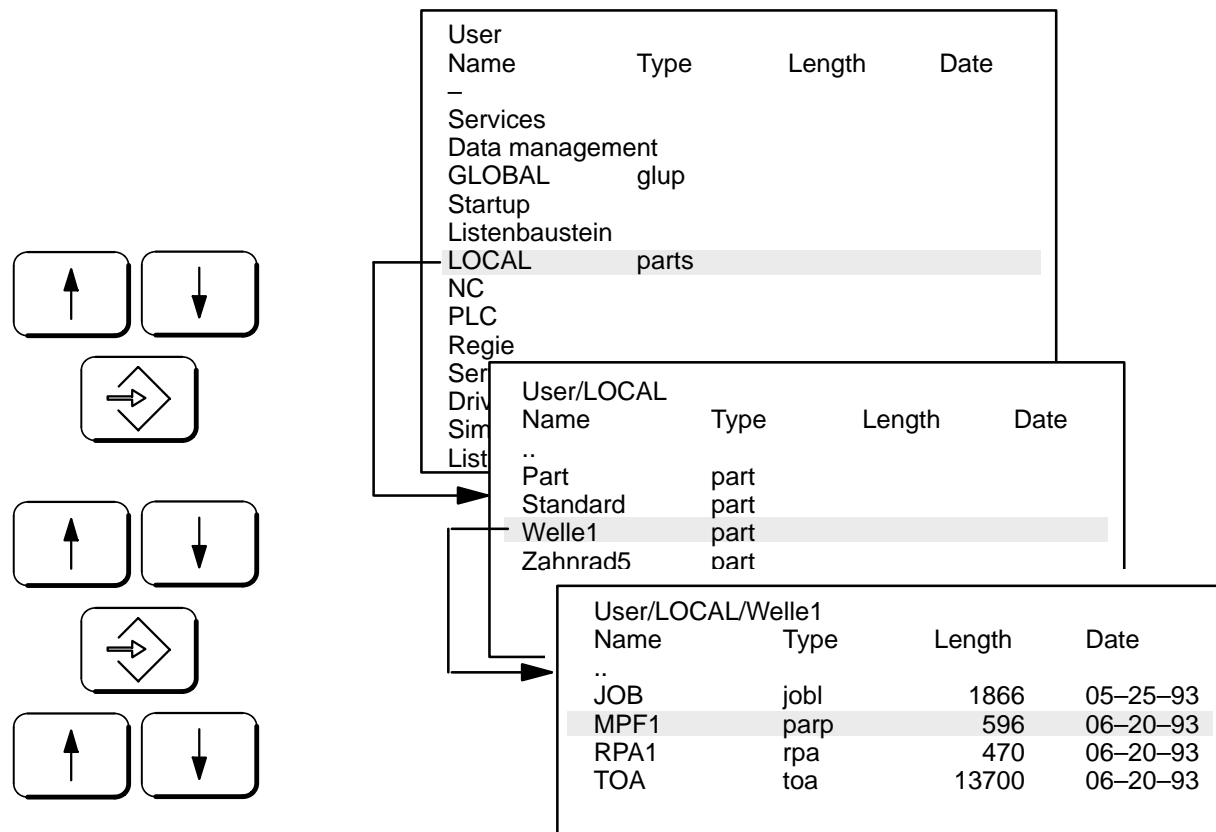


## 6.4.3 Data output

### Preconditions

- The SERVICES/DATA OUTPUT area is selected.
- The bottom "User" window has been activated with the HOME key.

**Operating sequence** Browse through the directories as follows:



When you have reached the source directory, use the cursor keys to select the file you wish to output (you can recognize files by the entry in the LENGTH column).

Press the OUTPUT START softkey to start transmission. The following dialog screenform is displayed:

Output Start

!!! Transmission PC/external active !!!	PC format	MPF 1	1024
---	-----------	-------	------

Press the ABORT softkey to interrupt data transfer.

If an error occurs during data transfer, the following dialog text is displayed:

Abort

110019 Observe error log
--------------------------

Press the OK softkey to acknowledge the dialog text.

If you press the ERROR LOG softkey, all files in which errors have occurred during data transfer are displayed.

Error log

## 6.5 Data input and output with the FD–E2 floppy drive (option)



### Warning

The user is responsible for ensuring that no “virus-infected” data are imported into the system.

The control has a virus protection activated in BIOS that monitors the boot sector and the DOS file allocation table (FAT).

### Preconditions

The FD–E2 floppy drive (6FC5135–0AA01–0AA0) can be optionally integrated in the front panel of a color screen. The “MMC interface” module must be plugged in. If the optional software module has been activated, the SERIAL 2 and SERIAL 3 interfaces are available. The parameters can be set for these interfaces from the device settings menu.

Use the following device settings for the LDF floppy (Long Distance Floppy):

Leader, trailer	yes
Stop at	End character
End of block	CRLF
Transmission format	PC format

**Operating sequence** From the SERVICES basic display select the setting for the floppy drive:

Interface	Device	Status
Floppy	LDF	inactive
Serial1	STANDARD	inactive
Serial2	PG_750	inactive
Serial3	PC	inactive

The parameterization of the interface is described in the section “Device settings”. The operator actions for transferring data with the FD–E2 are identical to those of the FD–E1.

### Overview

Overview of serial interfaces:

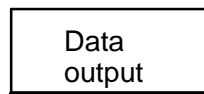
Interface (monitor)	Module	Designation on modul
Floppy	Int. MMC	X131
Serial 1	MMC	X151 (SERIAL 2)
Serial 2	Int. MMC	X121
Serial 3	Int. MMC	X111

(Int. MMC : Interface MMC)

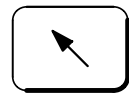
### Note

For data input and output with floppy drive FD–E1 no longer available from the end of 1995 see also next Section (6.6).

## 6.5.1 Data output with the FD-E2 floppy drive

**6.5.1 Data output with the FD-E2 floppy drive**

Press the DATA OUTPUT softkey from the basic display of the SERVICES area.  
The screen is divided into two windows for the data output function.



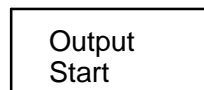
Use the HOME key to switch between the windows. The active window is highlighted by a yellow frame. The current interface parameter appears in the top window, e.g.:

SERIAL 1  
FLOPPY



Use the selection key in the FORMAT toggle field to select whether the data are to be transferred in PC format or punch tape format. In the "Archive name" input field enter the name of the archive file where the output data are to be stored on diskette.  
The name must not be longer than 8 places.

Press the HOME key to switch to the bottom "User" window. Use the arrow keys and the INPUT key to browse through the directories until you find the directory or file that you want to transfer to diskette.

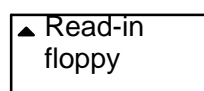


Press the OUTPUT START softkey and the selected file or directory is stored on diskette. The file or directory is stored under the archive name together with all of the related files.

*Read diskette*

Press the FLOPPY softkey to display a screen with the boxes

- Floppy directory
- Archive name, length, date, time.



Press the READ-IN FLOPPY softkey. The following message appears:

Floppy being read

When the floppy has been read, the message disappears and the contents of the floppy appear on the screen. The display lists all of the archive files on the diskette.

## 6.5.1 Data output with the FD-E2 floppy drive

Machine	Parameter	Programm.	Services	Diagnosis	
					16:06
Data output/Floppy					
Floppy directory:		1390592	Bytes free		
Archive name	Length	Date	Time		
ARCHFILE.001	33792	13.01.1994	15:56		
MACH1.001	4096	13.01.1994	16:00		
MACH2.001	5120	13.01.1994	16:01		
PLCPRG.001	22528	13.01.1994	16:02		
Read-in floppy			Archive Select		

Fig. 6.11 Floppy directory

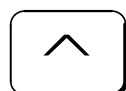


You can delete archive files on the diskette. Use the arrow keys to select an archive file and press the DELETE ARCHIVE softkey.



The following message appears:

Delete archive?	PART . 001
-----------------	------------



Press RECALL to cancel the process.



If you press the OK softkey, the archive and all files assigned to it are deleted from the hard disk.

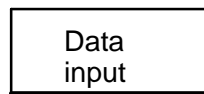


You can use the ARCHIVE SELECT softkey to enter the setting in the "Archive name" input field in the "Data output" menu. Select an archive file with the vertical arrow keys and press the softkey.



Press RECALL to switch back to the basic SERVICES display.

