SIEMENS

SINUMERIK 840C
Software Version 6

Operator's Guide   **09.2001 Edition**

User Documentation

SIEMENS

SINUMERIK 840C

Software Version 6

Operator's Guide

Valid for

Control   *Software Version*
SINUMERIK 840C/CE
(Standard/Export version)   6

09.2001 Edition

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	**A**
06.93	6FC5198–2AA00–0BP0	**C**
12.93	6FC5198–3AA00–0BP0	**C**
10.94	6FC5198–4AA00–0BP0	**C**
03.95	6FC5198–4AA00–0BP1	**C**
09.95	6FC5198–5AA00–0BP0	**C**
04.96	6FC5198–5AA00–0BP1	**C**
08.96	6FC5198–5AA00–0BP2	**C**
07.97	6FC5198–6AA00–0BP0	**C**
01.99	6FC5198–6AA00–0BP1	**C**
09.01	6FC5198–6AA00–0BP2	**C**

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

Edition	Order No.	Remarks
10.01	6FC5 198–6CA00–0BG2	C

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.

Other functions not decribed 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.

SINUMERIK 840C (BA)

Contents

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

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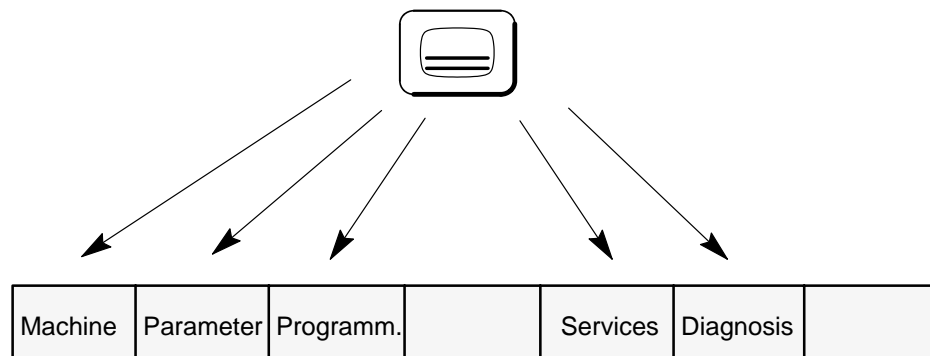
Fig. 1.1 shows you which operator actions are possible in the different
operating areas. With the SINUMERIK 840C, you can edit the part
programs in two different memories:

• 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.

Operator panel

Operating areas

Machine	Parameter	Programming	Services	Diagnosis

Operating mode selection
• JOG
– Set-up mode
– Approach reference point
– Incremental traversing
– REPOS
– Handwheel
– Man. traversing
– Overstore
• TEACH IN
– Set-up mode
– Incremental traversing
– REPOS
– Handwheel
– Overstore
– Program modification
– Search
• MDA
– Overstore
– Program modification
• AUTOMATIC
– Overstore
– PRESET
– Program modification
– Search
– Workpiece selection
	• Program
	  parameters
	– Tool
	  offset
	– Zero
	  offset
	– Angle of rot.
	– R parameters
	• Setting data
	– Working area
	  limitations
	– Spindle data
	– Scale
	– General data
	  and bits
	– Axial bits
	– Position
	  signals
	– Axis/spindle
	  converter
	– Gearbox
	  interpolation
	– Travel against
	  fixed stop
	• Computer
	  link (option)
	• User
	  function
	  (option)
		• Editing and
		  generating
		  part programs
		– Editing in the
		  NCK area
		– Editing on the
		  hard disk
		• Generating
		  workpieces
		• Delete,
		  copy and
		  insert
		  workpieces
		  and files
		• Creating
		  directories and
		  files
		• Generating
		  job lists
		• Graphic
		  programming
		  system WOP
		  (option)
		• DIN simulation
		  (option)
		• Graphic tool path
		  simulation
			• Load NC
			– Load data from
			  the hard disk
			  into the
			  NCK memory
			• Save NC
			– Save data
			  from the
			  NCK memory
			  to the hard disk
			• Data in/out
			– RS232C (V24) in/out
			– Printing data
			– Generating and
			  editing archive
			  lists
			– Device settings
			– Clipboard
			• Data management
			– Copy, delete and
			  insert directories
			  and files
				• Alarm
				  displays
				• Start-up
				  functions
				– NC diagnosis
				– PC diagnosis
				– PLC diagnosis

RAM memory

• Part programs and IKA3
  data on volatile memory
• All other NC data buffered

NCK

SAVE

LOAD

• Storing data on the hard
  disk with:
– SAVE NC softkey in the
  SERVICES area
– SAVE TO HARD DISK or
  SAVE UNDER softkey

MMC–CPU

• Interface control
  to the machine

PLC

Input/output
machine
control

SINUMERIK 840C (BA)

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:
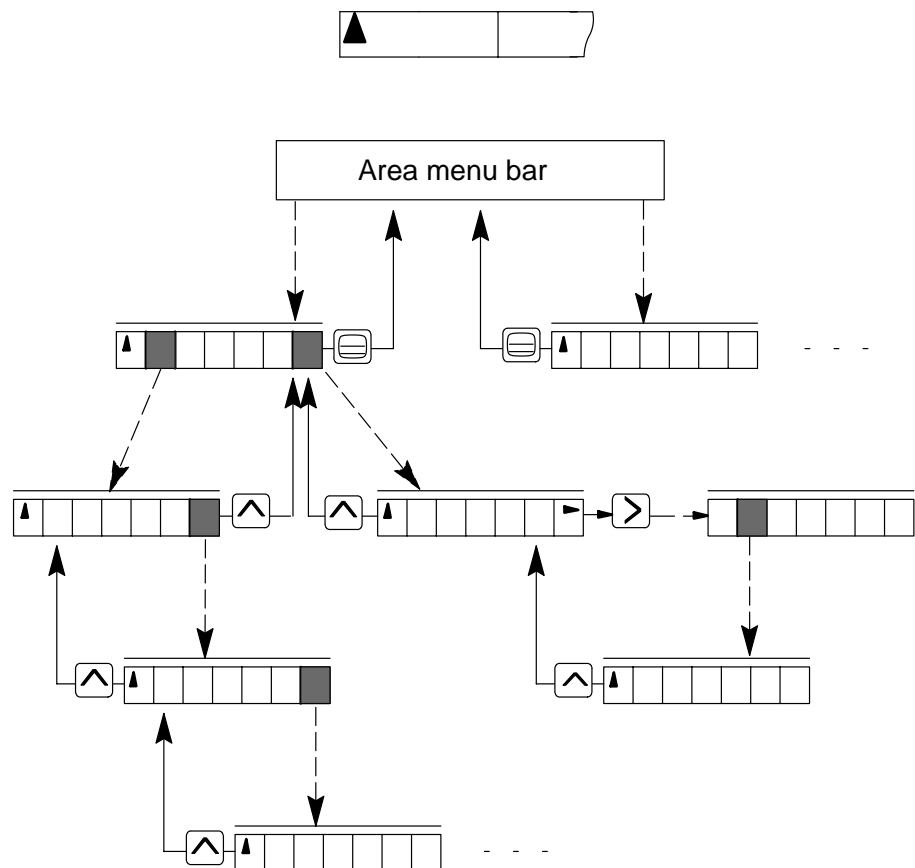
With the ETC key, you can expand the softkey menu if an extension is available.
If there is an arrow in the 7th softkey window

 (on the right), this key is active.

In each area, the menu last active is stored on area switchover (exception: MACHINE area).
If you leave an area and do not wish to return to the menu level currently selected, you switch over to the main menu of the area with RECALL. This brings you to the defined initial status of this area. This procedure simplifies operation when first selecting the area. By pressing the SHIFT and RECALL keys you will obtain the basic display immediately from any branch in the PARAMETER, SERVICES, DIAGNOSIS and PROGRAMMING areas.

When leaving the area the control "remembers" the deselected area. This area is selected again by pressing the area switchover key twice.

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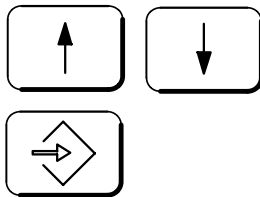
Data selector   A program that interprets your keyboard entries (cursor, input) as a file
                     or directory selection, displays it on the screen (bar) and passes it on to
                     other programs (e.g. editor).

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.

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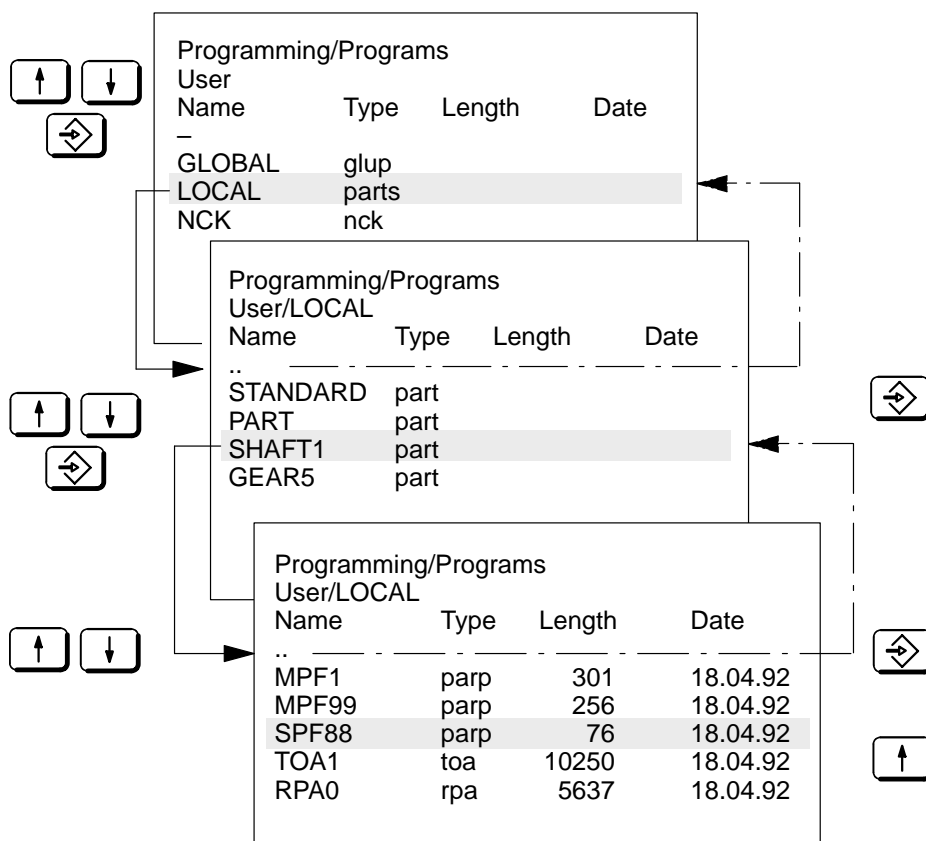
Fig. 1.1   Representation of the directory path PROGRAMMING/LOCAL/SHAFT1/SPF88

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	sycnf	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	mdd
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)	mmcasm
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	kbtab	Status data (GIA)	gia
Kinematics	kine	System	ncks

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		

> 250 directories or files can be created for each directory.
> e.g.:   Directory LOCAL   : 250 workpieces
>   Directory GLOBAL   : 250 workpieces
>   Directory SHAFT1   : 250 files
>   etc.

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:
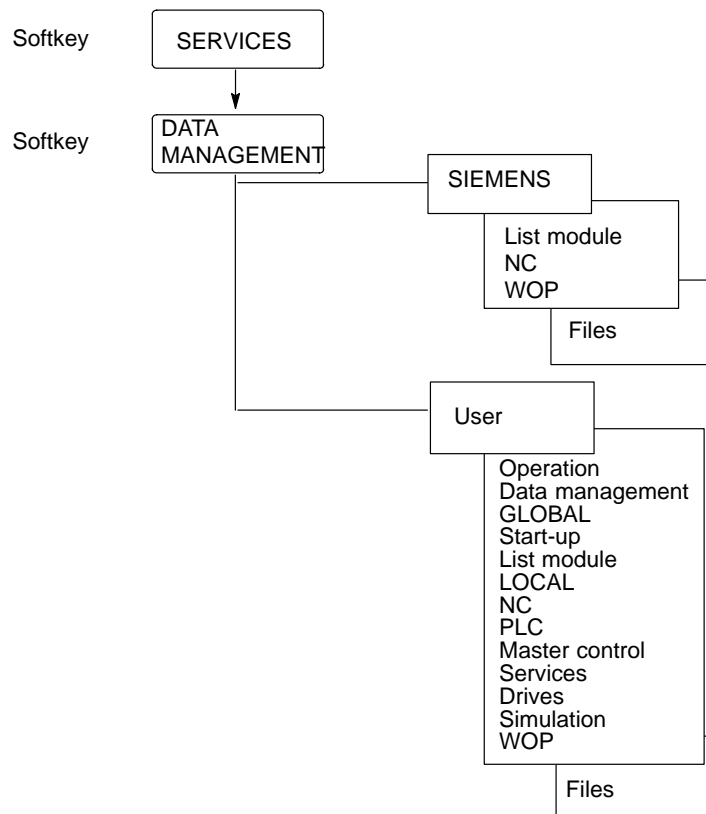
```
Softkey      ┌─────────────┐
             │  SERVICES   │
             └─────────────┘
                    │
                    ▼
Softkey      ┌─────────────┐
             │    DATA     │
             │ MANAGEMENT  │──────────┬──────────────┐
             └─────────────┘          │   ┌──────────────┐
                                      │   │   SIEMENS    │
                                      │   └──────────────┘
                                      │        ┌──────────────┐
                                      │        │ List module  │
                                      │        │ NC           │
                                      │        │ WOP          │
                                      │        └──────────────┘
                                      │             ┌──────────┐
                                      │             │  Files   │
                                      │             └──────────┘
                                      │
                                          ┌──────────────┐
                                          │    User      │
                                          └──────────────┘
                                               ┌──────────────────┐
                                               │ Operation        │
                                               │ Data management  │
                                               │ GLOBAL           │
                                               │ Start-up         │
                                               │ List module      │
                                               │ LOCAL            │
                                               │ NC               │
                                               │ PLC              │
                                               │ Master control   │
                                               │ Services         │
                                               │ Drives           │
                                               │ Simulation       │
                                               │ WOP              │
                                               └──────────────────┘
                                                    ┌──────────┐
                                                    │  Files   │
                                                    └──────────┘
```

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.

Caution!
All internal processes which generate screen outputs do not cancel screen darkening. This also applies to alarms and operational messages.
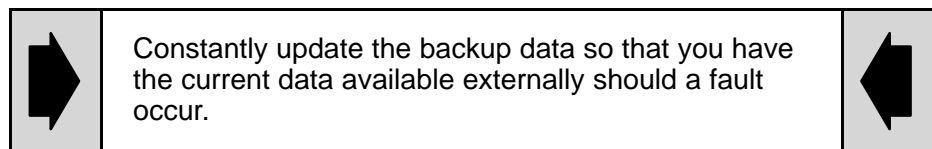
1.7 Data backup options

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.

> Constantly update the backup data so that you have the current data available externally should a fault occur.

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:
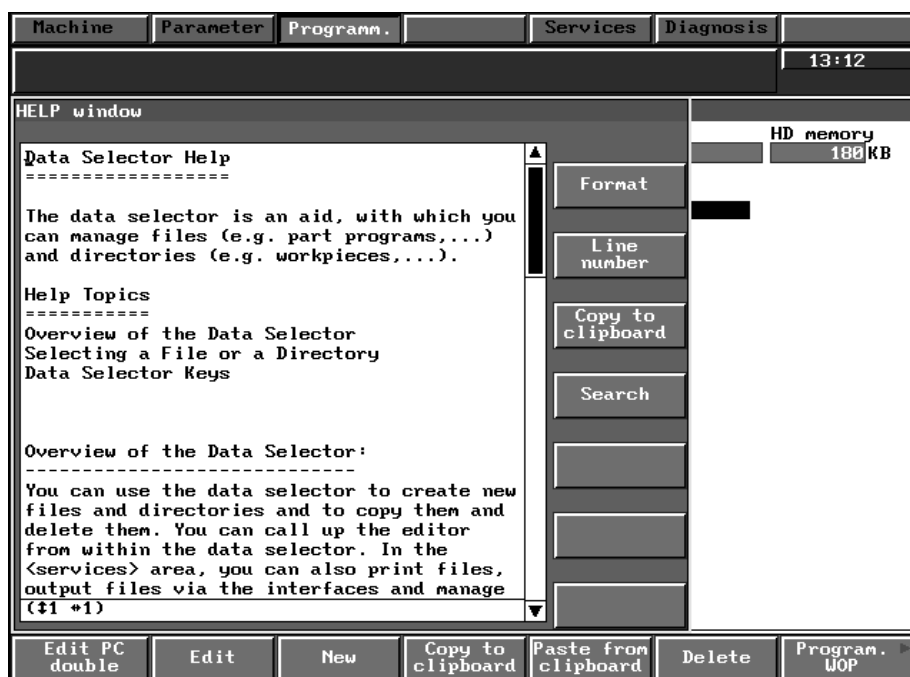