## 6.5.2 Data input from the FD-E2 floppy drive

**6.5.2 Data input from the FD-E2 floppy drive**

Press the DATA INPUT softkey in the basic display of the SERVICES area.

The screen is divided into two windows for the data input function.

Use the HOME key to switch to the top window. In the "Archive name" input field enter the name of the archive file that you want to read in from the diskette.

Pressing the ARCHIVE CONTENTS softkey starts transfer of the files and directories stored on diskette under the archive name.

The following message appears:

Data transfer PC/external	Punch tape format	MPF 5	1024
---------------------------	-------------------	-------	------

If the data are read-in in punch tape format, the transfer is executed completely. If no target directory has been selected, the data are stored in the clipboard (see the section "Data input from an external device"). During transfer of the data in PC format the following display appears, for example:

Machine	Parameter	Programm.	Services	Diagnosis
16:08				
<b>Data input</b>				
Interface		Serial 1	Archive name ARCHFILE	
Archive ARCHFILE				
/PC/User/LOCAL/SHAF T1/JOB /PC/User/LOCAL/SHAF T1/MPF 13 /PC/User/LOCAL/SHAF T1/MPF 14 <b>/PC/User/LOCAL/SHAF T1/SPF 103</b> /PC/User/LOCAL/SHAF T1/MPF 4711 /PC/User/LOCAL/SHAF T1/WERKSTA /PC/User/LOCAL/SHAF T1/RPA1 /PC/User/LOCAL/SHAF T1/SEA /PC/User/LOCAL/SHAF T1/ZOA1 /PC/User/LOCAL/SHAF T1/SHAF T				
Display again		remove		OK

Fig. 6.12 Data input from diskette

The data of the selected archive are displayed.

Use the arrow keys to select a file.

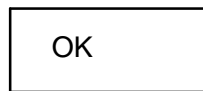




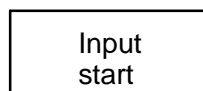
Press the REMOVE softkey to delete the selected file from the buffer, i.e. the contents of the diskette remain the same.



Press this softkey to display the contents of the diskette again.

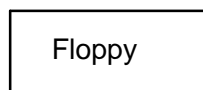


Press the OK softkey to terminate the data transfer, i.e. the files are transferred to the hard disk.



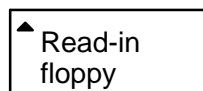
Press the INPUT START softkey (when the floppy interface parameter is set). The data input is performed without displaying the contents of the archive. The archive file specified by the archive name is transferred.

#### *Read-in diskette*



Press the FLOPPY softkey. A display with the following fields appears:

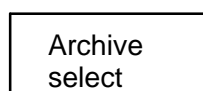
- Floppy directory
- Archive name, length, date, time.



Press the READ-IN FLOPPY softkey to display all of the archive files on the diskette.



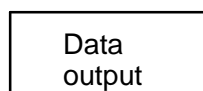
Use the arrow keys to select an archive file from this list.



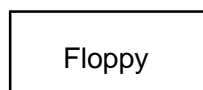
Pressing the ARCHIVE SELECT softkey enters the name of the selected archive file in the "Archive name" input field. The data input can then be started with the INPUT START or ARCHIVE SELECT softkeys.

### **6.5.3 Formatting diskettes**

#### *Operating sequence*



Select the DATA OUTPUT softkey in the basic SERVICES display.



Press the FLOPPY softkey.

<div style="border: 1px solid black; padding: 5px; width: fit-content;">Format diskette</div>	<p>Press the softkey and the following message appears:</p> <div style="border: 1px solid black; padding: 10px; margin: 10px 0;">Please insert floppy</div>
<div style="border: 1px solid black; padding: 5px; width: fit-content;">OK</div>	<p>Press the OK softkey to start formatting the diskette in the external floppy drive FD–E2 and the following message appears:</p> <div style="border: 1px solid black; padding: 10px; margin: 10px 0;">Now formatting</div>
	<div style="border: 1px solid black; padding: 10px; margin: 10px 0;"> <div style="display: flex; align-items: center; justify-content: space-between;"> <div style="background-color: #cccccc; padding: 5px; text-align: center;">➔</div> <div style="flex-grow: 1; text-align: center;"> <p>You cannot abort the formatting process. Press RECALL to exit the menu and perform other operator actions.</p> </div> <div style="background-color: #cccccc; padding: 5px; text-align: center;">➔</div> </div> </div>
<div style="border: 1px solid black; padding: 5px; width: fit-content; text-align: center;">⤴</div>	<p>Press RECALL to return to the basic SERVICES display.</p>

## 6.6 Data input and output with the SINUMERIK FD–E1 floppy drive (option)

### *Preconditions*

- The SERVICES area is selected.
- Interface “Serial 1” is selected in basic display SERVICES
- The FLOPPY interface parameters have been set and selected (see the section on device settings).
- The floppy drive is correctly connected (the FD–E1 is connected to the RS–232/V24 interface of the MMC–CPU).
- A 1.44 MB diskette (3.5” HD) is inserted in the drive (not copy-protected).

### *Operation*

The operations required for data transmission with the FD–E1 are identical to those of the FD–E2.

### *Note*

As data transmission with the FD–E2 is much faster we recommend installation of module Int.MMC with FD–E2.

➔

The FD–E1 is no longer available. The above description is only included to complete the description.

➔

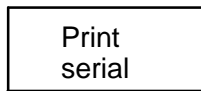


## 6.7 Printing out files

### Preconditions

- You are in the SERVICES area and DATA IN/OUT has been selected.
- The printer is connected correctly and the interface has been edited and configured accordingly (see the “Device settings” section).

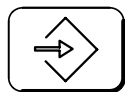
### Operating sequence



Press the PRINT SERIAL softkey. The user and the Siemens branch are displayed.



Use the cursor keys and the



INPUT key to select the user branch. The following directory is displayed:

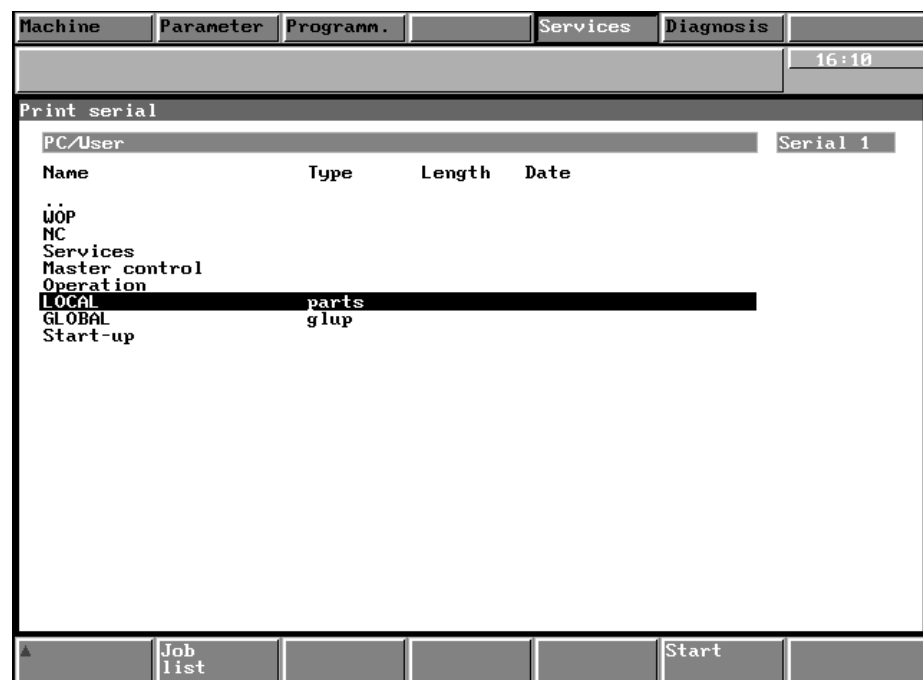
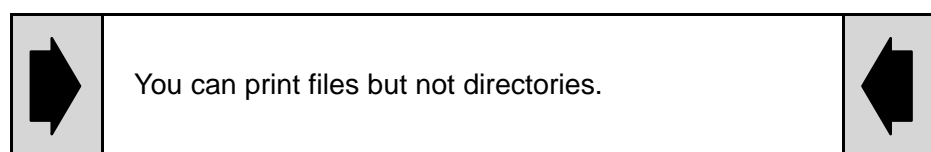


Fig. 6.13 User branch directory in the SERVICES/PRINT area

Using the cursor keys and the INPUT key, page through the directories until you have encountered the files you wish to print.