Fig. 1.3   Help display

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
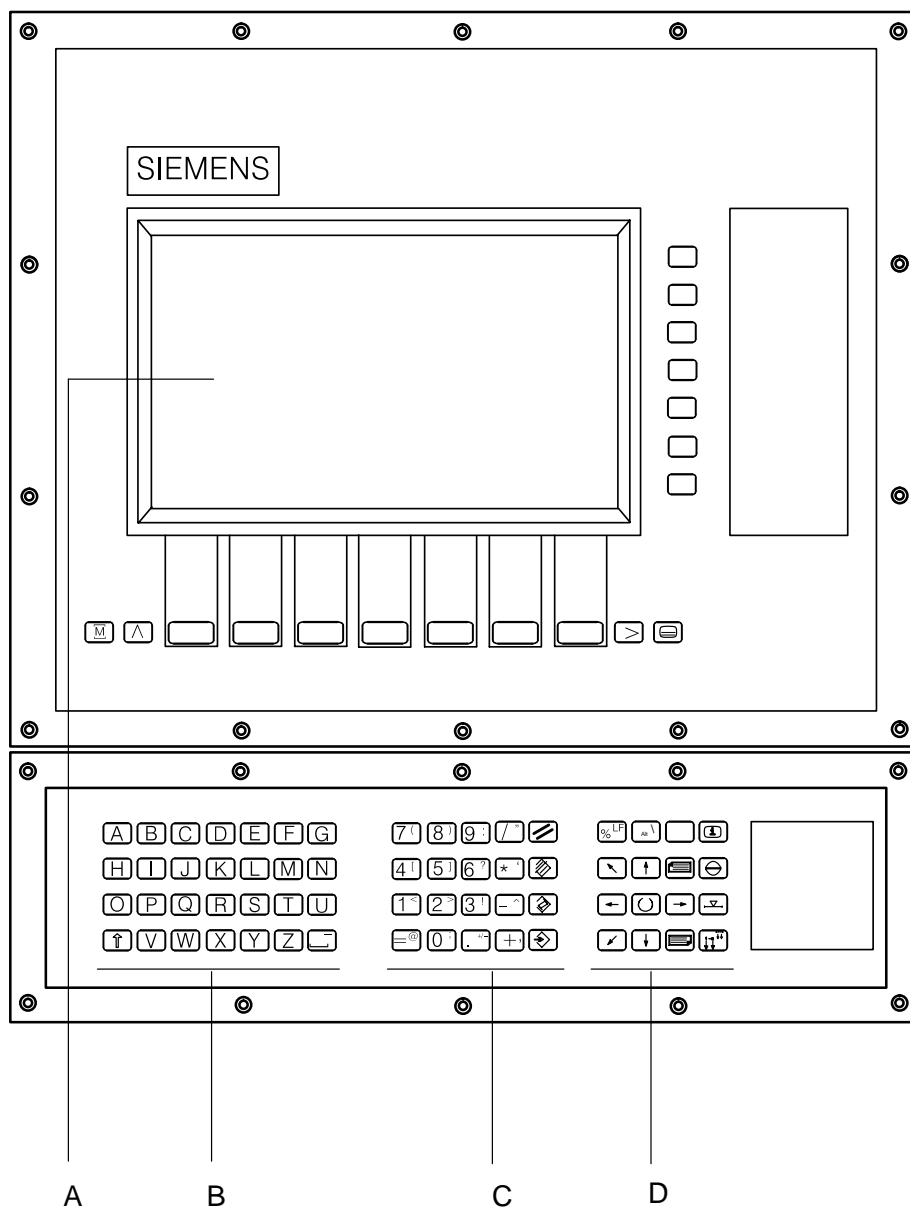
Fig. 2.1   19" operator panel with 14" colour monitor
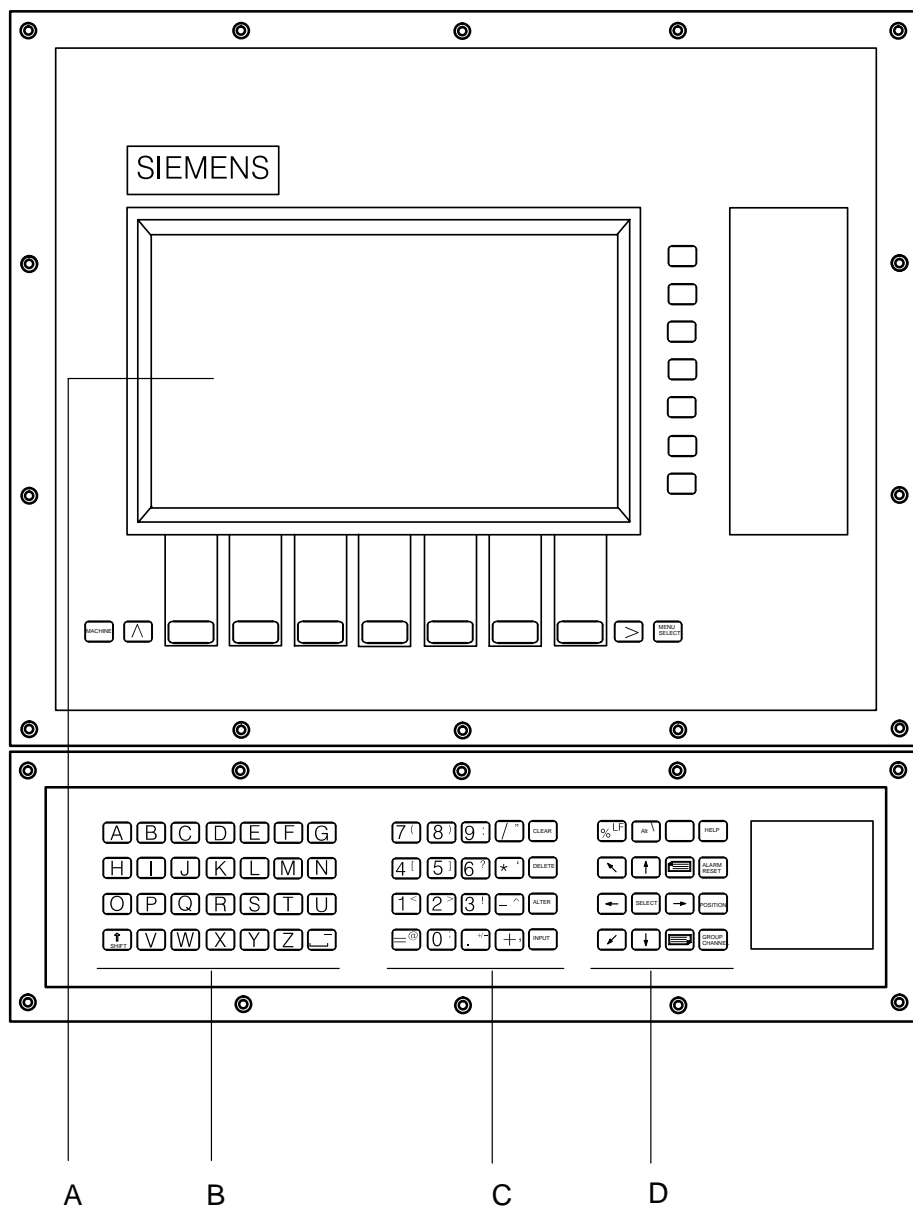
Fig. 2.2   19" operator panel with 14" colour monitor in US layout
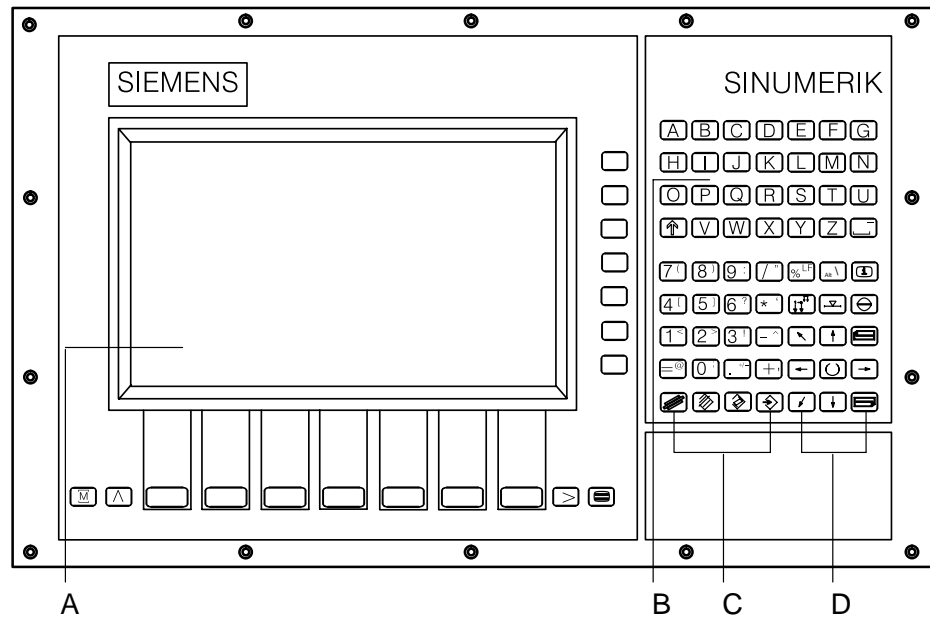
2.2.1   Operator panel

Fig. 2.3   SINUMERIK slimline operator panel
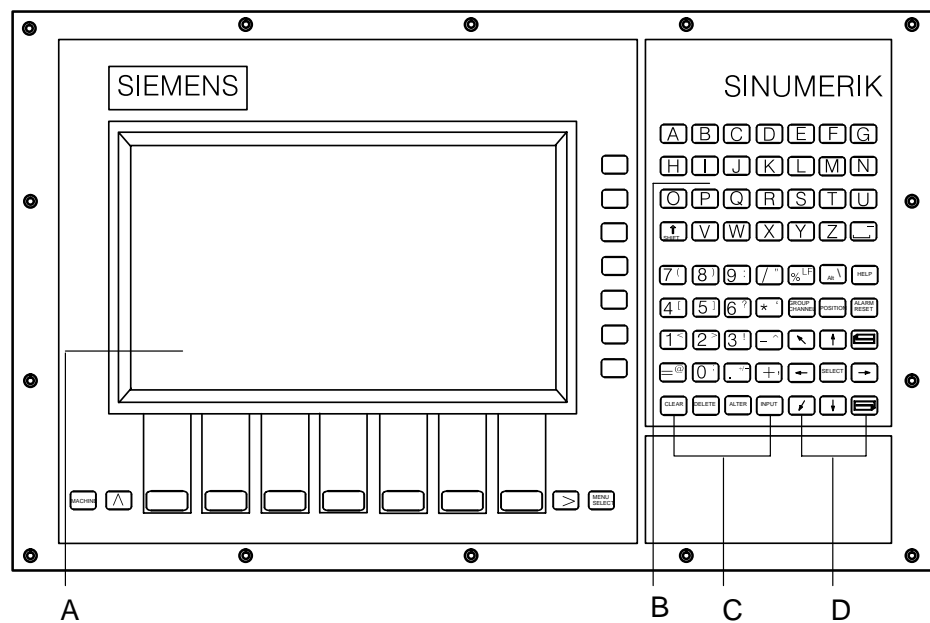
Fig. 2.4   SINUMERIK slimline operator panel in US layout

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.

> The full standard keyboard does not comply with the conditions (noise immunity) of a SINUMERIK control; therefore it should be used only for start-up and during servicing. The MF2 keyboard is connected to the operator panel interface.
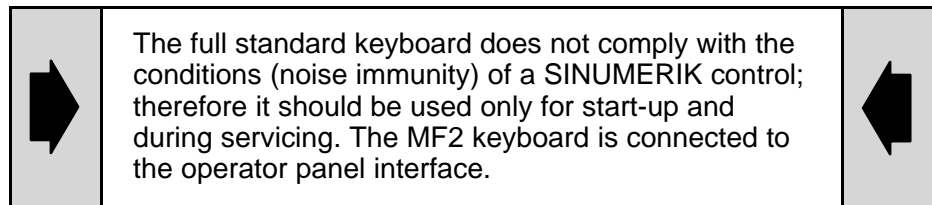
2.2.2   PC standard keyboard (MF–2)
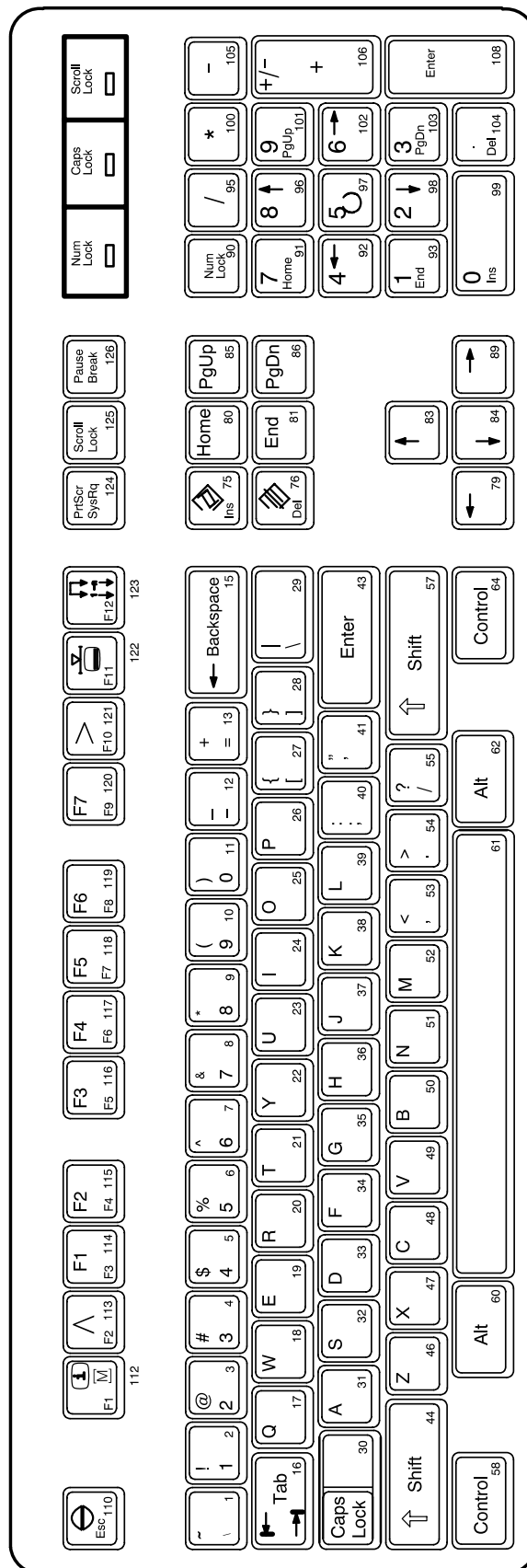
Fig. 2.5   Full keyboard

SINUMERIK 840C (BA)

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)
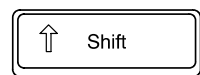
Vertical softkey bar (full keyboard)
The designation F1 to F7 corresponds to the designation on the additional key caps of the standard keyboard
(see also Figure "Full keyboard").
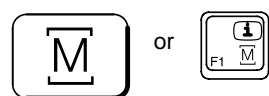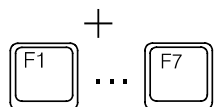
Machine area

RECALL key: Return to the higher level menu
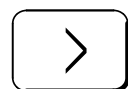
ETC key: Extension of the softkey bar in the same menu

1) 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.

1) 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
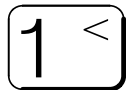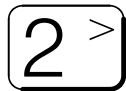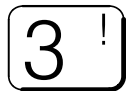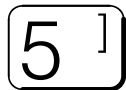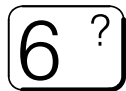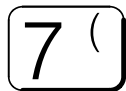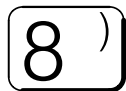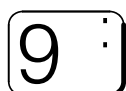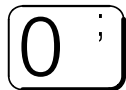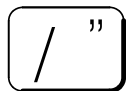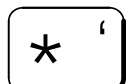
Underscore
Blank, space

1) US layout

2.2.3   Operator panel keyboard / full keyboard

$\boxed{1\ ^<}$	<u>Special character: less than</u> Digit 1
$\boxed{2\ ^>}$	<u>Special character: greater than</u> Digit 2
$\boxed{3\ ^!}$	<u>Exclamation mark</u> Digit 3
$\boxed{4\ ^[}$	<u>Square bracket</u> Digit 4
$\boxed{5\ ^]}$	<u>Square bracket</u> Digit 5
$\boxed{6\ ^?}$	<u>Question mark</u> Digit 6
$\boxed{7\ ^(}$	<u>Round bracket</u> Digit 7
$\boxed{8\ ^)}$	<u>Round bracket</u> Digit 8
$\boxed{9\ ^:}$	<u>Colon</u> Digit 9
$\boxed{0\ ^;}$	<u>Semicolon</u> Digit 0
$\boxed{/\ ^"}$	<u>Inverted commas</u> Oblique stroke, division
$\boxed{*\ ^'}$	<u>Apostrophe</u> Multiplication

Special character
Hyphen, subtraction
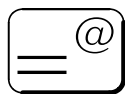
Special character
Equal sign
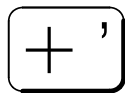
Comma
Plus, addition
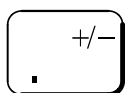
Change of sign
Fullstop (point/period)

or

Shift + +

1) Delete character/entry

or

← Backspace
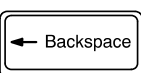
1) Delete word

1) Edit word
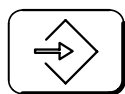
1) Confirm input (store an edited value in the memory)

or

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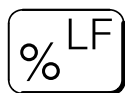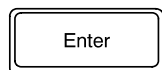
Line Feed; identifier for end of block
Percent sign; identifier for beginning of main program
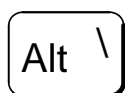
Backslash
Alt

1) US layout

Help

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

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).
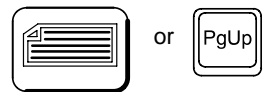
You "page" up by one display.
You change the current screen display by operating one of these keys if further screen displays of the same layout (lists) are available.
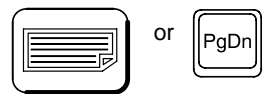
You "page" down by one display. In a part program you can page down the display (towards the end of the program), or up (towards the beginning of the program).

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

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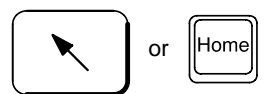
Change to another input window
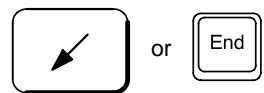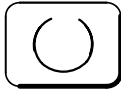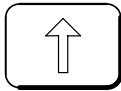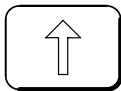
Activate/deactivate cursor in the input line
With this key you activate the cursor in the input line (NCK only), move it by means of the cursor keys, in order to correct or insert. When operating the key again, you deactivate the cursor.
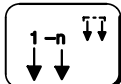
_____

1) US layout

1) 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

1) 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.

1) 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.

---

1) US layout

2.2.4   The operating elements of the machine control panel

Machine control panel for a milling machine
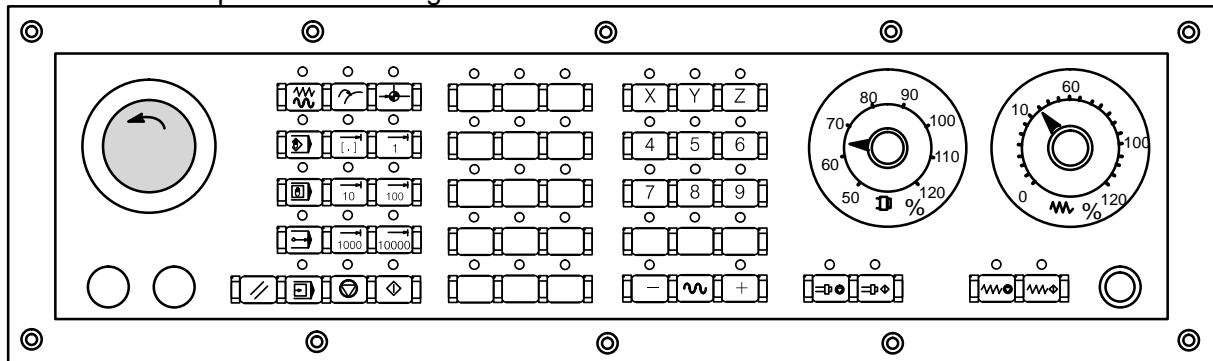
Machine control panel for a milling machine in US layout
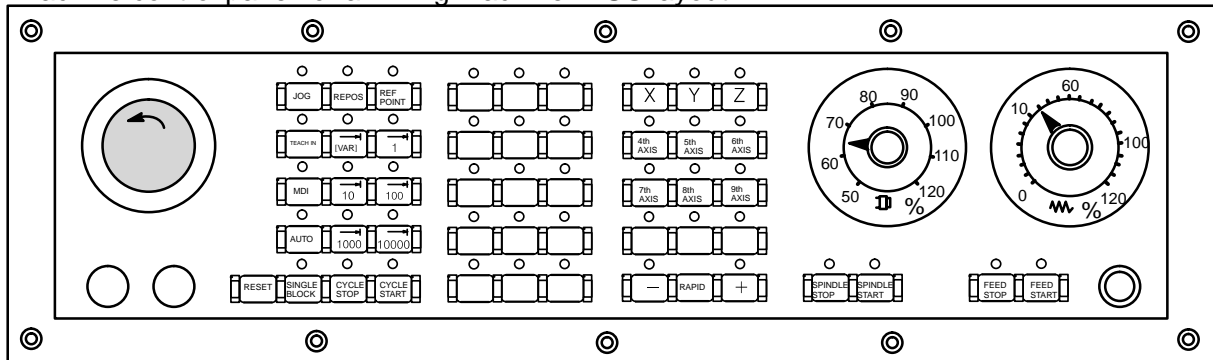
Machine control panel for a turning machine
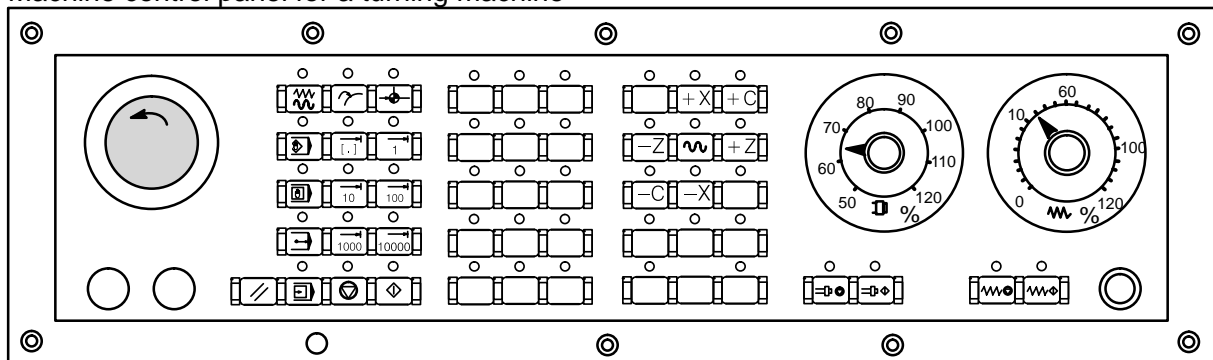
Machine control panel for a turning machine in US layout

Fig. 2.6   Machine control panels

SINUMERIK 840C (BA)

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
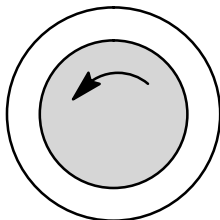
You operate the red button in emergency situations:

- 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.	

Operating modes

Key symbol	Definition	Designation of operating mode
JOG [1]	The axes are traversed continuously in JOG mode using the direction keys or incrementally using the direction keys or the handwheel.	**JOG** Set-up (**Jog**ging)
TEACH IN [1]	Interactive creation of programs	**TEACH IN**
MDI [1]	The machine is controlled by executing a block or a series of blocks. The operator panel is used to input blocks.	**MDA** **M**anual **D**ata **A**utomatic
AUTO [1]	The machine is controlled by automatic execution of programs.	**AUTOMATIC**

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
REPOS [1]	Repositioning, approach contour again in **JOG** mode	**REPOS** **Repos**ition
REF POINT [1]	Approach reference point in **JOG** mode	**REFPOINT** Approach reference point
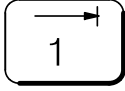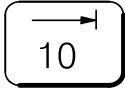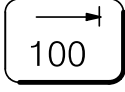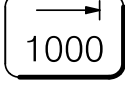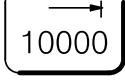
The "REPOS" and "Approach reference point" functions are only active in JOG mode and can be selected only in JOG mode.

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.

---

[1] US layout

INC function

Key symbol	Definition	Designation of function
[.] [VAR] 1)	Increment mode with variable step size (setting data).	**INC VAR** Incremental Feed variable
1	Increment mode with fixed step size of 1 increment	**INC** Incremental Feed
10	Increment mode with fixed step size of 10 increments.	
100	Increment mode with fixed step size of 100 increments.	
1000	Increment mode with fixed step size of 1000 increments.	
10000	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
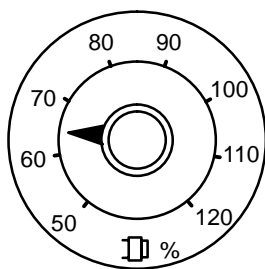
- The rotary switch with 16 notched positions (of which 15 are used) enables you to reduce or increase the programmed spindle speed S (corresponds to 100%).

- 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

---

1) US layout

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

SPINDLE STOP [1)]

When you press the SPINDLE STOP key:
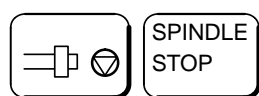
- The spindle speed is reduced down to zero
- The associated LED lights up as soon as SPINDLE STOP is accepted by the control.

Example for use of SPINDLE STOP

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

Spindle start

SPINDLE START [1)]

When you 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 associated LED lights up as soon as SPINDLE START is accepted by the control.

The following values are specified by machine data or setting data:

- The maximum spindle speed
- The values for spindle override positions

(refer to the machine tool manufacturer's documentation)

Feedrate control

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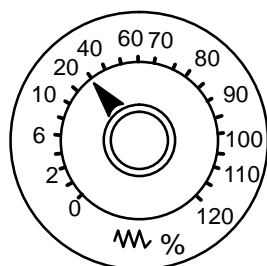
The rotary switch with 23 notched positions allows you to reduce or increase the programmed feedrate value F (corresponds to 100%).

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%

1) US layout

┌─────┬──┬─────┐
│ ⚙   │ 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.   │   │
└─────┴──┴─────┘

Feed stop

〰 ⊘ │ FEED │ [1)]
      │ STOP │

When you operate the FEED STOP key:

- 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

〰 ◇ │ FEED │ [1)]
      │ START │

When you operate the FEED START key:

- 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.

┌─────┬──┬─────┐
│   │ **The following is configured by the machine │   │
│   │ manufacturer:**   │   │
│   │   │   │
│ ⚙   │ - The speed and rapid traverse rates   │ ⚙ │
│   │   │   │
│   │ - The values for feedrate and override positions │ │
│   │   │   │
│   │ - Whether the feedrate override switch is also │   │
│   │   active for rapid traverse   │   │
│   │   │   │
│   │ - The axis names   │   │
│   │   (see machine manufacturer's specifications) │   │
└─────┴──┴─────┘

Direction keys, turning machine

│ +X │ │ −X │   You traverse the axis marked X.

│ +Z │ │ −Z │   You traverse the axis marked Z.

│ +C │ │ −C │   You traverse the axis marked C.

─────────────────────────────

1) US layout

Rapid traverse override

  ¹⁾ 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*

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

 ¹⁾

---

1) US layout

     6FC5198–☐AA00
SINUMERIK 840C (BA)

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

[1]   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

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

Single block

SINGLE BLOCK [1)

This function allows you to execute a part program on a block-by-block basis.

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

CYCLE STOP [1)

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

CYCLE START [1)

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.

The axis positions programmed in the current block are approached with linear interpolation after the "NC START" key has been activated.
Caution: Risk of collision!

---

1) US layout

2.2.5   Operating elements on the hand-held unit

> 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.
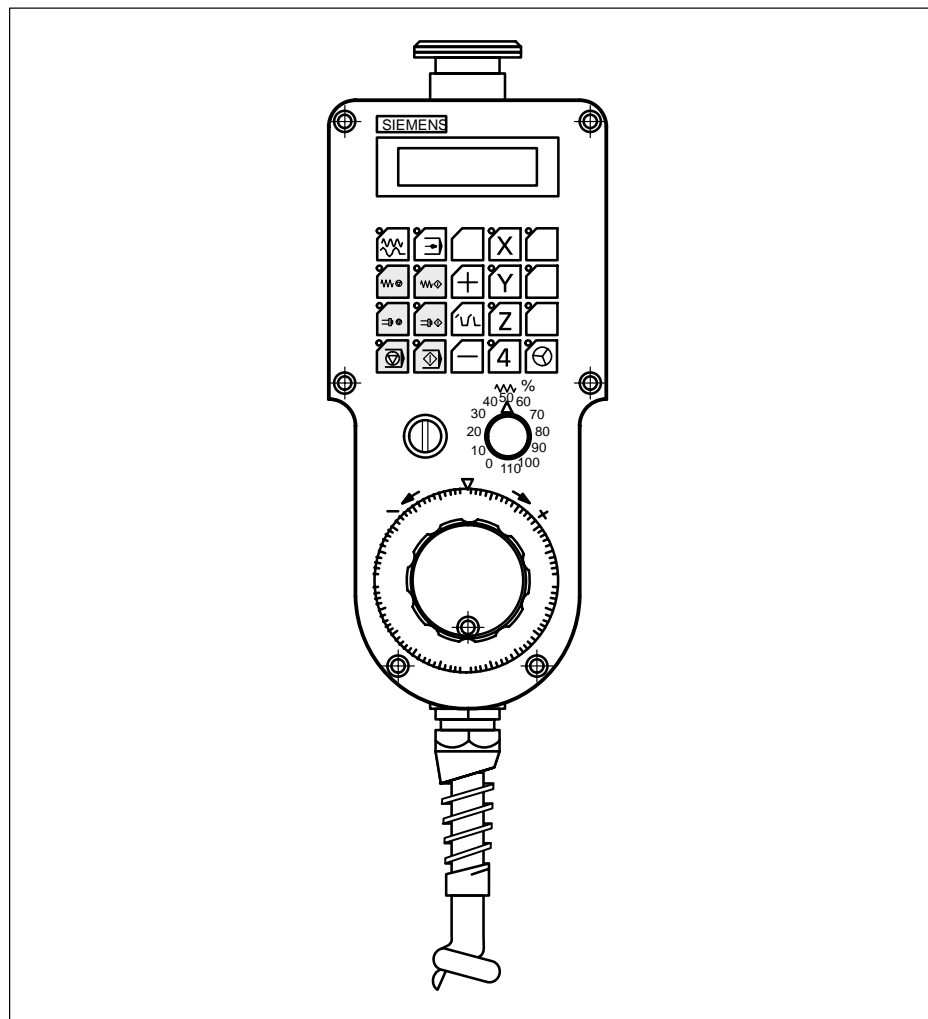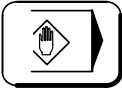
Fig. 2.7   Operating elements of the hand-held unit

*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	**JOG** set-up (**Jog**ging)
	Creating programs interactively with the machine	**TEACH IN**

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
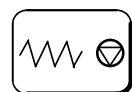
When you press the FEED STOP key:

- 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.

	The machine manufacturer configures the following:	
	• The feed and rapid traverse velocities	
	• The values for the feedrate override settings	
	• Whether the feedrate override switch is also active for rapid traverse	
	• The axis names (see information supplied by machine-tool manufacturer)	

Spindle stop
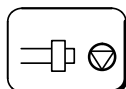
Press the SPINDLE STOP key:

- 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
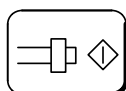
Press the SPINDLE START key:

- 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.

	The following is defined in the machine data or setting data:	
	• The maximum spindle speed	
	• The values for the spindle speed override settings (see information supplied by the machine-tool manufacturer)	

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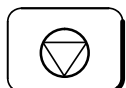
If you press the NC STOP key processing of the current part program is interrupted. All axis movements are brought to a controlled stop. H and M function responses on NC STOP are configured by the machine-tool manufacturer. Please refer to the documentation supplied by

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

NC Start
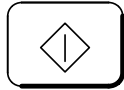
Press the key NC START to start the part program you have called at the current block; the corresponding LED lights up. After interrupting processing of a part program with NC STOP, processing is continued with NC START at the point of interruption.
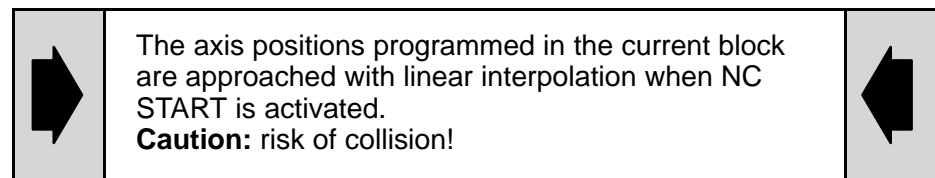
> The axis positions programmed in the current block are approached with linear interpolation when NC START is activated.
> **Caution:** risk of collision!

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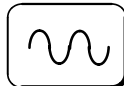
When you operate this key at the same time as any of the keys above, the axis is traversed in rapid traverse mode.

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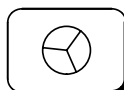
This handwheel function must be configured by the machine-tool manufacturer.

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
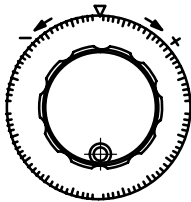
2.2.7   Screen layout
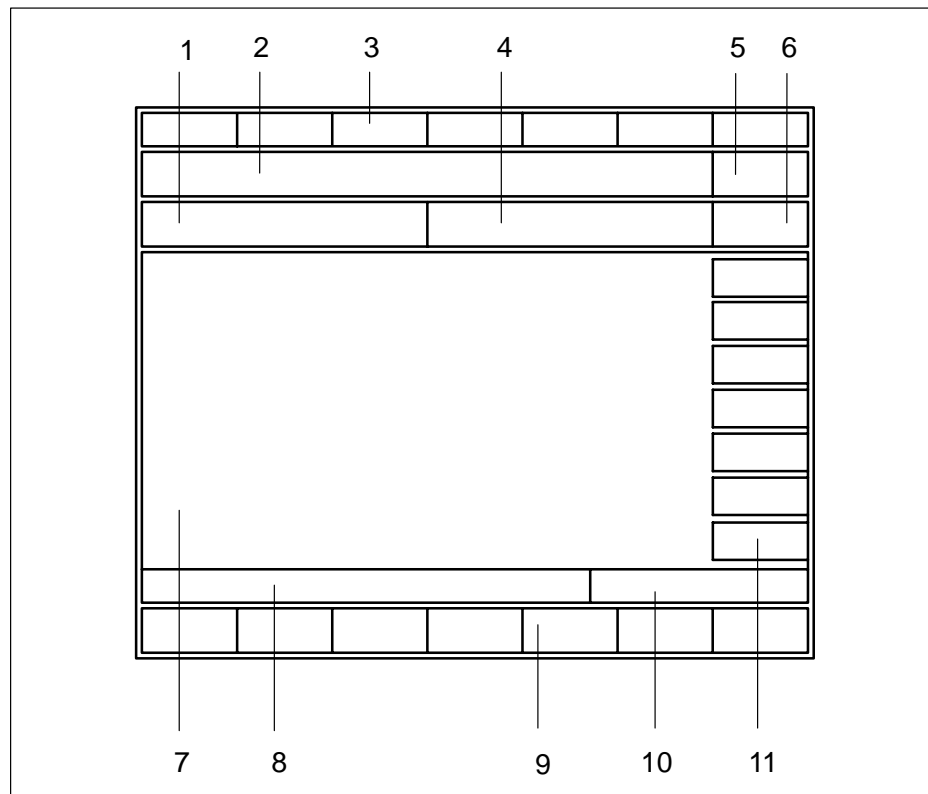
Fig. 2.8   Screen layout

*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

SINUMERIK 840C (BA)
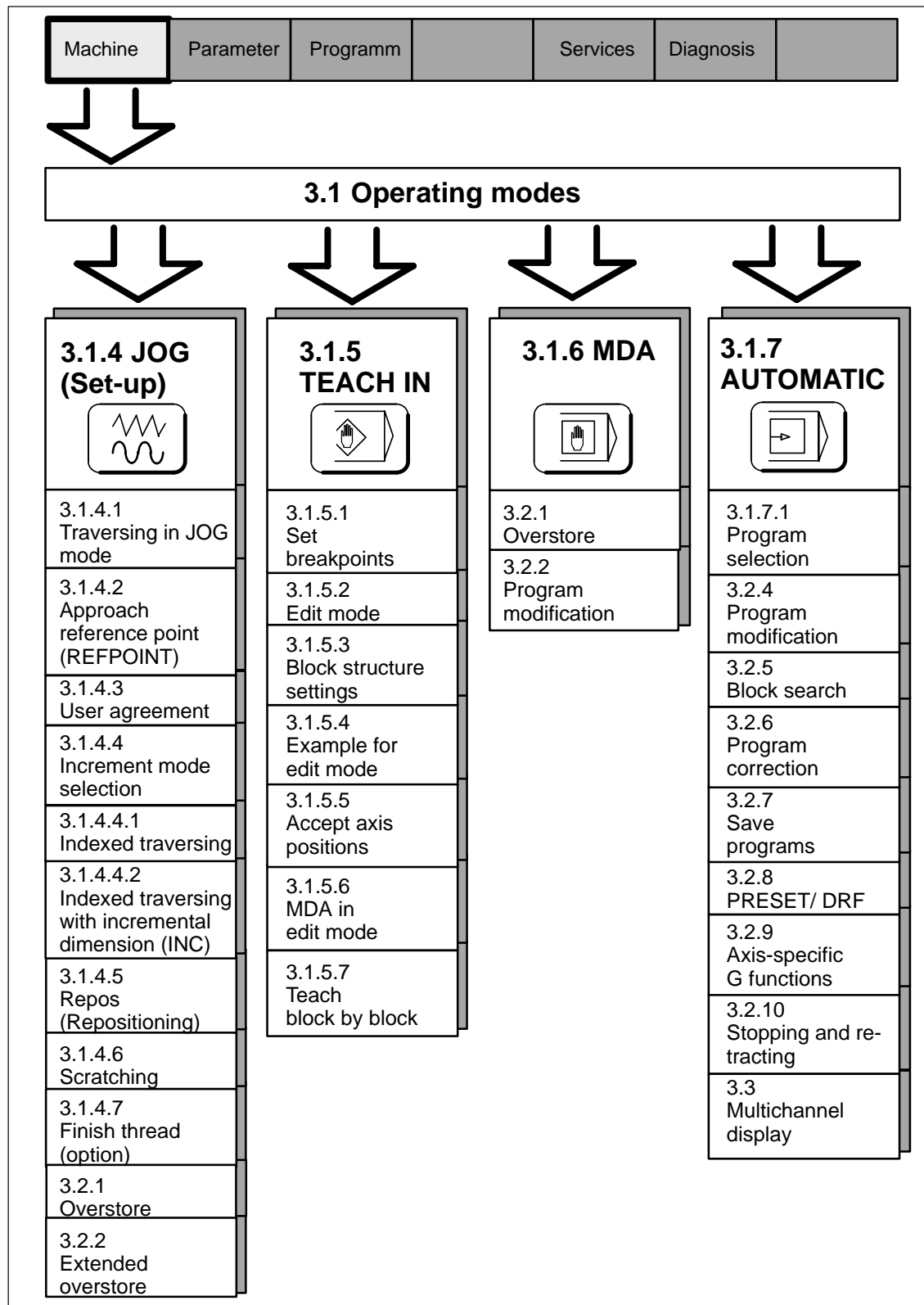
3 Machine

Fig. 3.1   Machine
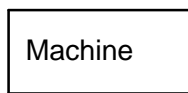
Machine

If you activate the MACHINE softkey in the area menu bar

M

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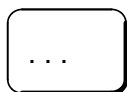
If you have two mode groups, and if each is assigned to a machine control panel, you select the operating mode by pressing the mode key on the appropriate machine control panel.