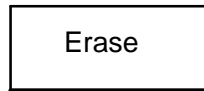
You can print files from various directories one after the other. Page with the cursor keys and the INPUT key through the corresponding directories.



Press the START softkey to transfer the files to be printed in the job list; printing of the first print job is immediately started.



If you press the JOB LIST softkey all files which you have selected for printing beforehand are displayed as well as the files which still have to be printed. A file which has been printed completely is deleted from the job list.



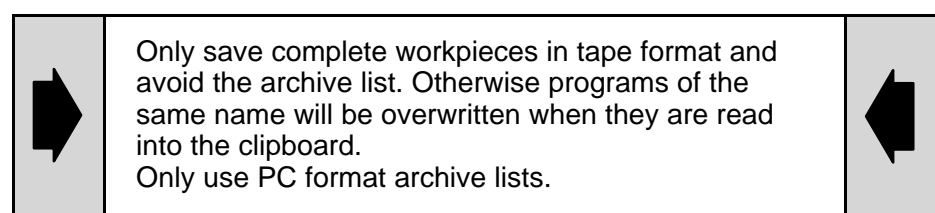
By pressing the ERASE softkey you can delete data in the job list which you have previously selected.

## 6.8 Archive lists

### General

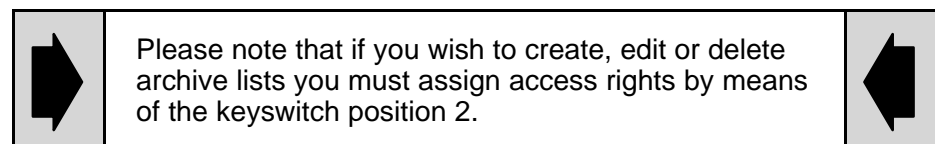
On the SINUMERIK 840C, you can file user data in an archive. An archive list is a list of files which can be read out via the RS232C(V.24) interface or floppy interface (FD-E2). The archive list contains only the names of directories and files of the user data to be archived. The list is not interpreted until data output takes place, i.e. the corresponding files are output to the external device (PC, programmer, etc.). The archive list contains a list of directories etc. that must be archived on diskette again and again without them being selected individually each time; it represents an instruction list for the data to be archived. When reading-in, the files and subdirectories are assigned again to the source directory. You can assign complete directories, workpieces, files and other archive lists to an archive list. You should select PC format as output format for archive lists. Only in this format will the information be included in the output via the source directory.

Archive lists can also be output in punch tape format. When an archive list is read in to the control in punch tape format, all of the files in the archive list are entered in the clipboard. Operator actions are provided to allow you to insert these files from the clipboard into the target directory.



### Precondition

The SERVICES area has been selected.



## 6.8.1 Creating new archive lists

### Operating sequence

Data  
output

Press the DATA OUTPUT softkey.

Archive  
list

When you press the ARCHIVE LIST softkey, the following display appears:

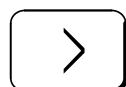
Name	Type	Length	Date
ARCH_1	arc	356	11-08-1994 15:44:02
ARCH_2	arc	398	11-08-1994 15:44:56

Fig. 6.14 Archive list directory

The bottom "Archive lists" window displays all of the existing archive lists.

### Note

Pressing the DATA softkey returns you to the SERVICES/DATA OUTPUT display. Press RECALL to return to the basic SERVICES display.



Press the ETC key to extend the displayed softkey bar with the following options:

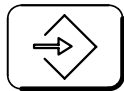
New

If you press the NEW softkey, the following menu is displayed:

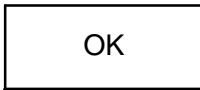
## 6.8.1 Creating new archive lists



You can enter a name for the new archive list on the alphanumeric keyboard and



confirm the entry with the INPUT key.



Press the OK softkey to acknowledge and the following display appears:

Machine	Parameter	Programm.	Services	Diagnosis	
					11:59
Data output/Edit archive list					
Archivelist				Type	CW1000
/PC/User/Operation/Basic settings				bdq1	
/PC/User/GLOBAL				glup	
/PC/User/PLC/Programs				progs	
/PC/User/Services/Archive lists				arcs	
/PC/User/Services/Interfaces				devs	
/PC/User/Simulation				simul	
/PC/User/WOP				wop	
/PC/User/LOCAL/IKA_BSP				part	
/PC/User/LOCAL/WELLE1/JOB				job1	
User/Operation					
Name	Type	Length	Date		
ANWENDE	mode				
Basic_settings	mode				
DG_PLC	mode				
DIAGNOS	mode				
store		remove		Include	

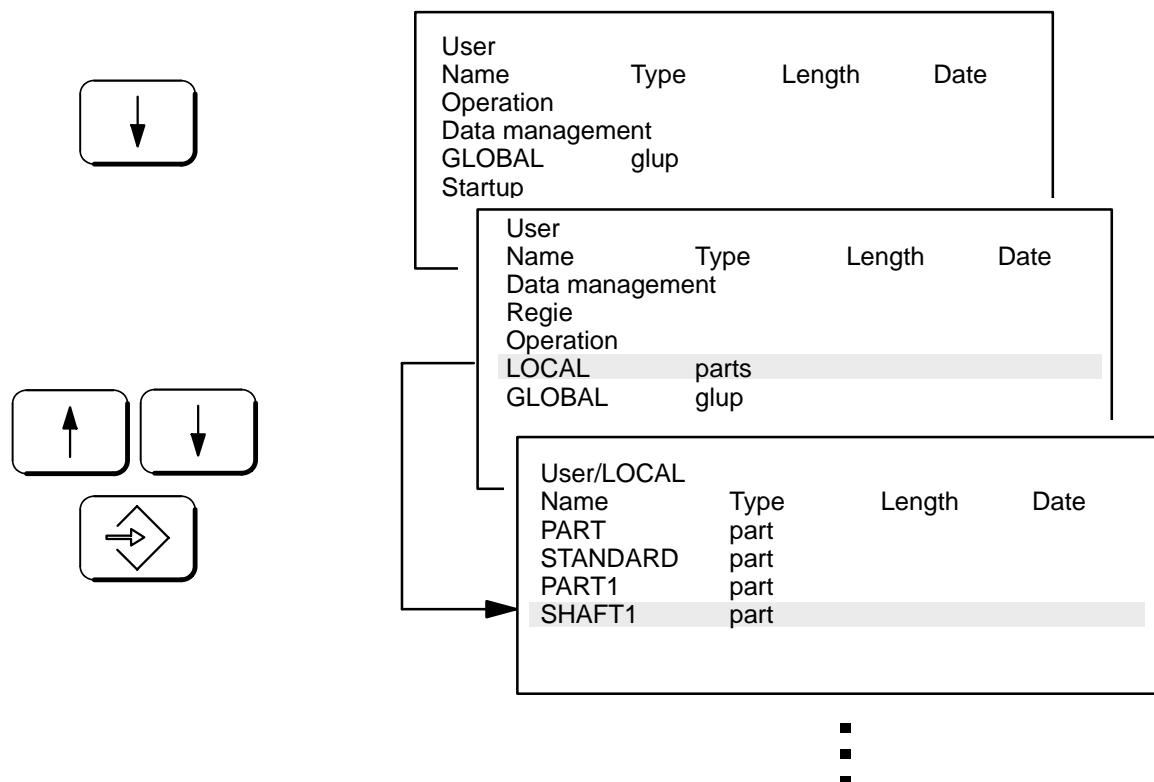
Fig. 6.15 Archive list interactive screenform

The display is divided into two windows. The top window displays the selected archive list.

If an archive list is newly opened, only the name of the archive list appears in the window.

You will find all of the directories and files of the user branch in the bottom "User" window.

Browse through the directories as follows until you find the file or directory to be recorded in the archive list:



Include

Press the INCLUDE softkey to include the previously selected element (directory, workpiece, files) in the archive list.

Save

Use the arrow keys and the INPUT key to search for the next element and enter it in the archive list with INCLUDE. Repeat the process until all of the directories and files have been included.

Remove

Press the SAVE softkey to save the archive list. The operating system automatically inserts the correct syntax for the archive list.

You can press the REMOVE softkey to delete individual files from the archive list. The files are displayed in the "Archive list" window as a path, e.g. PC/USER/LOCAL/SHAFT1. The file selected by the cursor is deleted.



Press the HOME key to switch to the "Archive list" window. The active window is highlighted by a yellow frame.



Use the arrow keys to select the file which is to be deleted by the REMOVE softkey.



Press RECALL to return to the directory of the archive lists.

## 6.8.1 Creating new archive lists

*Note*

The elements of the archive list are shown as directory paths. The type identifier is not displayed, e.g.:

PC/User/LOCAL/PART/MPF 13    part  
PC/User/Startup/Data/TEST    tea1.

**6.8.2    Editing existing archive lists***Precondition*

The SERVICES/DATA OUTPUT area is selected.

*Operating sequence*

Press the ARCHIVE LIST softkey.



Press the ETC key to extend the softkey bar. The softkeys NEW, EDIT, DELETE and COPY appear.



Use the arrow keys to select the archive list which you would like to modify from the "Archive list" window.



Press the EDIT softkey. The SAVE, REMOVE and INCLUDE softkeys provide you with a convenient means of modifying the archive list. The operator actions are described in the section "Creating new archive lists".

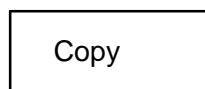


Press RECALL to exit the menu.

### 6.8.3 Edit, copy and delete archive lists

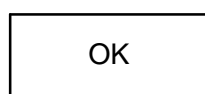
The COPY and DELETE softkeys provide you with a convenient means of copying and deleting complete archive lists.

Use the cursor keys to select an archive list in the archive list directory



Press the COPY softkey. The following dialog text is displayed:

Please enter name	CW500
-------------------	-------



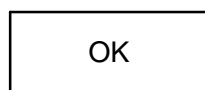
Enter the new name with the alphanumeric keyboard, acknowledge with the INPUT key and press the OK softkey. The copied archive list is stored in the directory.



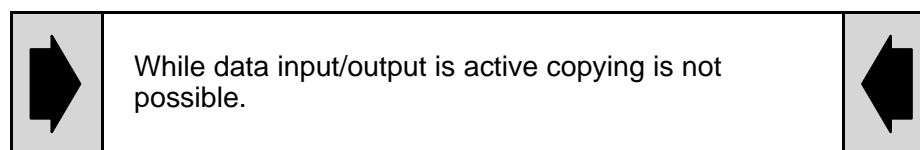
Use the cursor keys to select the archive list you wish to delete and press the DELETE softkey.

The following screenform is displayed:

Do you want to delete the data?	FZ10
---------------------------------	------



Press the OK softkey to delete the archive list.



## 6.9 Device settings (up to SW 6.2)

Via the RS232C(V.24) interface on the SINUMERIK 840C you can output files to an external data saving device. The RS232C(V.24) interface and the data backup device must match. The SINUMERIK 840C offers suitable input screenforms in which you can define specific data for your device. You can store several interface configurations in the control and activate them when required. The operating system offers in the Siemens branch the STANDARD and FLOPPY device settings. If you wish to modify these settings, first copy these files to the user branch.

### Precondition

The SERVICES/DATA IN-OUT area has been selected.

### Operating sequence

All of the existing interfaces are listed in the bottom window of the basic display of the SERVICES area. For standard configurations only the Serial1 interface is available. The Serial2 and Serial3 interfaces can be integrated as an optional extra (see the section "Data input and output with the FD-E2 floppy drive").



If several interfaces exist, use the arrow keys to select the interface you would like to configure.

Interface	Device	Status
Serial1	STANDARD	active
Serial2	PG_750	inactive *
Serial3	PG_685	inactive

Devices

Press the DEVICES softkey and the following is displayed:

Devices: Standard			
Interfaces			Serial 1
Name	Length	Date	
-			
FLOPPY	158	11-02-1994 09:46:52	
STANDARD	158	11-02-1994 09:46:52	
USB00A	158	11-02-1994 09:46:54	

Devices: User			
Interfaces			
Name	Length	Date	
-			
LAPTOP	158	11-08-1994 15:47:38	
PG730	158	11-08-1994 15:48:02	

Fig. 6.16 SIEMENS branch and USER branch for the device settings function

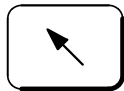


**Note**

- The selected interface (e.g. Serial1) is displayed in the standard device settings window.
- The \* identifier in the status line of an interface in the SERVICES basic display indicates that a transmission fault has occurred on that interface and an entry has been made in the error log. Reading the error log does not reset the error message. The identifier is deleted or set again if data input/output is tried again.

## 6.9.1 Selecting an interface

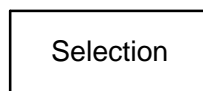
### Operating sequence



Use the HOME key to switch from the Siemens window to the user window and vice versa. The window which is active is marked.



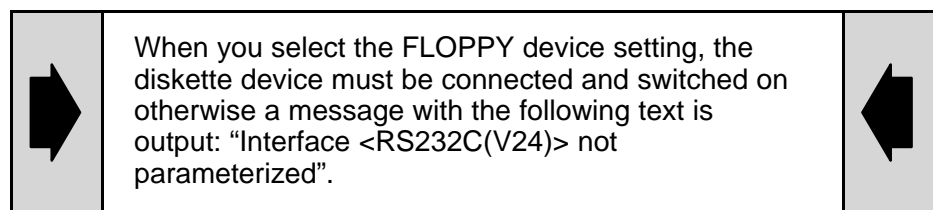
Use the cursor keys to select an interface in the user branch or in the Siemens branch (copy to the user branch "Editing interfaces" section).



Press the SELECTION softkey and the following dialog text is displayed:

Interface being parameterized!

After parameterization has been successfully completed the text disappears. The interface is selected and is displayed in the basic display.

**Note**

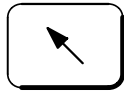
The interface remains selected after switching the control on or off.

## 6.9.2 Editing interfaces

### Preconditions

Interfaces for data input and output can only be edited in the user branch. This means that you must copy an interface configuration from the Siemens branch into the user branch first:

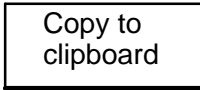
### Operating sequence



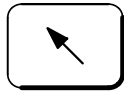
Press the HOME key to switch to the Siemens window.



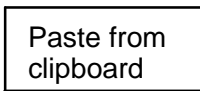
Select an interface (e.g. STANDARD)



and press the COPY TO CLIPBOARD softkey



Press the HOME key to switch to the user window

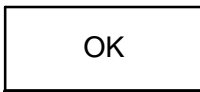


and press the PASTE FROM CLIPBOARD softkey; the following screenform is displayed:

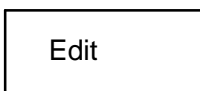
Please enter name	TREADER
-------------------	---------



Use the alphanumeric keyboard to enter a name and acknowledge with the INPUT key.



Press the OK softkey to accept the name in the user interface directory.



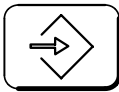
Use the cursor keys to select an interface in the user branch and press the EDIT softkey.

Machine	Parameter	Programm.	Services	Diagnosis	
					12:12
<b>Devices/Edit</b>					
PG740					
<b>Comments</b> Dies ist die standardnaessige Einstellung					
Device type	Universal	Leader, trailer	no		
Protocol	RTS/CTS	Ready	Yes		
Baud rate	9600	Stop at	End char.		
Stop bits	1	End of block	CRLF		
Parity	None	Beginning of output after XON	no		
Bits per character	8	Timeout in sec.	60		
XON char. (hex)	11				
XOFF char. (hex)	13	Transmission format			
End of transmission character (hex)	03	PC format			
store					

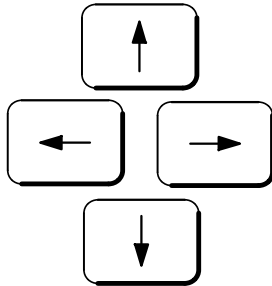
Fig. 6.17 Interactive screenform for editing an interface



Use the cursor keys to select the desired settings in the selection fields.



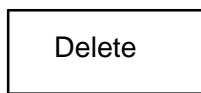
Enter the values in the input fields by means of the alphanumeric keyboard and conclude with the input key.



Use the cursor keys to move from one field to the next.



When you have completed editing, press the STORE softkey to store the setting on the hard disk.

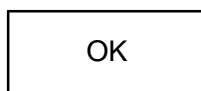


If you wish to delete a device setting in the user branch, select it with the cursor keys and press the DELETE softkey. The following screenform is displayed:

Do you want to delete the data?	FLOPPY
---------------------------------	--------



Press RECALL to cancel the process.



Use the OK softkey to delete the device settings from the directory.