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.

> The machine tool manufacturer's guide provides information on:
>
> • Number of mode groups
>
> • Number of machine control panels
>
> • Assignment of machine control panels to the mode groups.

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		M.grp : 1
Program reset		Channel : 1

Display of the currently selected operating modes depends on the preset mode group
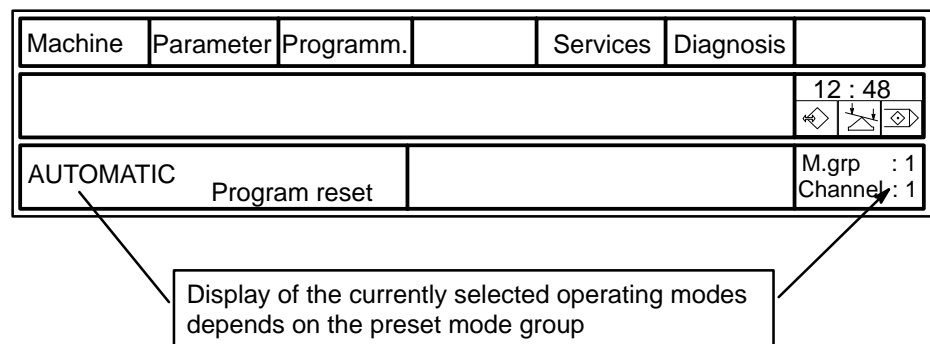
> Switchover of the machine control panel is configured by the machine manufacturer. Please read the machine manufacturer's specifications.

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:

Program reset

The machine is in the reset state, e.g. after switching on or after end of program (M30).

*Program running (not in **JOG** mode)*

A program has been started and program execution is running.

Program stop

A program in execution has been interrupted.

NC Stop

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).

NC Start disable

NC START is disabled in the relevant mode group in certain error states, e.g. if machine data are incorrectly parameterized.

> The NC Start disable only applies to a single mode group.
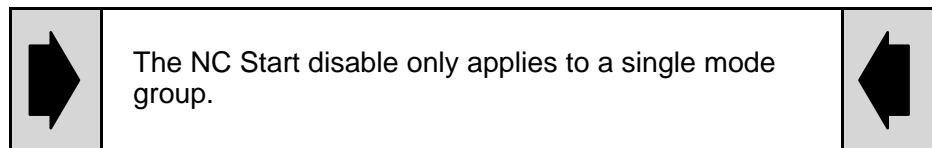
Example

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

From \ To		AUTOMATIC Program reset	Running	Stop	JOG Program reset	Stop	TEACH-IN Program reset	Running	Stop	MDA Program reset	Running	Stop
AUTO-MATIC	Program reset				X		X			X		
	Running					X	X			X		
	Stop					X	X			X		
JOG	Program reset	X					X			X		
	Stop			X			X			X		
TEACH-IN	Program reset	X			X					X		
	Running	X			X					X		
	Stop	X			X					X		
MDA	Program reset	X			X		X					
	Running	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 (%)			

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            Program stop                        │              Mode Gp 1  │
│                                                    │              Channel 1  │
├────────────────────────────────────────┬──────────┴──────────────────────┤
│ Actual value              REPOS offset  │ Program pointer                 │
│                                         │                                 │
│   X        -307.196          0.000      │                                 │
│   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*
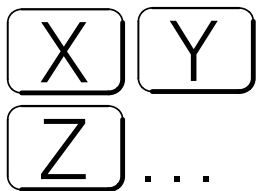
Select the axes.

Press these keys to traverse the selected axis in the respective direction.
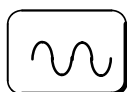
If you press the key RAPID OVERRIDE as well, you traverse the selected axes at rapid traverse speed.

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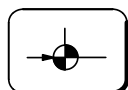
In JOG mode, "Approach reference point" may be selected by pressing
the appropriate key.

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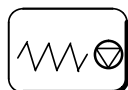
Using the FEED HOLD key, you can stop the selected axis before the
reference point has been reached.

> 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
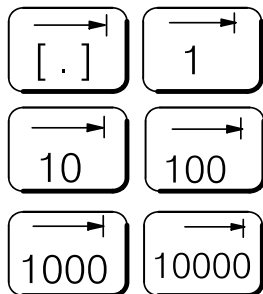
Before traversing the axes using the direction keys, select the appropriate incremental weighting on the associated machine control panel.
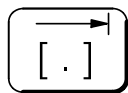
This key (INC variable) is used for traversing an increment previously set:
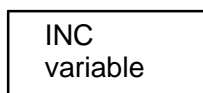
For setting the variable increment, activate this softkey and enter the value you require in the input field.
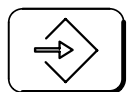
Press the INPUT key for entering the value in the "Incremental weighting for INC variable" window.

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 μm) 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.

> 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.
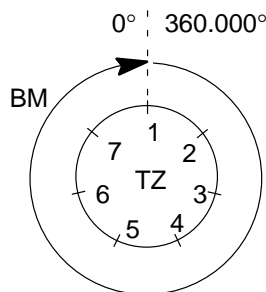
Example of "Indexed traversing" with a rotary axis:

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
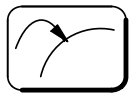
Press the REPOS key on the machine control panel.

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.

> **Caution:**
> After a tool change, the REPOS function can only be used with the same tool dimensions.

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.

> 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!

3.1.4.7   Finish thread
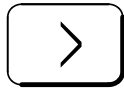
Press the ETC key to expand the softkey bar.

Finish/Set-up
thread
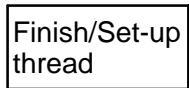
With this function you can load a thread that has been already cut and finish it.
Only threads with constant pitch can be finished.

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.
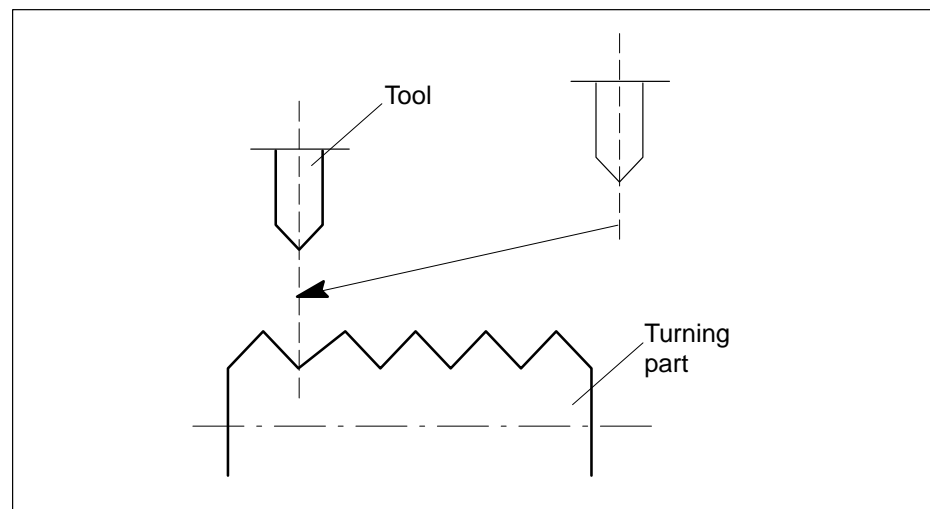
Note

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 inter-
actively. 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.
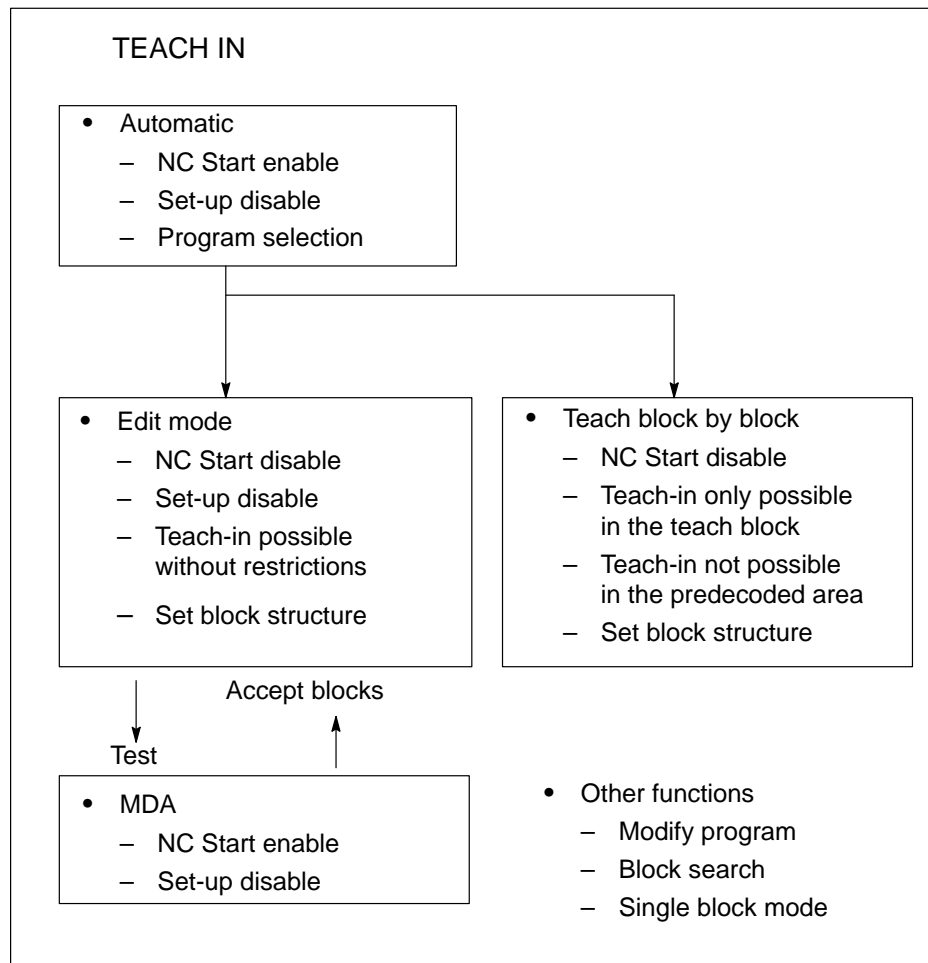
Fig. 3.3   TEACH IN

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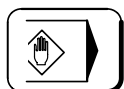
Press this key to select TEACH IN mode. The associated LED lights up.

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.

> Modify
> 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.
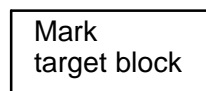
> Mark
> target block

press the MARK TARGET BLOCK softkey. The target block is
highlighted in colour.
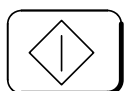
◇   When you press NC START, the block containing the breakpoint is
displayed. The started program executes up to and including the
selected block with the breakpoint.

```
MPF 4711   Workpiece  SHAFT1


 N0005   G0  X0 Z0 Y0  L_F
 N0010   G1  F500 X–80  L_F
 N0015   G3  X–80 Y0 I–80 J–60  L_F


 N0005   G0  X0  Z0 Y0  L_F
```
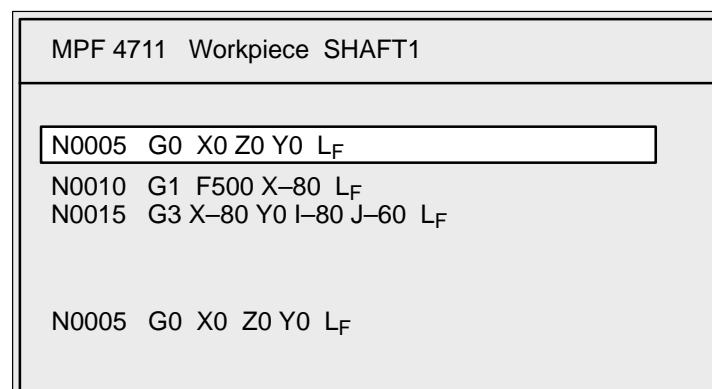
The block containing the breakpoint (N0005 in the example) is
highlighted.
When the block with the breakpoint has been processed, you can
interrupt program execution with PROGRAM STOP. You can continue
program execution with NC START.

*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".

> Teach-in is only possible if decoding and processing take place on the same level. In other words, a program must be interrupted with a decode stop (DSB, PST, BRK, End) and an advanced decoding (tool radius compensation, contour definition, etc.) may not be active.
> You can use the single block key on the machine control panel to interrupt a program but you cannot perform a teach-in at this breakpoint (it is not a decode stop).

3.1.5.2 Edit mode

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.

> In edit mode you can edit but not start a part program (NC start disable).
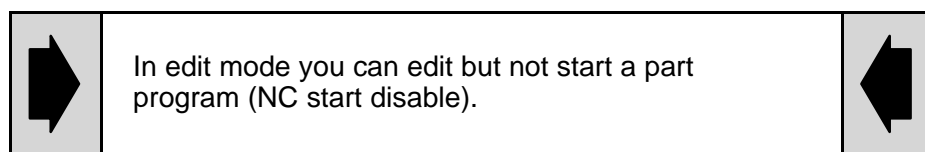
Note

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
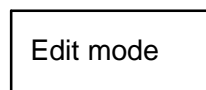
Edit mode

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

3.1.5.3   Block structure settings

Block struc-
ture 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       │
│                                           │ Block number              no   │
│  X      -76.006            0.000          │ Next block no.             5   │
│  Y        0.000            0.000          │ Block no. diff.            5   │
│  Z        0.000            0.000          │ Settable zero           none   │
│                                           │ offset                         │
│                                           │ Workpiece dim.          none   │
│                                           │ Axes               selected    │
├──────────────────────────────────────────┼────────────────────────────────┤
│ Act. value  Override Position Set value   │ Axis selec.                    │
│ F=    0.00 120%          F=    0.00M       │       X    yes                 │
│ S 1=    0  115%    0.00  S 1=     0        │       Y    yes                 │
├──────────────────────────────────────────┤       Z    yes                 │
│ Edit mode    WELLE1   MPF1012   Insert  9  │                                │
│ N025 G04 X2↳                               │                                │
│ N030 G90 G00 X150 Y150↳                    │                                │
│ ]035 Z100↳                                 │                                │
│ N040 G91 G01 X50 F100↳                     │                                │
│ N045 X50 Y50 F200↳                         │                                │
├────────────────────────────────────────────────────────────────────────────┤
│ Select with the selection key            │ ▮                              │
├─────────┬──────────┬─────────┬─────────┬──────────┬───────────┬────────────┤
│ Accept  │ Generate │         │         │ Insert/  │ Search    │            │
│ position│ block    │         │         │ overwrite│           │            │
└─────────┴──────────┴─────────┴─────────┴──────────┴───────────┴────────────┘
```

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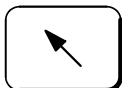
You can use the window switchover key to switch to the "Block structure/settings" screen. The active window is always highlighted by a yellow frame.

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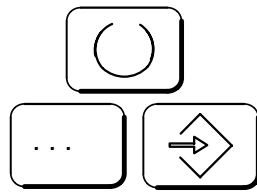
Use this key to select the toggle field options.
You enter values in the input fields, using the keyboard and the input key.

Example

Block structure/settings	
Block number	Yes
Next block number	10
Block number difference	5
Settable zero offset	G53
Workpiece dimensions	None
Axes	traversed

– Generated NC blocks are numbered

– Generated TEACH block is No. N10

– Block spacing (N15, N20)

– G53 is selected

– No switchover in inch/metric measurement system

– 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.	

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

Note

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_F.

Operating sequence
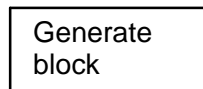
Generate block

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	traversed
Axis selection	
X	yes
Y	no
Z	yes
A	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_F

– with traversing movement on axes X and Y
N10 X = 100 L_F

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$_F$

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$_F$

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$_F$

Note   The accepted positions are displayed in the format X=100 and not X100.
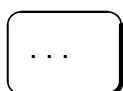
. . .   Once you have generated a block, you can insert G commands,
              technology data, etc. in the block using the keyboard.

3.1.5.5   Accepting axis positions

Milling machine

X   Y   Select the appropriate axes and
. . .

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

Turning machine

+ X **−X**

+ Z **−Z**

⋮
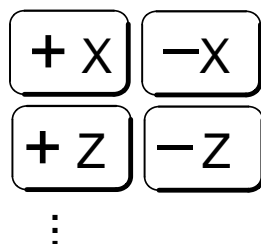
Traverse the axis according to the desired dimension.
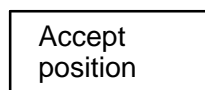
Accept position

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.
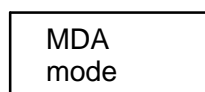
MDA mode

Press the MDA MODE softkey.
The MDA window is highlighted by a yellow frame.

. . .
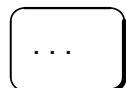
Use the keyboard to enter blocks in this window.
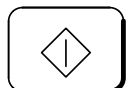
You can press NC start to start the blocks and run them like a program.

↑	↓

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_F		
L200 L_F		
M3 S1025 M7 L_F		
M4 M8 L_F		

⌃

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

3–29

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.
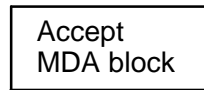
| Accept MDA block |

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