#### Note

For data input via the RS232C (V24) interface, the timeout can be set as desired as from SW 5.6. The setting is not specific for the interface but applies to all serial interfaces. If data (e.g. part programs without end character) are to be sent to a serial interface over any period of time, the value 0 must be entered in the fields "Time in 100 ms between two characters" and "Timeout in sec.". Data input remains active until data with end characters are received or the ABORT softkey is pressed.

A value entered in the field "Time in 100 ms between two characters" becomes active when the corresponding interface is specified via the SELECTION softkey. After switching on and off of the control the standard value applies again.

## 6.9.3 Device settings (from SW 6.3)

In software version 6.3 and later, the operator interface has been modified in the SERVICES, DATA INPUT and DATA OUTPUT area. The operator interface for interfaces and device settings has been changed.

The selection is made in the SERVICES area.

Select

Interface	Device	Status
Floppy	FLOPPY	inactiv
Serial 1	STANDARD	inactiv
Serial 2	STANDARD	inactiv
Serial 3		

Fig. 6.18

The screenshot above shows the assignment between the interfaces (Floppy, Serial 1, Serial 2 and Serial 3) and the device (Floppy, Standard, WS800A) and, for example, a user-generated file New. The current setting is displayed on a grey background.

Softkey CHANGE DEVICES opens a screen in which the assignment between the interface and the device can be changed.

Change devices

The interface selected in 6.18 can be assigned to a device in screen Devices: User. The device is selected by means of the cursor keys.

Name	Length	Date
-		
FLOPPY	158	02-11-1999 08:08:14
<b>STANDARD</b>	<b>158</b>	<b>02-11-1999 08:08:14</b>
WS800A	158	02-11-1999 08:08:14
XON_XOFF	158	02-11-1999 08:08:16

Interface: Floppy  
Device: FLOPPY ' - ' not initialized!

Buttons: Delete, New, Selection

Fig. 6.19

Select

Press softkey SELECT to assign the interface selected in Fig. 6.18 to the device selected in Fig. 6.19.

New

Press softkey NEW to enter a name for the device to be created (e.g. New) in a subdialog.

The default setting for the serial 1 interface is "Standard" and for the floppy interface "Floppy".

Delete

Press softkey DELETE to delete device settings.

The current interface–device assignment is also displayed in screen De-  
vice settings.

Machine	Parameter	Programm.	Services	Diagnosis															
				14:28															
Device settings for interface: Floppy FLOPPY																			
Protocol	RTS/CTS	Leader, trailer	no																
Baud rate	9600	Ready	no																
Stop bits	1	Stop at	End char.																
Parity	Even	End of block	CRLF																
Bits per character	8	Beginning of output	no																
XON char. (hex)	17	after XON	10																
XOFF char. (hex)	19	Timeout in sec.																	
End of trans- mission ch. (hex)	03	Transmission format	PC format																
Data input: Target path																			
<table border="1"> <thead> <tr> <th>PC</th> <th>Name</th> <th>Type</th> <th>Length</th> <th>Date</th> </tr> </thead> <tbody> <tr> <td>Buffer</td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>User</td> <td></td> <td></td> <td></td> <td></td> </tr> </tbody> </table>					PC	Name	Type	Length	Date	Buffer					User				
PC	Name	Type	Length	Date															
Buffer																			
User																			
Into new workpiece	Archive contents	Floppy	Input Start	Buffer Error log															

Fig. 6.20

#### Notes

The device setting files (Floppy, Standard, WS800A) are automatically stored in the user branch during system power-up. These files can be deleted, but will be created again when the system next powers up.

The first interface (serial 1) is the only interface to be automatically initialized because it is part of the standard configuration. The user must "select" all other interfaces. However, the system then automatically initializes all previously selected interfaces with the appropriate devices files every time it powers up.

#### Device settings

Device settings for interfaces can only be made under "Data Input/Out-  
put". The device settings are stored with Output/Input Start.

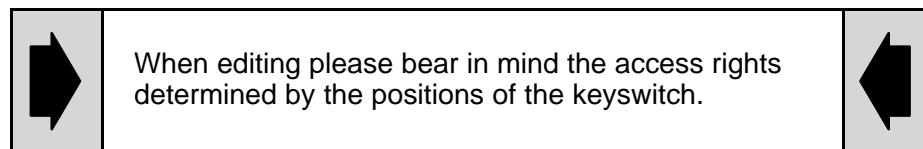
## 6.10 Data management

By means of the DATA MANAGEMENT function you can create directories and workpieces and copy and delete user data.

You can copy but not delete data from the SIEMENS branch.

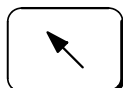
*Precondition*

You have selected the SERVICES area.



*Operating sequence* The "Data management" display is divided into two windows:

- Siemens and
- User



You can use the HOME key to switch between the two windows. The active window is highlighted by a yellow frame.

Data  
management

Press the DATA MANAGEMENT softkey; the following is displayed:

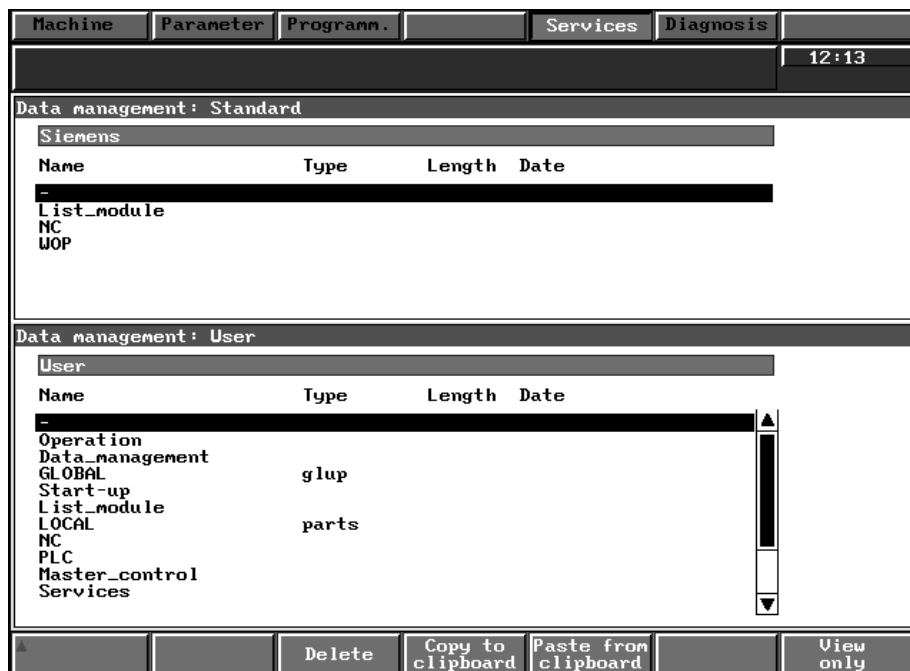
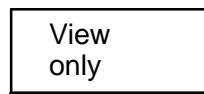


Fig. 6.21 SERVICES/MANAGEMENT area

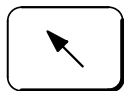
## 6.10.1 Copy and paste in the SERVICE/MANAGEMENT area

**6.10.1 Copy and paste in the SERVICES/MANAGEMENT area**

This view function calls a file previously selected. An ASCII editor is opened **without** the usual editor functions such as PASTE, CUT, etc. This function allows a file to be opened, viewed and checked before deleting or copying it without having to switch over to another operating area.

*Operating sequence*

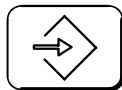
You can transfer data from the Siemens branch to the user branch; furthermore you can transfer data within the user branch (e.g. copy a part program from one workpiece into the other).



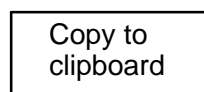
Use the HOME key to switch from the Siemens window to the user window and vice versa. The active window is marked (yellow frame).



Use the cursor keys and

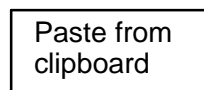


the INPUT key to page to the desired source directory: select the workpiece or the data with the cursor keys.



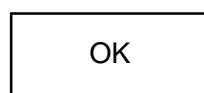
Press the COPY TO CLIPBOARD softkey to mark the file/workpiece as source for copying.

Use the cursor keys and the INPUT key to page to the directory in which you wish to copy the selected data.

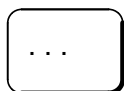


Press the PASTE FROM CLIPBOARD softkey; you are requested to enter the file name:

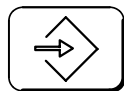
Please enter name	STANDARD
-------------------	----------



Press the OK softkey if the name of the source file and the copied file are to be identical.