```
MPF 123  Workpiece SHAFT1                    Insert  2


N10  G1  F500 X320  LF
N15  M3 S1025 M7  LF
```
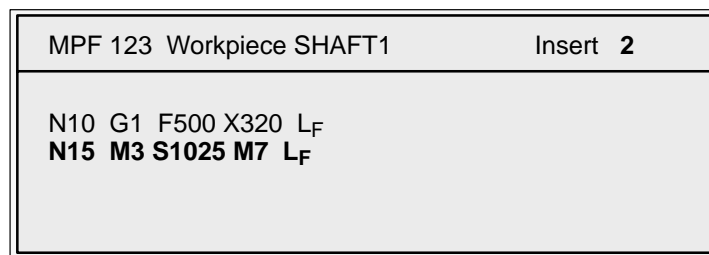
If you want to save more MDA blocks, switch to the MDA mode again. Use the arrow keys to select the next block, switch back to the edit mode and save the block in the teach-in program as described above.

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.

The MDA mode can be used to test part program blocks and save them in the teach-in program. This makes it possible to start program blocks without switching modes. The teach-in program can only be edited in edit mode.

MDA mode in edit mode and the extended overstore function are interlocked. It is not possible to overstore if MDA mode has not been exited correctly and it is not possible to call up MDA mode if the extended overstore function has not been completed.

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).

> Teach in is only possible at program breakpoints that do not require advanced decoding (no tool radius compensation, no contour definition), otherwise editing is not permitted and the handwheel is not enabled.

> G68 must always be programmed for block by block teach-in of rotary axes. Otherwise, the rotary axis might sporadically turn a whole revolution.
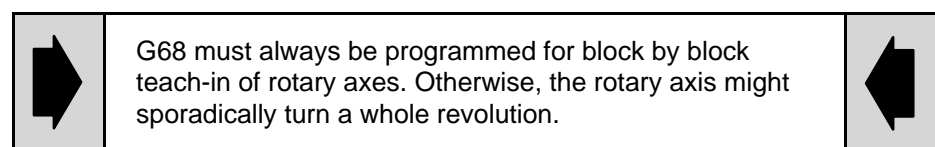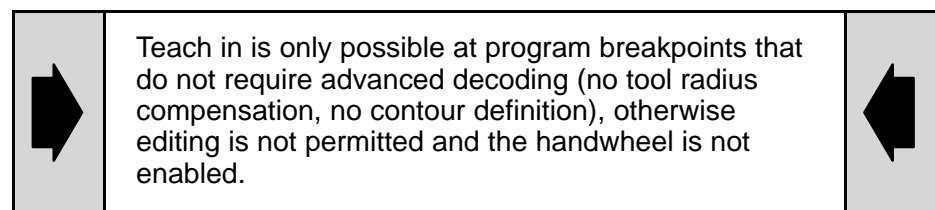
The following modifications are possible at the breakpoint:

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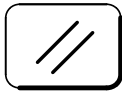
You can only switch between EDIT MODE and TEACH-IN BLOCK BY BLOCK after pressing RESET.

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

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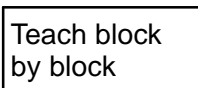
Press the TEACH BLOCK BY BLOCK softkey. The following softkey bar is displayed:

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

Block struc- ture settings
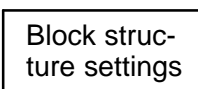
Press this softkey to enter the block structure settings.
The "Block structure/settings" and "Axis selection" selection screens are displayed (see the section "Edit mode"). Since it is possible to interrupt the execution of a program in teach block by block and to edit the program at the breakpoint, 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.
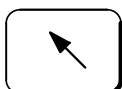
Switch to the "Block structure/settings" selection screen.

Use the arrow keys to move between the toggle fields:

- Block number

- Axes

and the input fields

- Next block number

- Block number difference
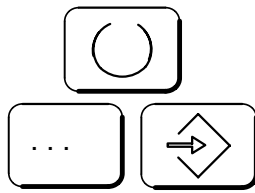
Press the selection key to display the different settings in the toggle fields.
You can enter the values in the input boxes using the keyboard and pressing the INPUT key.

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_F
N15   Y125 G1 F500  L_F
N20 G0 Y0  L_F
N30 L1  L_F
```

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

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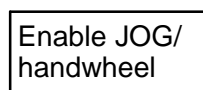
Press the ENABLE JOG/HANDWHEEL softkey to cancel the set-up disable. The "JOG/handwheel disabled" dialog text disappears and the dialog text "JOG/handwheel enabled" is displayed.

Milling machine

X   Y . . .

Select the appropriate axes and

+   −

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

Turning machine

$+X$ $-X$

$+Z$ $-Z$

:
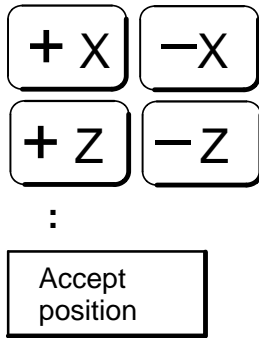
Traverse each axis to the required position.

Accept
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.

```
MPF 4711   Workpiece SHAFT1          Insert   4

N10  X0 Y0 Z0 G0   L_F

N15   Y125 G1 F500   L_F

L_F

N20 G0 Y0  L_F
```
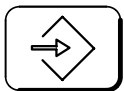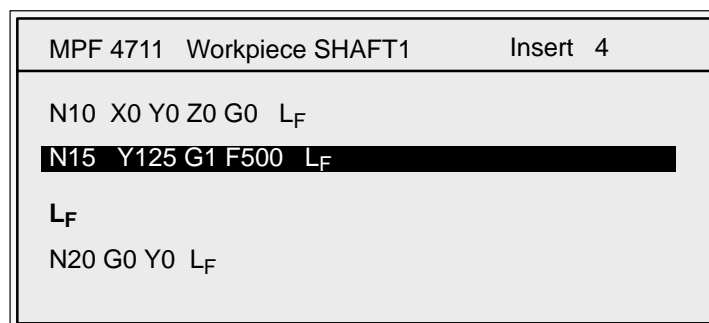
You can edit or insert the axis positions with "Accept position" in this generated block.

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_F
N15   Y125 G1 F500   L_F
N16  X= −25.600 Y= 68.300   L_F
N20 G0 Y0  L_F
```

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	yes
Next block number	16
Block number difference	5
Axes	selected
Axis selection	
X	yes
Y	yes
Z	no
A	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.

> Every teach block that you generate or insert should be confirmed with NC start. Only then can you generate a new block.
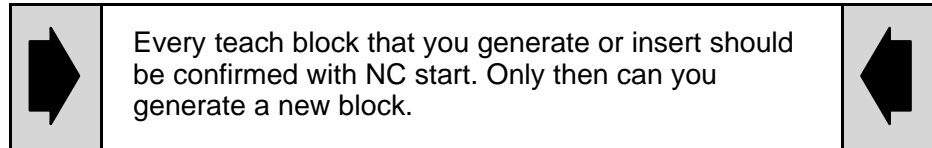
Note

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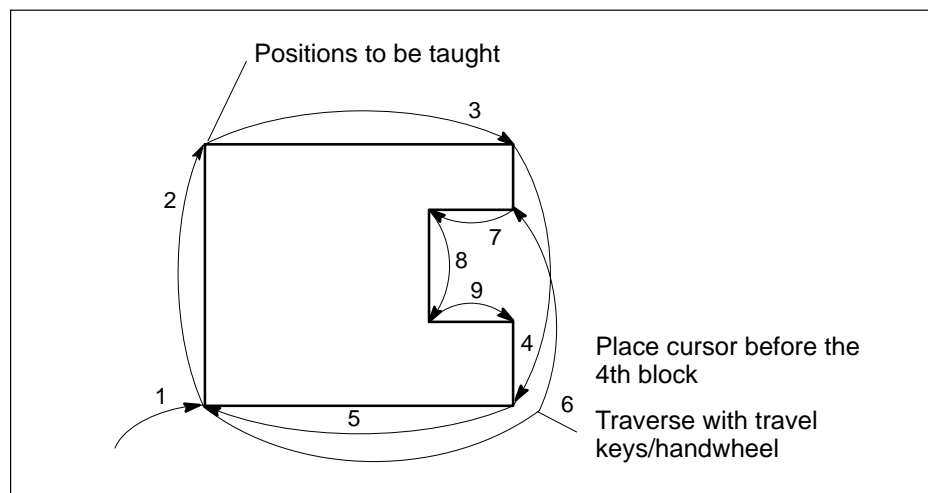
It is possible to edit the part program at any point in any order without traversing.

1st example for teach
block by block   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.
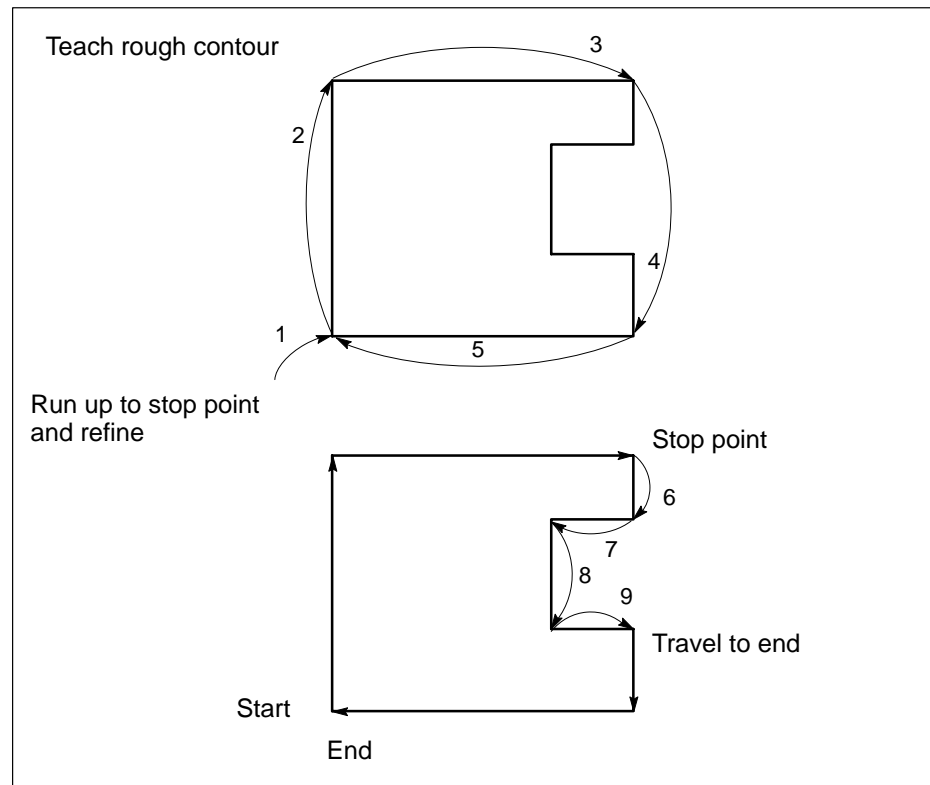
It is only possible to teach or edit in the following block. You must
observe the machining sequence. Every block that you modify must be
processed by pressing NC Start.

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.
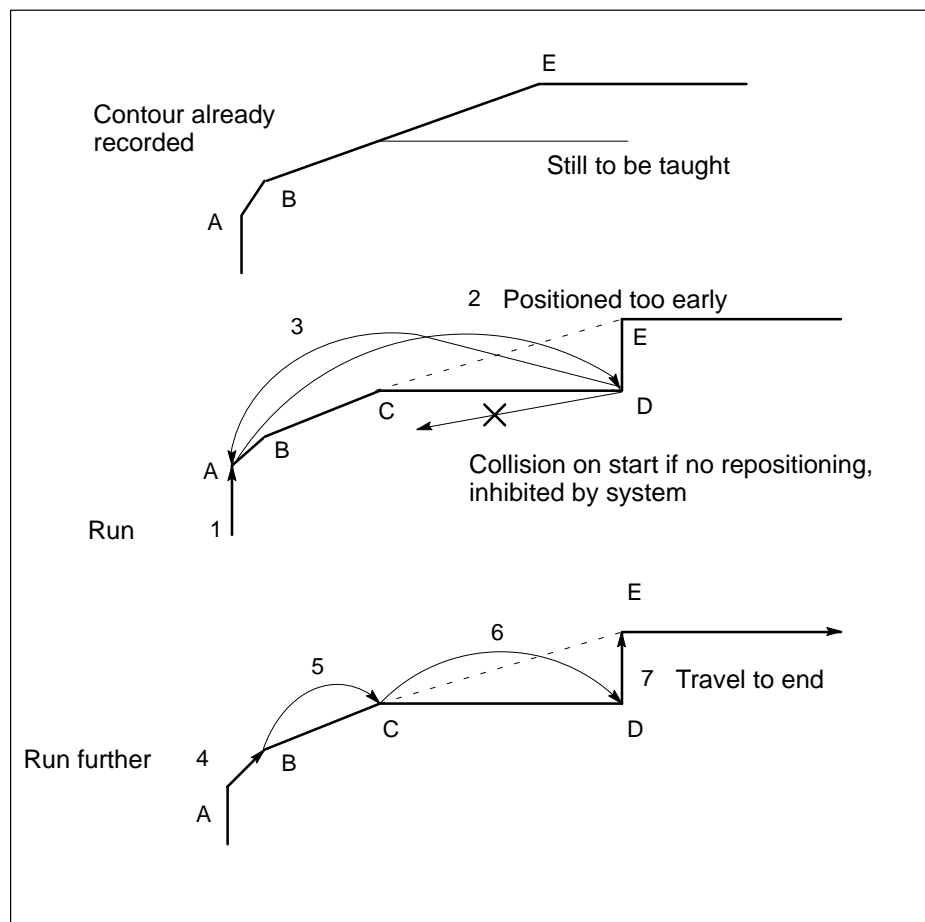
3.1.6   MDA mode

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.

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
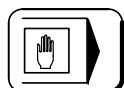
To select MDA mode, press this key on the machine control panel.
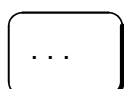
Use the keyboard to enter one or more blocks in the MDA... input window.
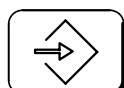
Press the INPUT key at the end of each block. An end of block character (L_F) is generated automatically and the cursor moves to the next line.

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.

Delete blocks

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.
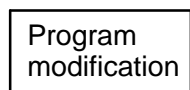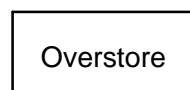
Overstore

The machine functions

- Overstore

Program modification

- 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).

```
User
Name          Type      Length        Date
–
GLOBAL        glup
LOCAL         parts
NCK           nck
```

```
User/NCK

  Caution! Programs are deleted on switching off!

Marker: EXT = external program  c = cycle protection   x = executable
                                r = readable            w = writable

Name          Type      Length    Marker     Comments in the 1st block  Marker

...
MDA           parp
MPF           parp
SPF           parp
```

```
User/NCK/MDA

  Caution! Programs are deleted on switching off!

Marker: EXT = external program  c = cycle protection   x = executable
                                r = readable            w = writable

Name          Type      Length    Marker     Comments in the 1st block  Marker

...
MDA 1         part      138       –rw–
MDA 2         part      1         –rw–
  ⋮
MDA 6         part      1         –rw–
```

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    ││
│                                                          └───────────┘│
│  JOG              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                                                    │
│  MPF         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

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

Save under

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.

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           
    Y        0.000               0.000           0
    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.
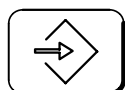
Now press the INPUT key,
the workpieces in this directory (LOCAL) are listed.

3–45

Now position the cursor on the required workpiece.

Example:

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.

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.

> 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.

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.

> 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.

| Stop work-
piece sel. | You can use this softkey to abort workpiece data transfer from the hard disk to the NCK memory. |

> **Note:** If workpiece data transfer is aborted, incomplete program parts might be left in the NCK memory.
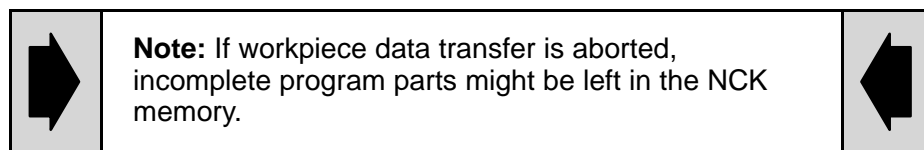
Dialog messages and alarms

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 **no** 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.
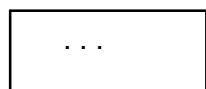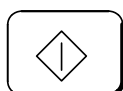
Select the machine functions by pressing the corresponding softkey.

SINUMERIK 840C (BA)

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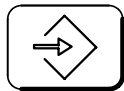
To select a main program, for example, the cursor is placed on the MPF directory and the INPUT key is pressed.

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 Zeichnungnr.: 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
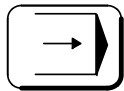
Press this key to select AUTOMATIC mode.

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.
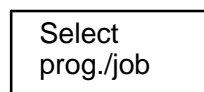
Select prog./job

Press the SELECT PROGRAM softkey.

The selected program is accepted by the program pointer.

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

3.1.7.3   Starting and interrupting a part program
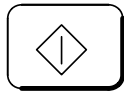
To start the part program, press the NC START key.

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.
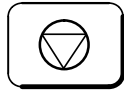
NC STOP
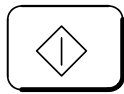
NC START

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

> The active block display only displays blocks related directly in terms of traversing logic. Subroutine calls, calculation blocks, etc. are not displayed.

> An NC program that only consists of arithmetic blocks and @ commands cannot be stopped with NC Stop! NC Stop is displayed but the program continues.
> Use RESET.

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
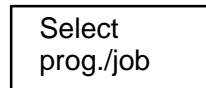
| Select prog./job | Press the SELECT PROGRAM/JOB softkey. |

| . . . | 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:

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

SINUMERIK 840C (BA)

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

| Over-store | Press the OVERSTORE softkey. |
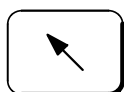
Activate the cursor in the "Overstore" input window.
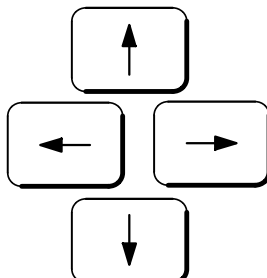
Now, using the cursor keys, set the reverse video marking on a desired input field.

"Overstore" input window

```
Overstore

S ■  =        M      =
T    =
D
H    =
```
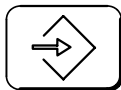
Enter the correction values via the keyboard and conclude each value by pressing the input key.

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.

%1234 | N10 | NC STOP, select
        | N15 | "Extended overstore"

MDA1 | N1 | "inserted"
       | N2 | program blocks

%1234 | N20 | The blocks prepared **before** the interruption are
        | N25 | traversed as if no interruption had taken place.
        | N30 |
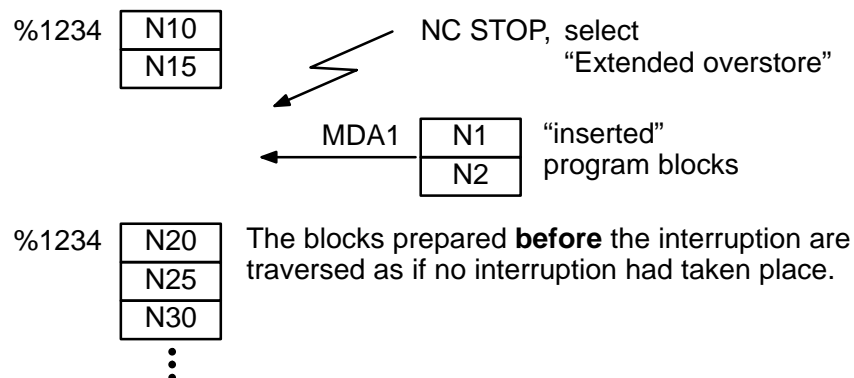
Note

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
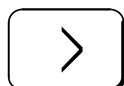
> Switch to the extended softkey bar by pressing key ETC.
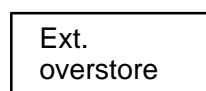
| Ext. overstore |

Select EXTENDED OVERSTORE.

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