If the copied file is to be named differently from the source file, enter the new name via the alphanumeric keyboard and conclude with the INPUT key.

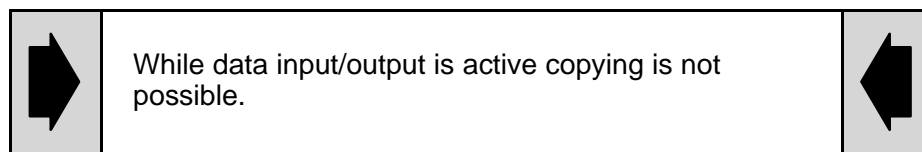


Press RECALL to cancel the process





Press the OK softkey to copy the file into the target directory.

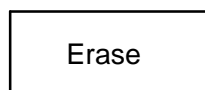


## 6.10.2 Delete data in the SERVICES/MANAGEMENT area

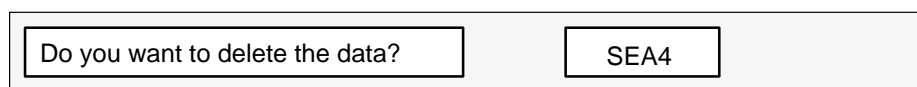
### Operating sequence



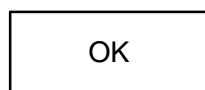
Use the cursor keys to select a data directory in the user branch and



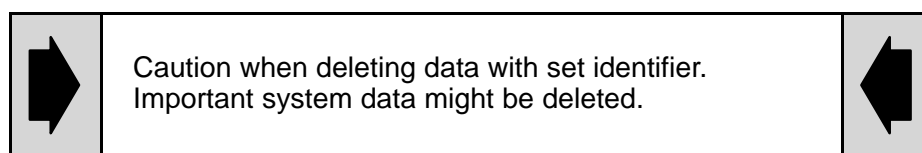
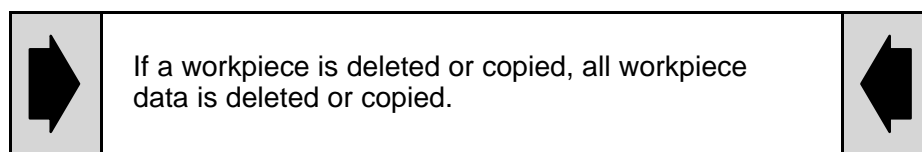
press the ERASE softkey. The following dialog text appears:



Press RECALL to cancel the process.



If you press the OK softkey, the selected data is irretrievably deleted from the hard disk.



END OF SECTION



## 7 Diagnosis

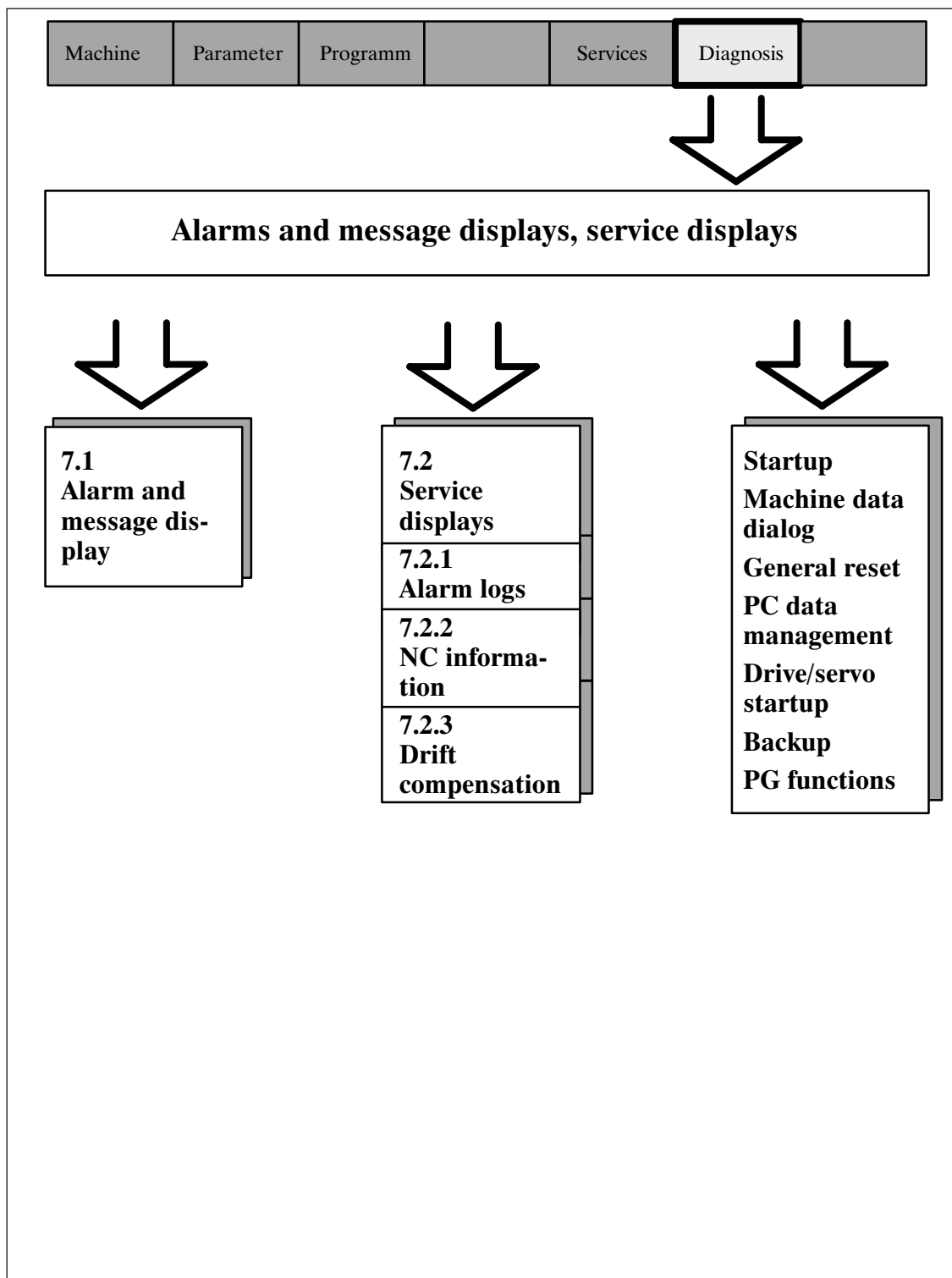


Fig. 7.1 Diagnosis

## 7.1 Alarm and message displays

## 7.1 Alarm and message displays

Diagnosis

The DIAGNOSIS area is selected by pressing the softkey in the area menu bar. The basic display shows a list of the current alarms. It is possible to switch over to the display of current messages.

Display by  
priority

These softkey functions allow you to display existing alarms/messages according to their priority or the order in which they occurred.

Display by  
time

Press this softkey to display a list of the current messages.

Messages

Alarms

Press this softkey to switch to the alarm display.



Fig. 7.2 Basic display of the DIAGNOSIS area



The majority of the functions in the DIAGNOSIS area are intended for startup or servicing. A description of these functions, which are protected by password against unauthorized access, is given in the "SINUMERIK 840C Startup Guide". Please refer to the SINUMERIK 840C Diagnostics Guide for a detailed list of alarms and their possible causes.



## 7.2 Service displays

### 7.2.1 Alarm logs

Service  
display

Press the SERVICE DISPLAY softkey in the basic display of the DIAGNOSIS area.

Alarm  
log 1

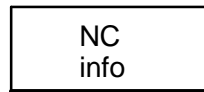
The machine tool manufacturer defines which alarms are displayed in alarm logs 1 and 2. The alarms logs are in a circulating buffer which means that the oldest entries are overwritten by new alarms. The latest entry can be found above the dotted line. The machine tool manufacturer can configure the size of the circulating buffer.