```
MDA1                                    Insert  5

G0  X100
G1  Z20  M3  S3000  M8  H103
G0  X–50  Z100
G1  Z30  M4  M7  H106  S4000
G0  X–20  Z50  M5
```
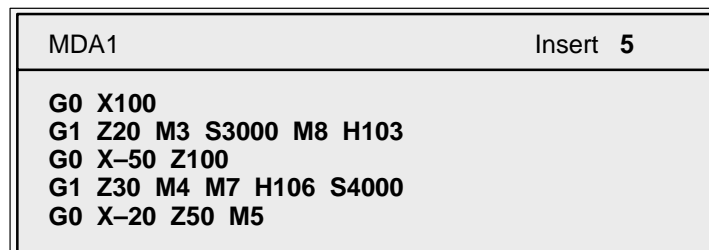
The text "Extended overstore" is inserted on the channel state display.

> If "Extended overstore" is active, the channel state display refers to the overstore channel. The overstore channel is not an additional channel, it is only virtual. As soon as the "Extended overstore" function has been terminated, the state of the NC channel is displayed again.
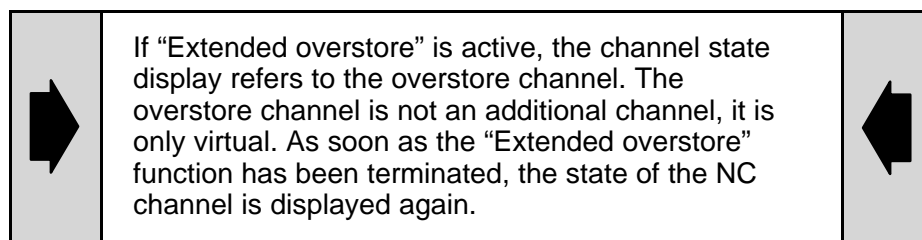
You can enter the required block sequence in the MDA1 window. The NCK editor is available to you here.

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

> NC START can be executed in EXTENDED OVERSTORE, even if the axes are **not** referenced.
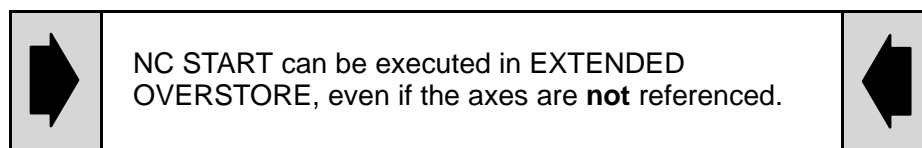
Press the RECALL key to exit "Extended overstore". This action takes you back to the aborted channel. The interrupted program can be continued at the point of interruption.

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
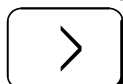
Extend the softkey bar with the ETC key.

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

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.

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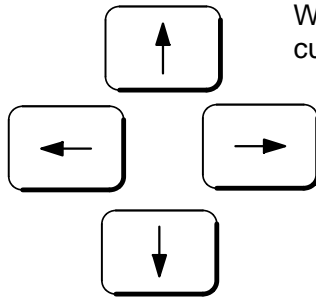
With the cursor keys you select the program modification function. A cursor marks the current position.

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 re-confirmation.

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.

DSB	**Decoding single block;** 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). The signal acts at the end of the block preset by being decoded in the current part program.
PST	**Program step;** Display: PST 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).
BRK	**Stop at target block;** Display: BRK (break) This feature allows you to interrupt the program at a defined point. The target block is selected by means of the • Cursor functions and • MARK TARGET BLOCK softkey 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).
CLR	**Clear;** Display: CLR CLR is **only** 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.

> 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).

Execution from external corresponds to the function
"Block–by–block reload" from hard disk referring to
Catalog NC 60.1.
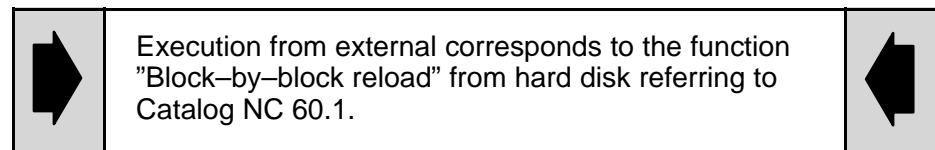
NCY   **Program start / End cycles off;** Display: NCY (no cycle)
             Automatic execution of program start and end cycles can be disabled
             with this function.

The machine manufacturer defines whether
program start or end cycles is used and, if
so, which subroutines are used for them
(channel-specific).
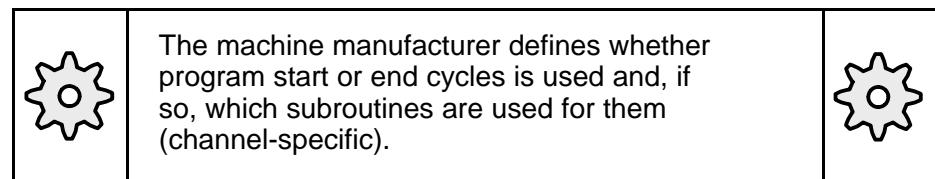
SAV   **Automatic saving:** Display: SAV in the NCK editor
             If this function is active a backup copy of the part programs in the NCK is
             automatically made in the relevant workpiece after they have been
             edited (e.g. with softkey: PROGRAMS/Edit or MACHINE/Program
             structure).

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

With technology MD the machine manufac-
turer can use workpiece NCKTMP instead
of workpiece STANDARD for this function.
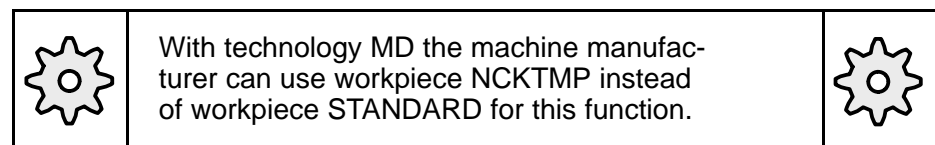
             To assign an original workpiece, use the softkey SAVE TO DISK and
             enter the workpiece name in the input line.

You can permanently activate the function SAV with
setting data 5001.2 = 1 (even after Power On).
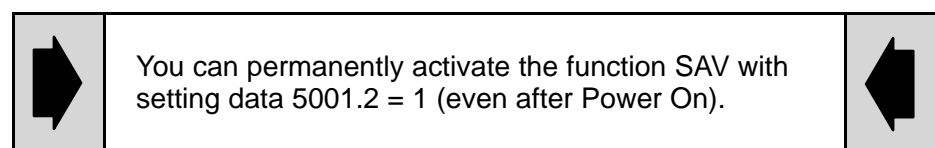
Limitations: Automatic saving is not active in TEACH IN mode. A program is only
                 saved if it has been altered or if you exit the editor or switch areas.

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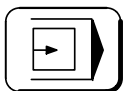
Pressing the SINGLE BLOCK key on the machine control panel a second time, deselects the function.

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[1]	X[1]
Blocks without position data	X[1]	X[1]
Arithmetic blocks		X[1]
Switching and auxiliary function blocks	X[1]	X[1]
Blocks generated within the control / CRC...) [3]	X[1]	
Threading blocks without dry run feedrate	[2]	X[1]
Threading blocks with dry run feedrate	X[1]	X[1]

- 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.

- A "Decoding single block" can be modified by "Overstore".

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

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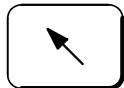
Switch to the search window (if the window is active, it is marked).
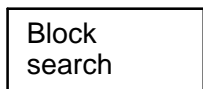
| Block search |

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)
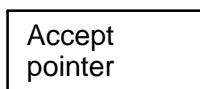
Accept pointer

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.
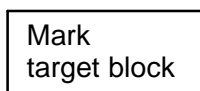
Mark target block

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.

> The target block for "Breakpoint" and "Block search" are identical.
> If the search destination is altered, the target block for the breakpoint is also altered.

Search with calculation
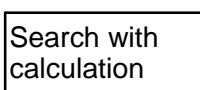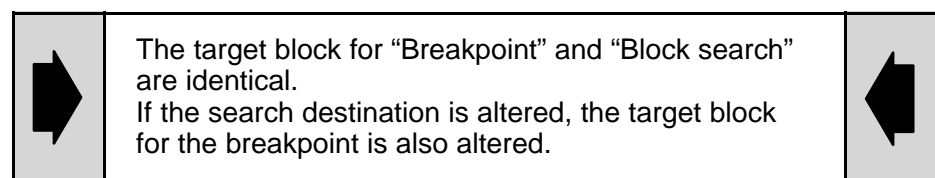
Press the SEARCH WITH CALCULATION softkey. The block search is carried out by the control.
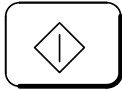
After NC START the program is started and executed.

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.
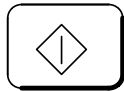
After NC START the program is started and executed.

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_F
N10 G1 F5000 X–80 L_F
N15 G32 X–80 Y0 I80 J–60  L_F
N20 G0 X0 Y25 Z1 L_F
N25 G01 Z–200 F500 L_F
```

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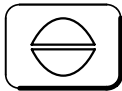
If a program is interrupted by a CANCEL alarm, execution of the program can be continued from the breakpoint by pressing NC START after the program error has been eliminated. When you press the CANCEL key you acknowledge the error message.
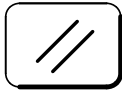
If a program is interrupted by a RESET alarm, the program can only be started after the RESET key is pressed. Execution begins from the start of the program.

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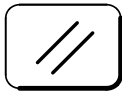
After the RESET key is pressed, you can edit anywhere in the entire part program. The selection of the correction block is cancelled.

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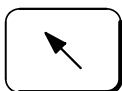
In the modes AUTOMATIC, TEACH-IN and MDA you can switch over to the part program input window and edit.
The active window is marked (yellow frame).

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	
Z	0.000	0.000	

Act. value	Override	Position	Set value		Auxiliary functions
F=	0.00	95%		F=	0.00M	T =	M =
S 1=	0	50%	0.00	S 1=	0	D =	M =
						H =	M =

WELLE1/MPF1012   Insert   1 | G functions

```
█ Z-Nr.:  34K100/12 )↳
( 25.10.94 Rm )↳
↳
N0010 G90 G00 Z↳
N0015 Y0↳
```

Save / edit program?   WELLE1█

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

Fig. 3.15   Save 01

Save to disk

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.

> ▶ If no workpiece is selected, only the path: USER/LOCAL is offered.
>
> The workpiece name must be entered via the keyboard. ◀

Note

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

Save under

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.

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.

Machine coordinate system **before** PRESET/DRF offset:

Machine coordinate system **after** PRESET/DRFoffset:

M   –   Machine zero
W   –   Workpiece zero
NPV –   Zero offset
M' –   Control zero
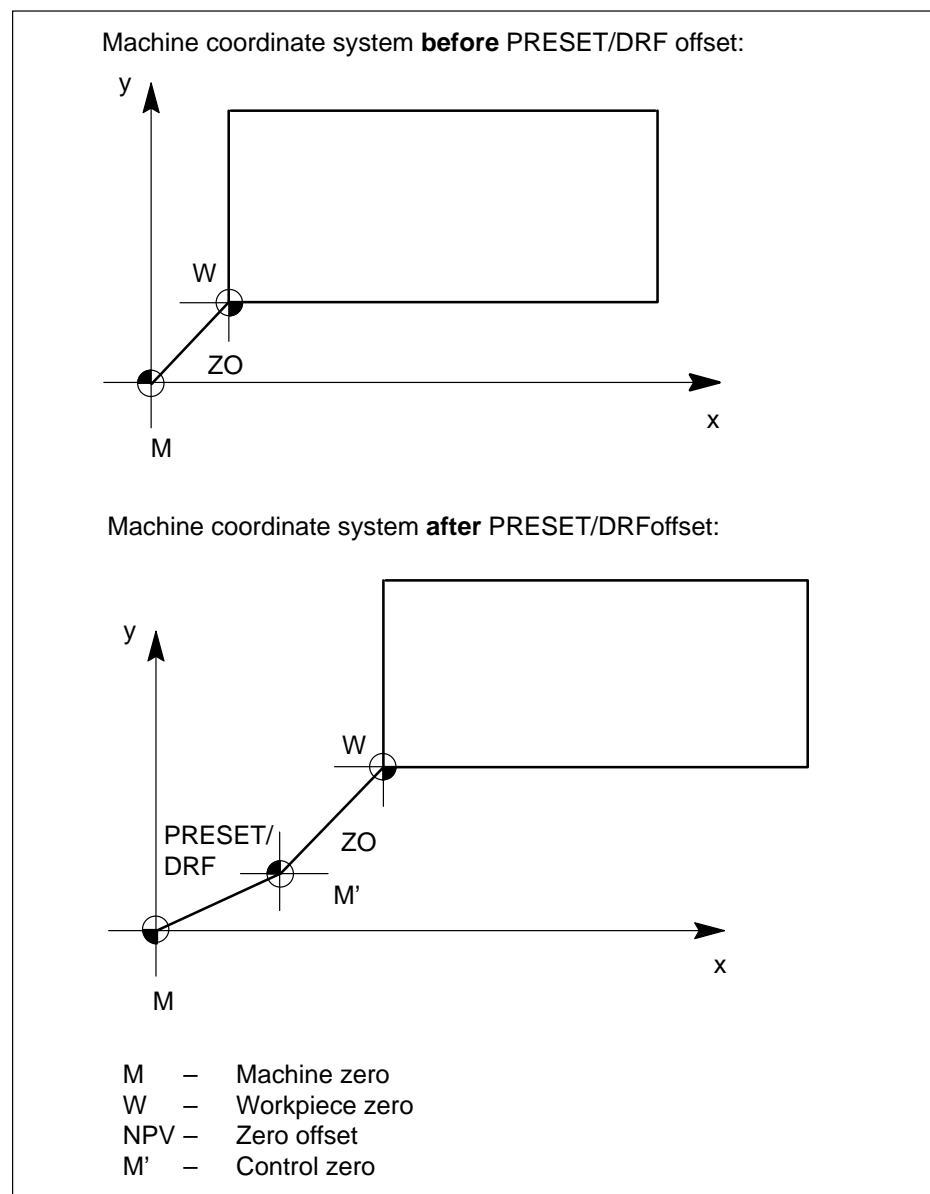
> The PRESET/DRF offset is not visible in the actual value display.
> The software limit switches and the working area limitation act as in the original coordinate system, i.e. they are not offset.

> ➡ | The PRESET offset can only be entered when **all** mode groups are in the RESET state. | ⬅

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*

| Preset DRF |

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.
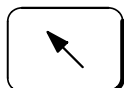
Switch to the upper window, if necessary.

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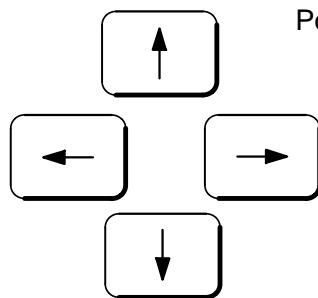
Position the cursor to the input field for the ID number.

Enter the appropriate value via the keyboard.
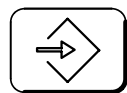
Confirm your input by pressing the INPUT key.

Switch to the lower input window.
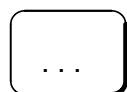
Enter the value for the correction number D... via the keyboard.
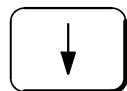
Position the cursor on the correction direction input field.
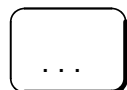
Enter the appropriate value (0 = positive, 1 = negative) via the keyboard.
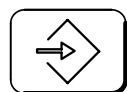
Confirm your input by pressing the INPUT key.

Without including the tool offset
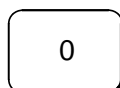
0	Enter the value 0 and press the CORRECTION NUMBER softkey.
Correction number	
	After having selected this data, enter the actual PRESET offset.
... =	Enter the axis name and the new actual value for the axis.
...	
Store act. value	Conclude the input by pressing the STORE ACTUAL VALUE softkey.
Delete offset	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.
i	If the "Help" key is pressed once the function has been selected, information on the function is displayed on the screen.
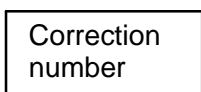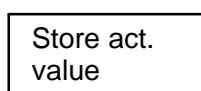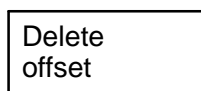
The PRESET offset remains stored:

- After "End of program"

- After RESET.

	Machine data is used to determine whether the PRESET offset is deleted automatically:
	• When the control is switched on
	• When the reference point is approached
	(See the machine tool manufacturer's documentation).

3.2.8.2   DRF offset

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

	The DRF offset is displayed on the screen in the DRFinput window. In the actual value display, however, the DRF offset is **not** visible.

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 endlessly turning rotary axis is an option.
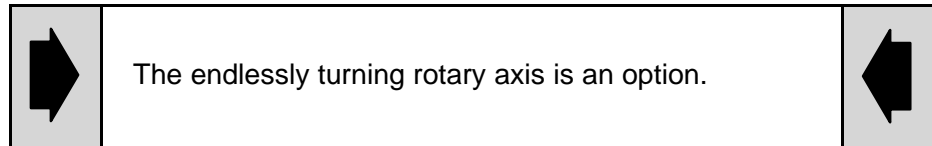
The G functions

- 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.

> The function must be configured by the machine-tool manufacturer. Please therefore consult the machine-tool manufacturer's documentation!

3.2.10 Extended stop and retract
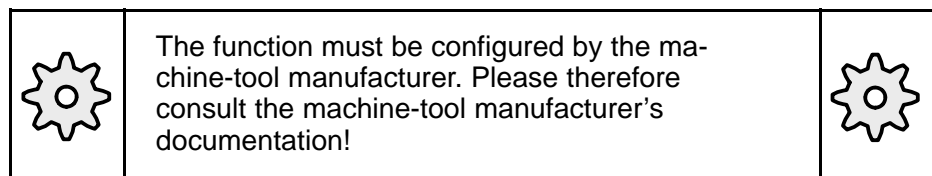
> Extended stop and retract is an option.
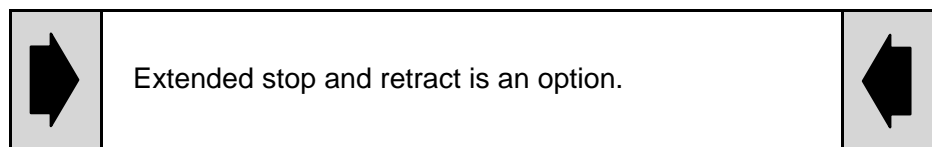
The EXTENDED STOP AND RETRACT function is used to protect the workpiece and machine.

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

| | The function must be configured by the machine-tool manufacturer.

It is, for example, dependent on the drives used.

Please consult the machine-tool manufacturer's documentation! | |

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

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

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.

Fig. 3.17   Multichannel display in AUTOMATIC mode
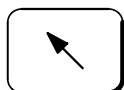
The "Home" key is used to switch from one channel display to the other. The channel display ("Active window") currently active is marked (e.g. yellow frame).
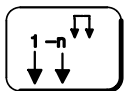
With the channel change key you switch through the channels in the active window.

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:
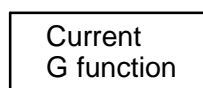
Current
G function

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".

Settings in the MD define which axes and spindles are displayed in each channel window (configured by the machine manufacturer).

The manufacturer can also configure displays for more than 2 channels of a mode group (WS 800 A).

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.