Alarm  
log 2

Machine	Parameter	Programm.	Services	Diagnosis
1040 ORD5 X speed setpoint value warning limit responded				
Reset				
Alarm log 1				
10:58.17	10:58.21	110004	No data selected	
10:59.10	10:59.12	110121	Option not available	
11:00.25	11:00.27	110121	Option not available	
11:03.07	11:03.09	110004	No data selected	
11:08.25	11:08.27	110011	There is no error log	
11:14.18	11:14.22	110002	Name <TEIL1> already exists	
11:16.36	11:16.40	110063	No workpiece can be created here	
11:20.47	11:20.50	110022	No workpiece selected	
11:24.11	11:24.26	110019	Observe error log	
11:25.49	11:26.00	110037	The hard disk is full	
11:54.41	11:54.44	110004	No data selected	
12:14.14	12:14.46	110037	The hard disk is full	
12:15.29	12:15.37	110037	The hard disk is full	
12:24.08	12:24.08	106002	OPTIONEN OPT is being read ...	
12:24.17	12:26.29	1120	ORD1 X zero speed control	
12:26.53	12:27.03	1040	ORD2 X speed setpoint value warning	
12:26.53	12:27.03	1560	ORD3 X speed set val. alarmlimit res	
12:27.35		3004	1 N0 error in CL 800 programming	
12:27.58	12:28.11	1121	ORD4 Y zero speed control	
12:28.14		1040	ORD5 X speed setpoint value warning	
12:28.14		1560	ORD6 X speed set val. alarmlimit res	
-----				
10:53.21	10:53.23	110063	No workpiece can be created here	
10:54.26	10:54.31	110019	Observe error log	
(\$25 *1)				
Format				
Line number				
Copy to clipboard				
Search				
Display again				

Fig. 7.3 Alarm log 1

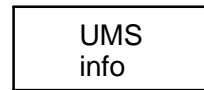
## 7.2.2 NC information



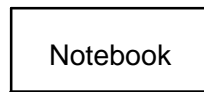
If you press the NC INFO softkey an information menu is displayed showing the NCK software version and the type of control. This information is important for servicing.

### *Note*

As from software version 5, the sum software version will also be included as the first entry in the log book (softkey Diagnosis/Start-up/Log book).



Press the UMS INFO softkey for information on the contents of the UMS (languages, menus, cycles etc.).



The NOTEBOOK function is a servicing and startup function and is protected by password.

## 7.2.3 Drift compensation

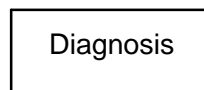
If the < or > display remains visible after a traversing movement when the axes no longer move, the drift has exceeded the value defined in MD 204\* and MD 208\* and drift compensation must be executed.

Drift compensation is performed semi-automatically and is activated via softkey.

### *Preconditions*

- The < or > display (position not reached) remains visible after a traversing movement.
- The NC axes and the drives are controlled (closed-loop control).
- The axes are stationary.

*Operating sequence* Press the following softkeys one after the other



DIAGNOSIS



SERVICE DISPLAY

As from SW 6.3, the safety software version is displayed in the log book.

NC  
service

NC SERVICE

Axis  
single


AXIS SINGLE

Drift comp.  
axis 1...

DRIFT COMPENSATION AXIS 1 etc.  
Drift compensation is performed for the selected axis.


Machine	Parameter	Programm.	Services	Diagnosis
				15:53
JOG		Program reset		Mode Gp 1 Channel 1
Service axes single display			Axis: 1	
Following error 7 Absolute actual value 113778 Absolute setpoint value 113785 Abs. compensation val. 0 Speed set value (0.01%) 6 Part actual value 0 Part set value 0 Contour deviation 4 Error synchronism 0 Parameter set pos. control 1 Parameter set for ratio 1 Service number 0				
<div> <div>Further axis/spind</div> <div>Axis single</div> <div>Spindle single</div> <div>Drift comp axis 1</div> <div>Drift comp axis 2</div> <div>Drift comp axis 3</div> <div>Drift comp axis 4</div> </div>				

Fig. 7.4 Drift compensation axis 1 in the DIAGNOSIS area



**Caution!**

In the case of high-precision machines drift compensation should be performed several times a day due to temperature differences during operation since the drift is added to the following error.



### 7.3 Softkeys for the start-up engineer in the DIAGNOSIS basic display

Language	The start-up engineer or service specialist can change the control language with the softkey LANGUAGE.
Password	The password is activated with the softkey PASSWORD.
PG function	Pressing softkey PG function starts program Step5–MT (stage 6) (option).
<i>Preconditions</i>	Step5–MT is installed on the hard disk. For on-line operation interfaces X151 on the MMC CPU and X111 on the PLC CUP must be connected with a standard RS232 cable (e.g. 6FC 9340–8W).
Service display	Access to information about alarms, NC, PLC and drive is accessible with softkey SERVICE DISPLAY, see Section 7.2 “Service displays”.
Start-up	The (first) start-up engineer can configure the control with softkey START-UP.

END OF SECTION



## 8 Maintenance

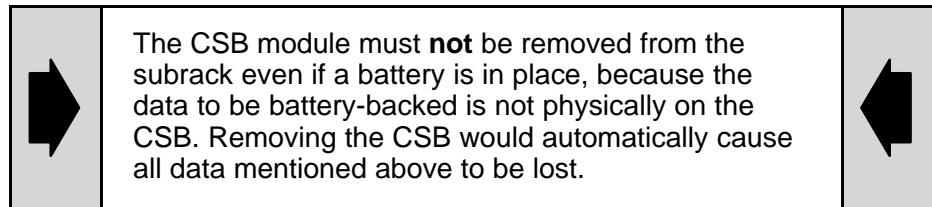
### 8.1 Operating data

Humidity in accordance with DIN 40040	F
Air pressure	860 to 1080 hPa
Shock protection, safety class in accordance with DIN VDE 0160	I
Type of protection in accordance with DIN 40050	
• Front of operator panel	IP54
• Rear of operator panel	IP 00
• Front of machine control panel	IP 54
• Rear of machine control panel	IP 00

Complete data on use and operation can be found in the documentation “Interface Part 2, Connecting Conditions” and in the relevant “Loose-leaf Special Instructions”.

## 8.2 Replacing the battery

There is a backup battery (6LR61, 550mA, 9V) on the central service board (CSB) 6FC5114-0AA02-0AA0. The following data is backed up with this battery voltage: PLC program, RPA, TOA, SEA, SEA4, ZOA TEA.

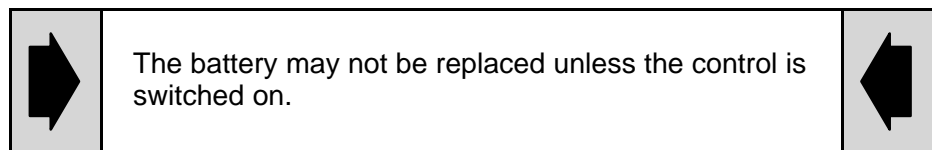


The battery backup voltage is monitored by the control.

Monitoring is performed in two steps:

*Warning 1st step* Advance warning if the battery voltage falls below 6V, Alarm 101 "Advance warning battery replacement" is triggered.

*Warning 2nd step* Alarm 1 "Data lost at power off", the threshold value is 4.8 V.  
If the 2nd step is triggered, the battery must be replaced immediately.



### Battery replacement

- Store and transport the battery in its packaging.
- Before fitting the battery, check it over for damage. Do not use any battery that shows signs of damage.

### Procedure

- Undo the screw in the battery cover.
- Carefully remove the cover with battery.
- Using a screwdriver, push the battery out of the cover.
- Detach the contact lug from the battery.
  - On CSB with product version D and higher:  
Lithium batteries (Varta type 6127 503 074)
  - On CSB up to product version C:  
Alkaline batteries (type 6LR61, 550 mAh)

**Note**

On product version D of the CSB, the operating points of the battery monitoring system have been adjusted to the lithium battery.

When a CSB of product version C is combined with a lithium battery or a CSB of product version D with an alkaline battery, the advance warning period may be less than 4 weeks or the data support time less than 1 year if the ambient conditions (temperature) are not ideal.

- Push battery back into battery cover
- Screw battery cover back onto CSB

Weaker batteries may also be used for short periods. However, this results in a shorter data support time and increases the risk of battery leakage.

**Caution**

Batteries must never be thrown into fires.  
Batteries must never be taken apart.  
Used batteries must be disposed of separately from other waste (special waste disposal). The national regulations valid in the country of installation must be observed.

## 8.3 Handling modules

Nearly all the SINUMERIK modules have LSI chips. Owing to their technology, these are very sensitive to overvoltage and thus also to electrostatic discharge.

Modules using these components are identified on the component side by this warning symbol.



**ELECTROSTATICALLY ENDANGERED COMPONENTS** can be destroyed by voltages and energy levels well below those perceived by the human body.

Such voltages are present even if a component or module is touched by a person who has not undergone electrostatic discharge.