Please consult the machine manufacturer's documentation regarding these points.

END OF SECTION

4 Parameter

| Machine | **Parameter** | Programm | | Services | Diagnosis | |

4.1 Editing program parameters and setting data in the NCK

Computer link (Option)

4.2.1 Tool offsets	
4.2.2 Zero offsets	
4.2.3 Angle of rotation (coordinate rotation)	
4.2.4 R parameters	
4.2.5 Plane	
4.2.6 Setting data	
4.2.7 Additional protection zone adjustment via setting data (from SW 6.3)	
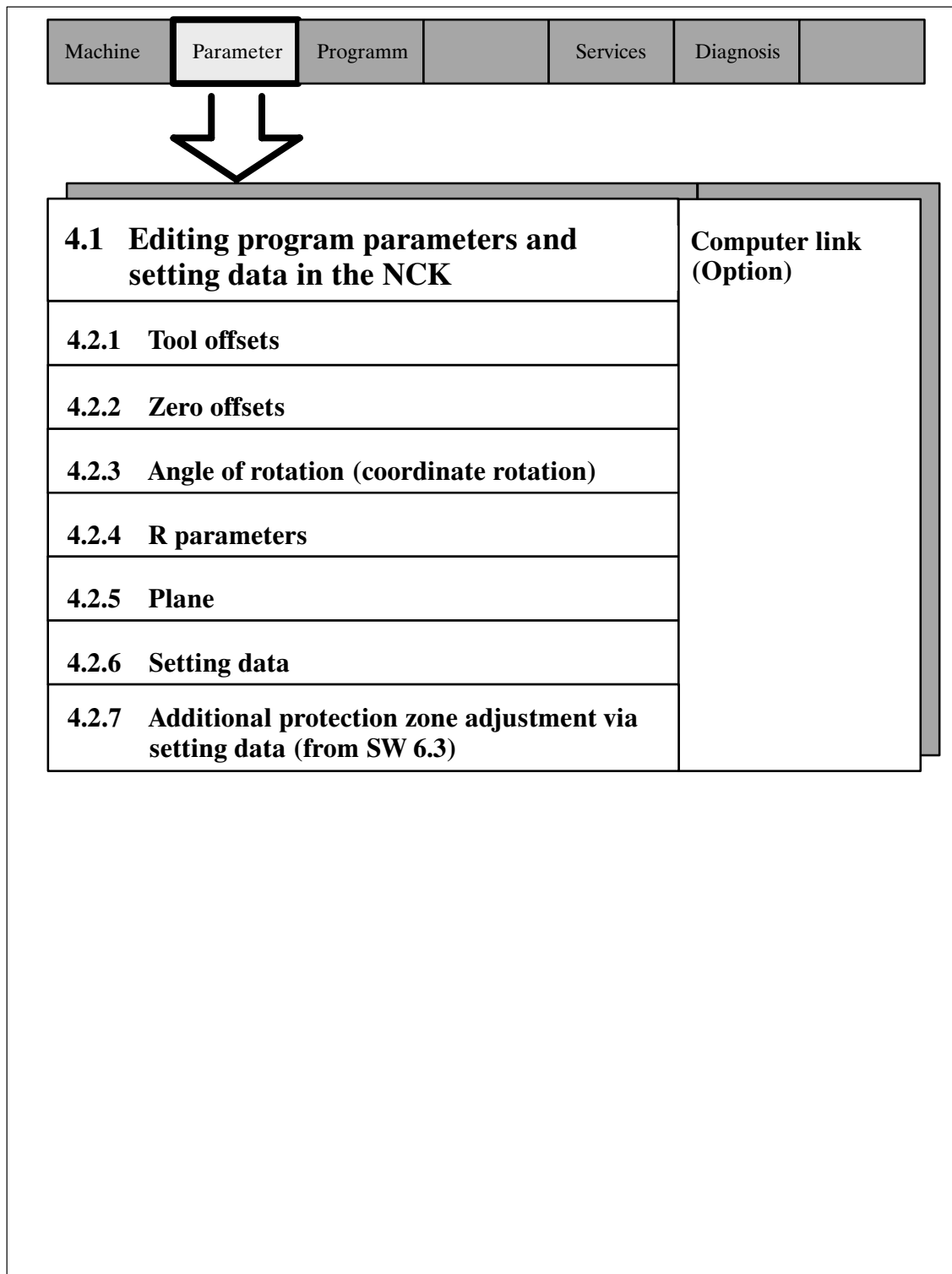
Fig. 4.1   Parameter

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:

```
┌──────────┬──────────┬──────────┬─────────┬──────────┬──────────┬────────┐
│ Machine  │Parameter │Programm. │         │ Services │Diagnosis │        │
├──────────┴──────────┴──────────┴─────────┴──────────┴──────────┴────────┤
│                                                            │   13:27   │ │
├────────────────────────────────────────────────┬─────────────────────┤ │
│ AUTOMATIC  Program reset                        │          Mode Gp 1  │ │
│                                                 │          Channel 1  │ │
├──────────────────────────────────────────────────────────────────────┤ │
│                      List of contents:                                │ │
│                                                                       │ │
│   Program parameters:     Tool offset, zero offsets,                  │ │
│                           angle of rotation, R parameters             │ │
│                                                                       │ │
│   Setting data:           Working area limitations, spindle data,     │ │
│                           scale, general data, general bits,          │ │
│                           axial bits, cycle data                      │ │
│                                                                       │ │
│   Computer link:          Data transfer                               │ │
│                                                                       │ │
│                                                                       │ │
│                                                                       │ │
│                                                                       │ │
│   User:                   User functions optional                     │ │
│                                                                       │ │
├──────────────────────────────────────────────────────────────────────┤ │
│                                              ▮                         │ │
├──────────┬──────────┬──────────┬─────────┬──────────┬──────────┬────────┤
│ Program  │Setting   │Computer  │         │          │          │ User   │
│parameters│data      │link      │         │          │          │        │
└──────────┴──────────┴──────────┴─────────┴──────────┴──────────┴────────┘
```

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
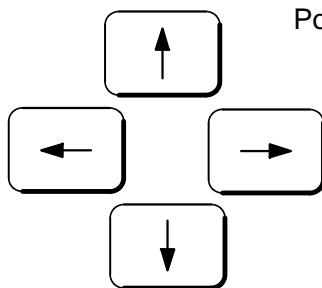
Position in any input field with the cursor keys.
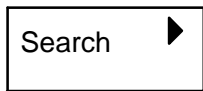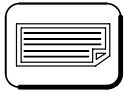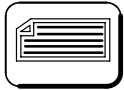
With these keys you page the screen display "down" and "up".

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.

Search   ▶

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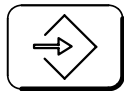
If the input value is correct, you conclude the input with this key. The value is transferred to the memory.

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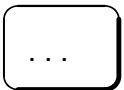
With the EDIT WORD key you can modify the current value by addition or subtraction (does not apply to setting data and R parameters).

Examples
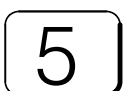
Current value: 50
New value: 45
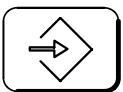
If you wish to delete the value:

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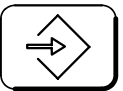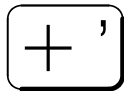
With the PLUS and INPUT keys you fix the cursor in the input line (e.g. with tool offsets). Press the keys before you enter the value. The cursor remains positioned on the selected input field even after you press INPUT.
Fixing of the cursor is deactivated with the next cursor movement.

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	

Automatic   Program reset

Mode grp. :1
Chan. :1

Test display for input in PLC texts

Text 1 DB59, from DW 0:   TURNING TOOL1

Text 2 DB59, from DW 30:   END MILL

Text 3 DB59, from DW 60:   TAP

Text 4 DB59, from DW 90:

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=         TO 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

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

Fig. 4.3   Tool offsets

TO wear	To display the other parameters, P5 to P7, you press this softkey.
TO geo + base	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.
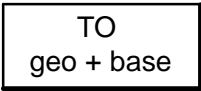
The meaning of the parameters is described on the following pages. Editing data is described in the section "Editing data in the PARAMETER area".
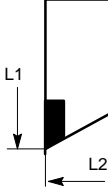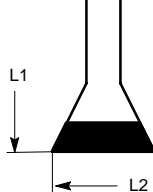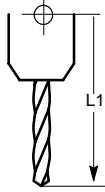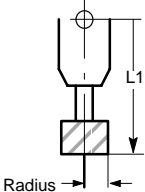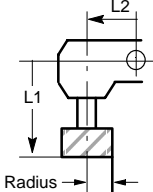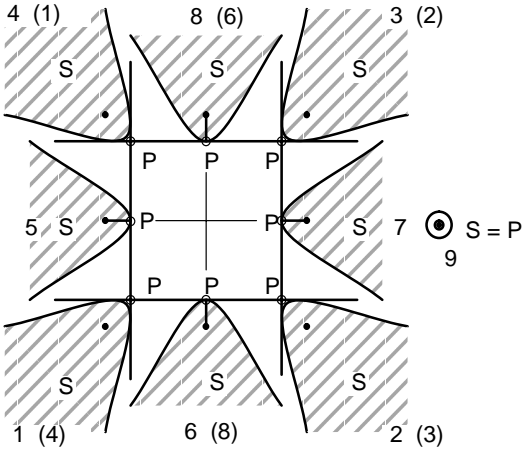
D No. Number for tool offset memory

P No. (Ident. No.) Tool offset parameters P0 to P9

TO parameter 0 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.	⚙

SINUMERIK 840C (BA)

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 off-set 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.
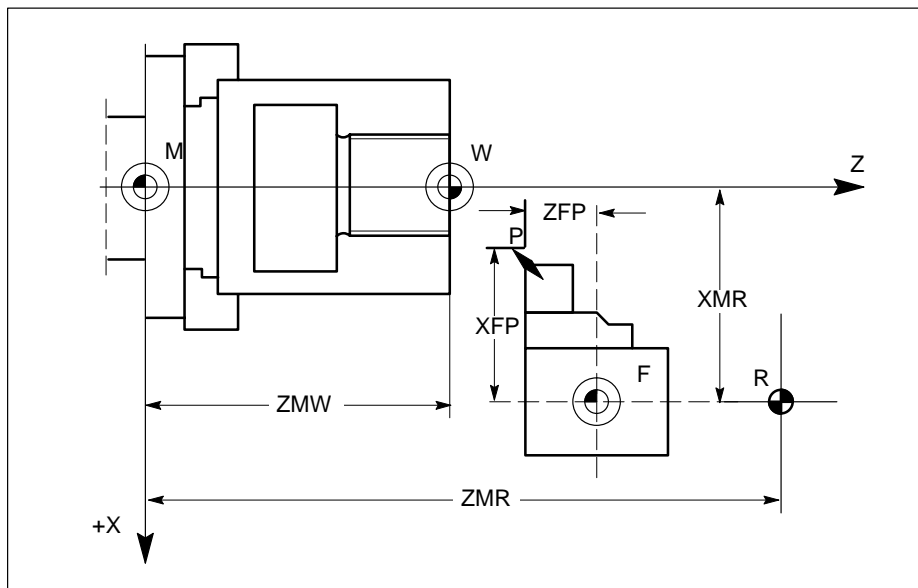
Fig. 4.4   Example: Lathe

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

> 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.

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.
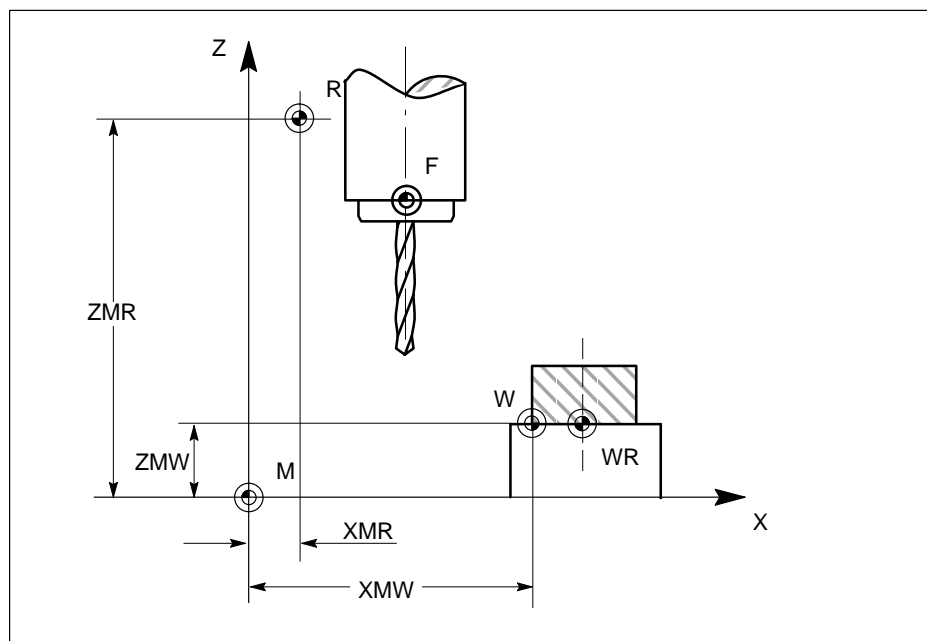
Fig. 4.5   Example: Drilling and milling machine

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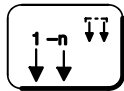
Select the desired channel for channel-specific parameters by pressing the CHANNEL SWITCHOVER key.

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

Channel 1 to channel 6

Transfer parameters	R00 ⋮ R49	R00 ⋮ R49	**R0 to R49** Typical use for each channel: assignment of cycles and subroutines.
Local parameters	R50 ⋮ R99	R50 ⋮ R99	**R 50 to R99** Typical use for each channel: for calculations within programs and subroutines. In the case of nested subroutines, the same local parameters can be used. An R parameter stack saves data hitherto used with cycles or on subrouutine call with @040 to 043 and stores it after returning to the program that made the call.
Global parameters	R100 ⋮ R199	R100 ⋮ R199	**R100 to R199** Typical use for each channel: store for data to which main programs and subroutines must have access. R100 to R109 are assigned if Siemens tool management is used. R110 to R199 are assigned if Siemens measuring cycles are used.
Reserved for internal functions	R200 ⋮ R219	R200 ⋮ R219	**R200 to R219** Internal assignment (cycle converters)
	R220 ⋮ R239	R220 ⋮ R239	**R220 to R239** WS800 compiler
	R240 ⋮ R299	R240 ⋮ R299	**R240 to R299** Provided for internal assignemnt.
	R300	R300	**R300** Stack pointers for @040, @041, @042, @043
	R301 ⋮ R599	R301 ⋮ R599	**R301 to R599** Stack area for @040, @041, @042, @043
Additional global R parameter supplement	R600 ⋮ R699	R600 ⋮ R699	**R600 to R699** Reserved for users
Central parameters	R700 ⋮ R999		**R700 to R999** (standard) Typical use: Higher level store for all NC channels, e.g. for buffering target positions which are being used by another channel.
	R1000 ⋮ R1299		**R1000 to R1299** (standard) Reserved for users
Chan. spec. ASCII–R parameters	R10000 ⋮ R10019		**R10000 to R10019** Reserved for user for the "Select part program for editing" function.

Channel specific R parameters *(bracket spanning Transfer parameters through Additional global R parameter supplement)*

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.
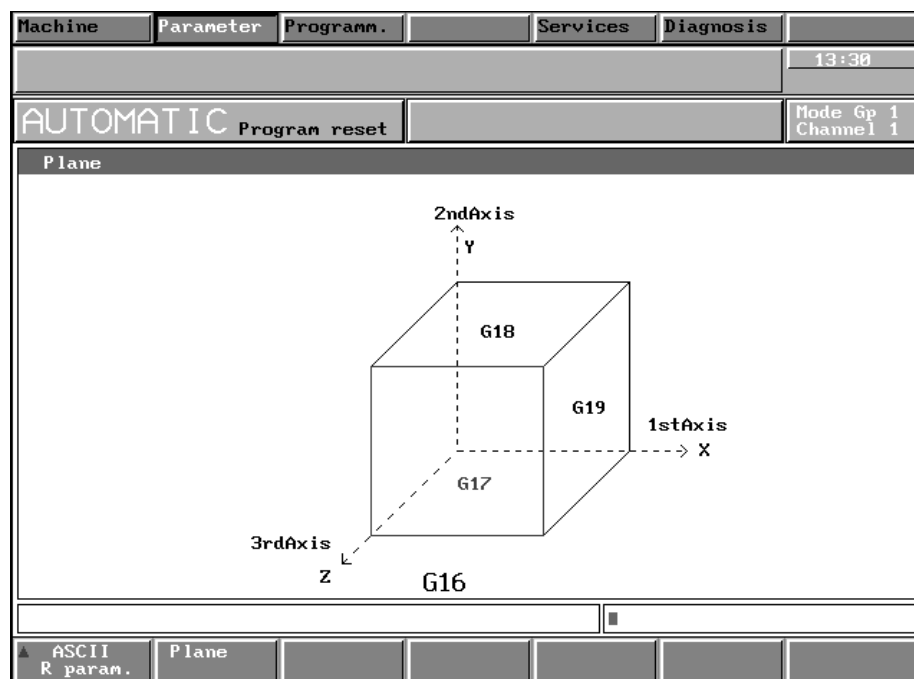
Fig. 4.6   Menu for plane selection in the parameter area

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 |

>
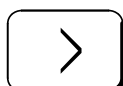
Press the ETC key to expand this softkey bar. The following softkeys are then displayed.

| ▲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
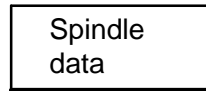
| Spindle data | With this softkey, you select the input menu for the spindle data. You can enter the following data: |

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
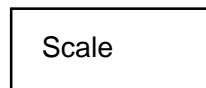
| Scale | With this softkey you select the display menu for the scale data. |

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)

```
  5001        0 0 0 0 0 0 0 1

                7 6 5 4 3 2 1 0

           Bit pattern for setting data 5001
```

Setting data No.

Possible bit states:
"0" = bit is not set
"1" = bit is set
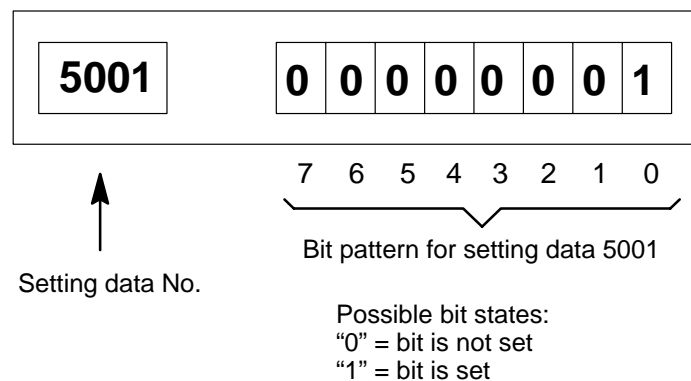
You can set the desired bit pattern for any byte in the area from 5000 to 5799 on the CRT by way of an input screenform.