Components exposed to such overvoltages are not immediately recognizable as defective in most cases since faulty behaviour sets in only after they have been operating for a relatively long time.

Therefore, please pay attention to the following guidelines:

- Electronics modules must not be touched unless it is absolutely vital for the work to be done.
- Before touching a module, any static in the body must be discharged. The easiest way to do this is by touching a conductive, earthed part of the structure (e.g. bare metal parts on the cabinet, socket earth contact) immediately beforehand.
- Modules must not be brought into contact with materials that are good insulators (e.g. plastic foil, insulated desk tops, clothing made from synthetic fibres).
- Modules must be placed on conductive surfaces only.
- Modules must be plugged in or unplugged only when at zero potential.
- The power supply must be switched on before any signal voltages are applied.
- Modules and components must always be stored and carried in conductive packing (e.g. metalized plastic boxes).
- If the packing is not conductive, the modules must be wrapped in conductive material prior to packing. For example, conductive foam rubber or household aluminium foil can be used.

## 8.4 Practical tips on remedying electromagnetic compatibility problems

**Electromagnetic compatibility (EMC)** means the capacity of an electrical unit to work properly in a given electromagnetic environment without affecting the environment in a manner which is not acceptable.

Faulty behaviour	Possible cause	Elimination of fault
Flickering screen characters or display surface	Adjacent unit producing a magnetic field (e.g. transformer, electric motor)	Increase distance between CRT and source fault (>300 mm). In extreme cases, shield CRT with MUMETALL.
The information on the CRT and the background information have colours missing over some areas	Magnetic field of an adjacent permanent magnet	Remove cause of fault and screen with MUMETALL. <b>Note:</b> Effect on screen will only disappear a few days after removal of the cause.
Sporadic failure	• No suppression elements for contactors	• Provide suppression elements for contactors (coils)
	• Adjacent machines producing sparks (welding sets)	• Lay cable to the control differently (increase distance to cause of fault)
	• Radio transmitter, centralized multi-station control system	• Additional screening
	• Cable with wrong screen connection, cable with wrong wire combinations or wrong characteristics	• Use original cable and check core assignment
	• Interruption in cable screening (e.g. if a cable distributor is interposed)	• Connection of cable screening, screening the point of interruption (e.g. the cable distributor)
	• Bonding conductor laid wrongly	• Re-lay bonding conductor
	• Control dirty	• Clean control (and modules) and ensure that cleaner air is supplied
Permanent axis offset	See possible causes as for 'Sporadic failure'.	See elimination of fault as for 'Sporadic failure'. The actual value cable acts as the fault sink.

## 8.4 Practical tips on remedying electromagnetic compatibility problems

Faulty behaviour	Possible cause	Elimination of fault
Rough irregular machining surface	<ul style="list-style-type: none"> <li>Poor equipotential bonding between position encoder and control</li> </ul>	<ul style="list-style-type: none"> <li>Increase cross section of bonding conductors (between control and position encoder), connect bonding conductor directly to the control (not to separate voltage bus)</li> </ul>
	<ul style="list-style-type: none"> <li>High equalizing current to setpoint cable screening when screen is connected to housings on both sides</li> </ul>	<ul style="list-style-type: none"> <li>Interrupt screen connection on the control side</li> </ul>
	<ul style="list-style-type: none"> <li>Bonding conductor between SINUMERIK and drive control laid too far from setpoint cable</li> </ul>	<ul style="list-style-type: none"> <li>Shorten distance between bonding conductor and setpoint cable</li> </ul>
	<ul style="list-style-type: none"> <li>Setpoint input on the drive control does not match SINUMERIK EMC concept</li> </ul>	<ul style="list-style-type: none"> <li>After input protective circuit on the drive control (in accordance with protective system of Siemens drive controls)</li> </ul>

## 8.5 Cleaning

The front of the monitor and the surface of the operating panel 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.

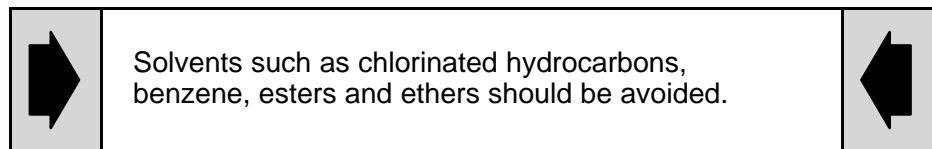
These cleaners will also remove dirt containing graphite.

Cleansing agents which contain one or more of the following ingredients can be used for a short period of time:

- Diluted mineral acids
- Bases
- Alcohol
- Organic hydrocarbons
- Detergent solutions.

The plastic materials used on the front of the SINUMERIK 840C are suitable for application of the control on machine tools. They are suitably resistant to

- Grease, oil, mineral oil
- Bases and lyes
- Detergent solutions and
- Alcohol.



END OF SECTION



## 9 List of Abbreviations

Abbreviation	Meaning
ACR	Axis/Spindle Converter = Axis/spindle converter, see Program control
ASCII	American Standard Code for Information Interchange
BCD	Binary Coded Decimal
BRK	Breakpoint (program modification)
CLR	Clear, delete blocks (program modification)
COR	Coordinate Rotation
CPU	Central Processing Unit
CRC	Cutter Radius Compensation
CSB	Central Service Board
CTS	Clear to Send
DAC	Digital-to-analog Converter
DB	Data Block
DIO	Data Input/Output
DRAM	Dynamic RAM, dynamic read/write memory
DRF	Differential Resolver Function
DRY	Dry Run
DSB	Decoding Single Block
DSR	Data Send Ready
DW	Data Word
EIA code	Special tape code; No. of punched holes per character is always odd
EPROM	Erasable Programmable Read Only Memory
ETC	ETC key; Extension To Text Line
EXT	Execution from external, see Program control
FIFO	First in First Out
FST	Feed Stop
GEO	Geometry
GIA	Gearbox interpolation
HRS	High Resolution Measuring System
HW limit switch	Hardware Limit Switch
IKA	Interpolation and compensation with tables
INC	Increment
JOG	Jogging
KYRU	Key Request Unit
K1...K6	Channel (max. 6 channels with SINUMERIK 840C)

Abbreviation	Meaning
LED	Light Emitting Diode
LF	Line Feed
M01	Conditional stop, see Program control
MD	Machine Data
MDA	Manual Data Automatic
MMC	Man Machine Communication (PC area SIN 840C)
MPF	Main Program File
NC	Numerical Control
NCK	Numerical Control Kernel
NCY	No Start/end cycle = Program Start/end cycles off, see Program control
OPI	Operator Panel Interface
PLC	Programmable Logic Control
PST	Procedure step, program step (program modification)
RAM	Random Access Memory
ROV	Rapid Override
RPA	R Parameter Active, NCK memory for R parameter numbers
RTS	Request to Send
SAV	Auto-Save = Automatic saving, see Program control
SBL	Single Block
SEA	Setting Data Active, NCK memory for setting data
SD	Setting Data
SKP	Skip
SPF	Sub Program File
SW limit switch	Software Limit Switch
TEA	Testing Date Active, refers to machine data
TNRC	Tool Nose Radius Compensation
TO	Tool Offset
TOA	Tool Offset Active, memory for tool offsets
TRC	Tool Radius Compensation
UMS	User Memory Submodule
ZO	Zero Offset
ZOA	Zero Offset Active, memory for zero offsets

END OF SECTION

SIEMENS AG  
A&D MC BMS  
P.O. Box 3180  
D-91050 Erlangen  
Federal Republic of Germany  
Tel. ++49 - 180 / 5050-222 [Hotline]  
Fax ++49 - 9131 / 98-2176  
email: motioncontrol.docu@erlf.siemens.de

<p><b>From</b></p> <p>Name _____</p> <p>Company/Dept. _____</p> <p>Address _____</p> <p>_____</p> <p>Telephone: _____ / _____</p> <p>Telefax: _____ / _____</p>	<p><b>Suggestions</b></p> <p><b>Corrections</b></p> <p>For Publication/Manual</p> <p>SINUMERIK 840C Software Version 6</p> <p>User Documentation</p> <p>Operator's Guide</p> <p>Order No.: 6FC5198-6AA00-0BP2 Edition: 09.2001</p> <p>Should you come across any printing errors when reading this publication, please notify us on this sheet. Suggestions for improvement are also welcome.</p>
---	---

**Suggestions and/or corrections**

# Overview of SINUMERIK 840C Documentation / OEM Version for Windows