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

Position
meas. signals ▶
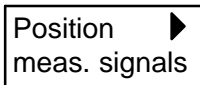
You select the softkey and the following display appears:

```
 Machine    Parameter  Programm.          Services  Diagnosis
                                                          15:01

 AUTOMATIC Program reset                              Mode Gp 1
                                                      Channel 1
  Setting data values:    Pos.meas.signals


          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: ± 99999999

> The software cam function is an option.

> The software cams function must be configured.
> Please refer to the machine-tool manufacturer's
> documentation.

> After you have changed the cam positions you
> must first perform an operation (see
> manufacturer's documentation) to activate the
> signal "Accept cam values" before the changes
> take effect.
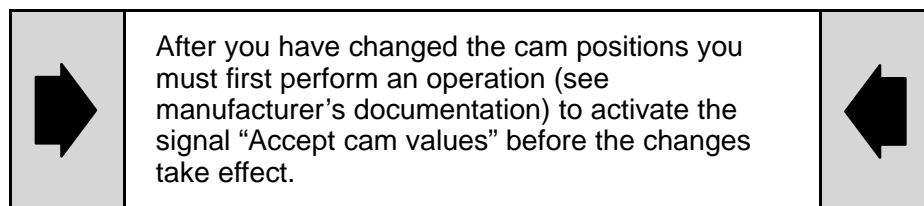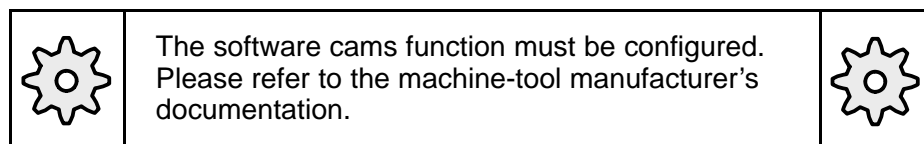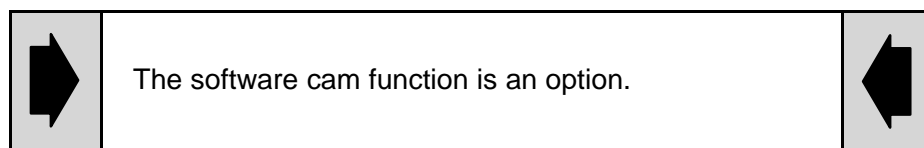
Notes

- 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
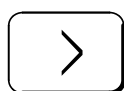
Press the ETC key to extend the SETTING DATA softkey bar.
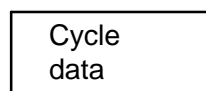
Cycle
data

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.

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.

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.

The machine manufacturer can configure an
input disable when the keyswitch is in the "0"
position.

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.
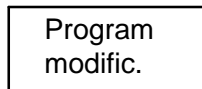
Program
modific.

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
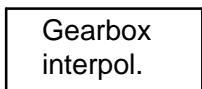
Gearbox
interpol.

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.

Fig. 4.9   Gearbox interpolation

The function gearbox interpolation must be configured by the machine-tool manufacturer (machine data).
Please consult the machine-tool manufacturer's documentation!

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.
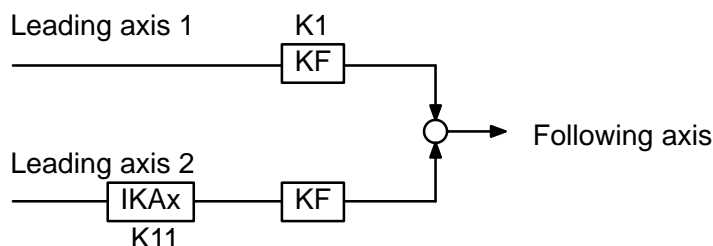
Link factor		Enter the link factor for the main axis by specifying the denominator and the numerator (8 decimal places + decimal point + leading sign). Activate the entry by pressing the LINK_ON/LINK_SWITCH softkey. The entered link factor is only accepted if a gearbox interpolation group has already 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

When you press a softkey, the prompt "Confirm input" is displayed. Press the key again to confirm.

| Delete 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".

| Define config. |

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_1 K_1.....LA_5 K_5 FA".

| Link_off 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".

| Link_on all |

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".

| Flying synchron. |

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_1 LA_1POS I J...LA_5 LA_5POS I J FA FAPOS".

| Link_on link_switch |

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_1 I J.....LA_5 I J FA"

| Start pos. Link_ON/Sw |

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.

<table>
<tr><td>⚙</td><td>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.

The actions "New configuration", "Switch link factor" and "Change start position" can be disabled independently of one another and axis-specifically.

Gearbox interpolation is an option. Please consult the machine manufacturer's documentation where necessary.</td><td>⚙</td></tr>
</table>

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.
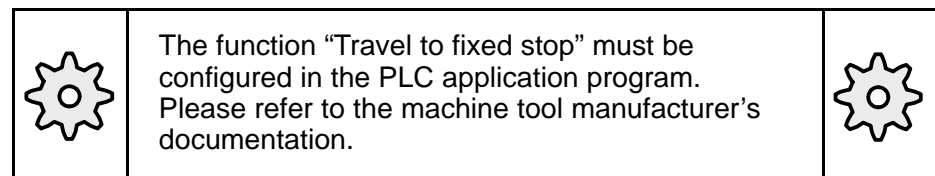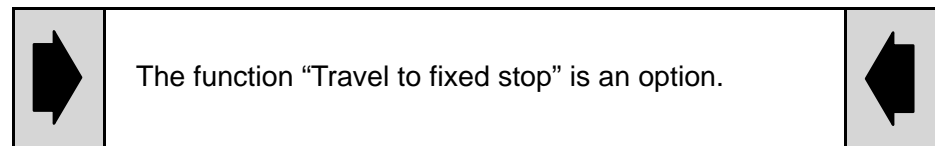
The function "Travel to fixed stop" is an option.

The function "Travel to fixed stop" must be configured in the PLC application program.
Please refer to the machine tool manufacturer's documentation.

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.

> For more detailed information about the protection
> zone adjustment function, please refer to the Installa-
> tion and Start–Up Guides for SINUMERIK 840C and
> SIMODRIVE 611D.
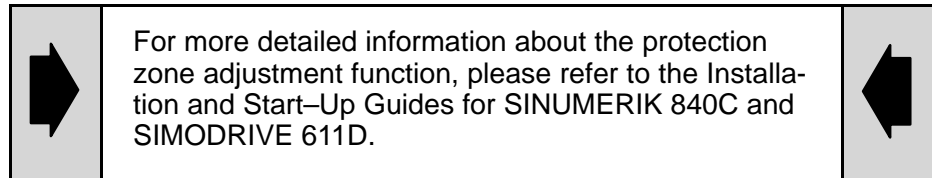
The "Protection zones" screen displays protection zone reference point
P1 and diagonal point P7 for individual zones. The displayed dimensions
are absolute with reference to the monitoring coordinate system.
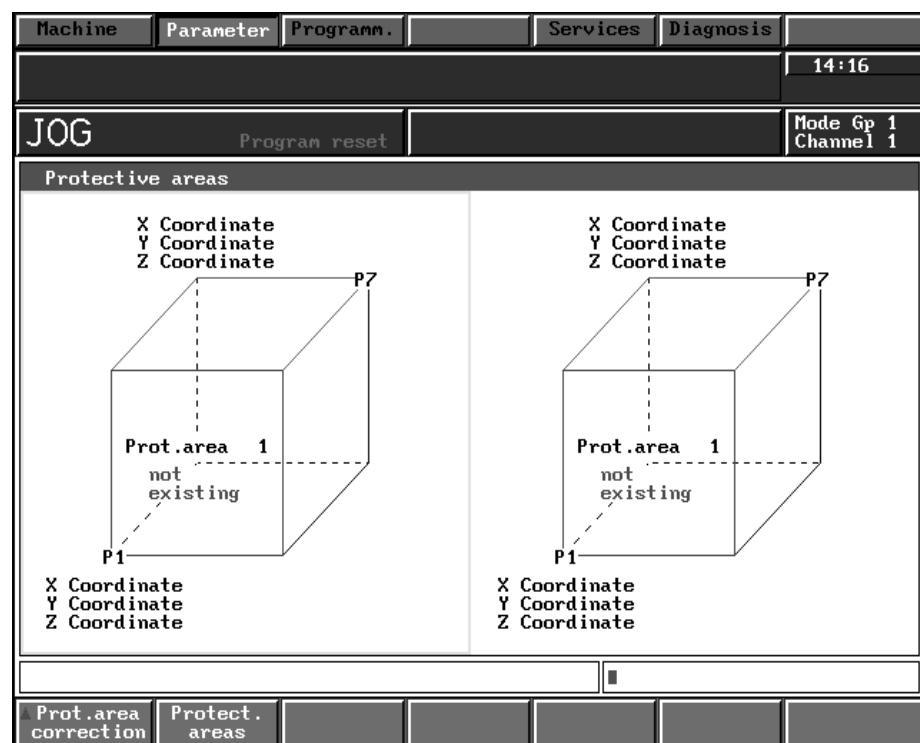
Fig. 4.10   Protection zone correction

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 │ Protect. │          │          │          │          │        │
│ correction│  areas   │          │          │          │          │        │
└──────────────────────────────────────────────────────────────────────────┘
```

Fig. 4.11   Protection zone correction

4.2.8   Inch/metric conversion (from SW 6.3)

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

END OF SECTION

SINUMERIK 840C (BA)

5 Programming

Machine	Parameter	Programm		Services	Diagnosis	

Editing part programs, creating workpieces and NCK files

5.1 Programming in the NCK memory area

5.1.1 Select program

5.1.2 Editing an existing program

5.1.3 Editing a new NC program

5.1.4 Grafische tool path simulation

5.1.5 Program input with operator support

5.1.6 Machining cycles

5.1.7 Plane

5.1.8 Program management

5.1.9 FANUC program converter

5.2 Programming on the hard disk of the MMC

5.2.1 Creating a workpiece

5.2.2 Creating a part program file

5.2.3 Creating job lists

5.2.4 Shopfloor sheet

5.2.5 Copy, paste and delete files in the hard disk data management area

5.2.6 Description of the ASCII editor

5.2.7 Editing IKA data SW 4.1 and higher

Fig. 5.1   Programming

In this area you can

- Edit part programs in the NCK user memory with the ASCII editor (NCK directory) and store them on the hard disk
- Create part programs with graphic support in the NCK user memory with the aid of the SUPPORT function
- Display overviews of the main programs and subroutines in the NCK memory
- Edit part programs, job lists, R parameters, setting data, cycle setting data, zero offsets and tool offsets which are assigned to a tool with the ASCII editor directly in the hard disk
- Create workpieces and associated data
- Create workpieces graphically with the "Graphic programming" option
- Test part programs with the DIN code simulation as soon as they have been programmed using the option "Simulation"
- Perform management tasks such as copy, delete and insert for the data in this area
- Display and edit shopfloor sheet and file of empirical values.

The option "Graphic Programming" is described in separate publications: User's Guide Graphic Programming System for:

- Drilling/Boring and Milling
- Turning

The option "Simulation" is described in the User's Guide "Simulation for Turning and Milling".

<table>
<tr><td>Program-
ming</td></tr>
</table>

You select the PROGRAMMING area with this softkey in the area menu bar.

Fig. 5.2   Basic display of the PROGRAMMING area

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.

```
┌──────────┬───────────┬───────────┬──────────────┬──────────┬───────────┬────────┐
│ Machine  │ Parameter │ Programm. │              │ Services │ Diagnosis │        │
├──────────┴───────────┴───────────┴──────────────┴──────────┴───────────┴────────┤
│                                                                     13:35        │
│                                                                                  │
│ AUTOMATIC  Program reset                                        Mode Gp 1        │
│                                                                 Channel 1        │
│  Part program   MPF1012    Workpiece   PART                    Insert       15   │
│                                                                                  │
│ ( 25.10.94 Rm )↳                                                                 │
│ ↳                                                                                │
│ N010 G90 G00 Z↳                                                                  │
│ N015 Y↳                                                                          │
│ N020 X↳                                                                          │
│ N025 G04 X2↳                                                                     │
│ N030 G90 G00 X150 Y150↳                                                          │
│ N035 Z100↳                                                                       │
│ N040 G91 G01 X50 F100↳                                                           │
│ N045 X50 Y50 F200↳                                                               │
│ N050 G90 X135 Y155 F500↳                                                         │
│ N055 G01 Z-10↳                                                                   │
│ N060 G02 X135 Y155 I50 J0 F250↳                                                  │
│ N065 ▊                                                                           │
│ ↳                                                                                │
│ ↳                                                                                │
│ ↳                                                                                │
│ ↳                                                                                │
│ ↳                                                                                │
│ ↳                                                                                │
│ ↳                                                                                │
│ ↳                                                                                │
│ ↳                                                                                │
│ ↳                                                                                │
├──────────────────────────────────────────────────────────────────────────────── │
│                                                           ■                       │
├─────────────┬──────────┬──────┬──────┬──────────┬──────────┬──────────┤
│ Generate    │ Support  │      │      │ Insert/  │ Search   │ Save     │
│ iblock no.  │          │      │      │ overwrite│          │ to disk  │
└─────────────┴──────────┴──────┴──────┴──────────┴──────────┴──────────┘
```

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.

> The part programs edited in this area must be
> backed up on the hard disk, otherwise they are lost
> when the control is switched off.
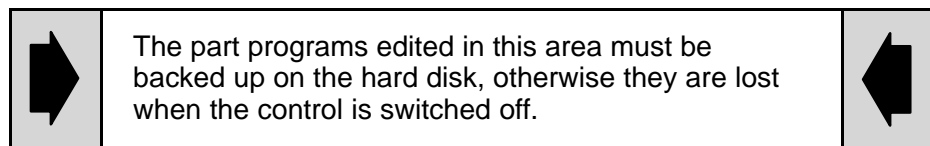
If the letter "i" is displayed in the 1st softkey window (left) notes relating
the selected area appear after pressing the HELP key.

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:

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).

Then press the softkey EDIT.

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").
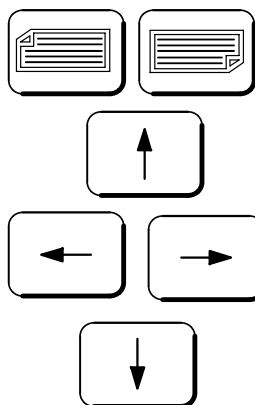
Using the page up/down and the cursor keys, you can position the
cursor at any position of a part program already stored. Pressing the key
combination SHIFT + ARROW or SHIFT + PAGE moves the cursor to
the start or end of the line or program file.
If you press RIGHT ARROW again at the end of a block, 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.
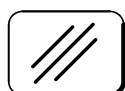
In the input line you can delete character by character from the right to
the left using the DELETE INPUT key.
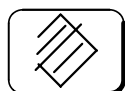
Press DELETE DATA to delete the character directly under the cursor.
The result of a text search is deleted completely.

Select the function CUT if you wish to delete a larger program section or
several blocks.
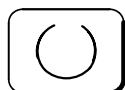
Use the selection key to select the first character of the character string
that you want to delete. Pressing the key twice cancels the selection.
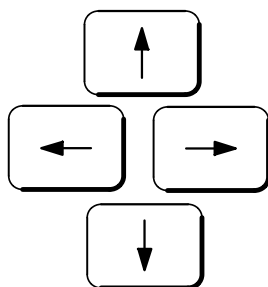
Use the arrow keys to select the desired text. You can press the Page
Up or Page Down keys to select larger areas of text.
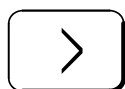
Press the ETC key to extend the softkey bar.
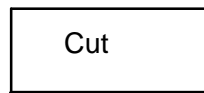
| Cut | Press the CUT softkey to delete the selected text. |

> The CUT function in the NCK editor does **not** store the deleted text in a buffer memory or clipboard, i.e. the text is deleted and cannot be restored.
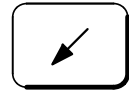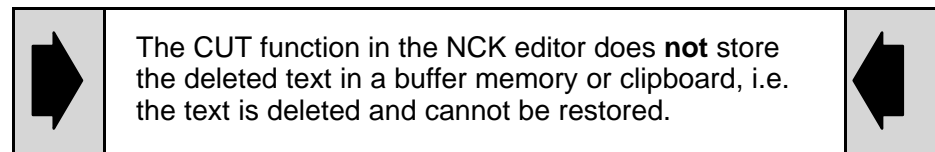
You can activate the cursor in the input line. The cursor is then inactive in the program, i.e. you can only enter characters in the input line (e.g. for the SEARCH function).

Pressing the key again deselects the activation.
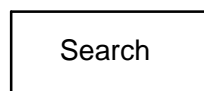
| Search | Press the SEARCH softkey to search for a block or a character string in the selected program. The message |

"Please enter search text".
appears.

| . . . | Use the alphanumeric keyboard to enter the search term and then press the SEARCH softkey. |

| Search | The cursor is positioned before the search term. The search is performed from the cursor position towards the end of the program. |
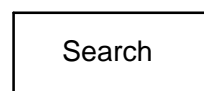
Note

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.
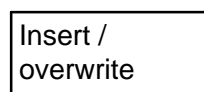
| Insert / overwrite | Press the INSERT/OVERWRITE softkey to change between the insert and overwrite modes. The cursor format changes as follows: |

Insert: The character is highlighted in adifferent 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

A presetting can be made before editing the program.

Softkey SET BLOCK NUMBER is used to preset block number generation.

Now press this softkey:

| Set block number |

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.
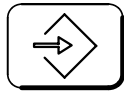
Press the INPUT key to terminate the program block. An end of block character (L_F) is automatically created.

Generate
block number
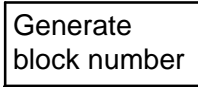
If you press the GENERATE BLOCK NUMBER softkey, the system adds the block number difference to the block number and inserts the result at the cursor position.

You must save the part program after you have completed editing.
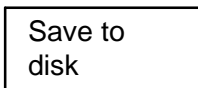
Save to
disk

The softkey SAVE TO DISK and

(in the softkey bar extension (with the ETC key))
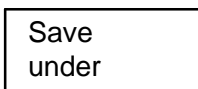
Save
under

the softkey SAVE UNDER are used to save the edited program on the hard disk.

This function is described in 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).

Caution: 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.

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.

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.

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/CO-LORS).

- 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.
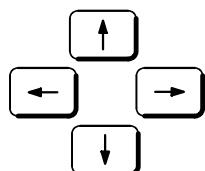
You can shift the window contents in the direction you want by using the CURSOR KEYS.

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	The following search methods referred to • a block in a program • a tool change and • a main block are provided. 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.
PROGRAM MODIFIC.	Program modification: You can set the skip levels here.
R PARAMETERS	Display of current R parameter values.

PARAMETER SIMULATION	This softkey calls a screen in which you can set the simulation parameters. For example, you can parameterize:
	
	• Transmit instead of 2D/3D transformation:
	  The simulation replaces a 2D or 3D transformation with Transmit.
	
	• Keep traversing paths after editing:
	  The traversing paths remain unchanged after you edit the program again or select a new channel.
	
	• Keep R parameters after editing:
	  The R parameters remain unchanged after you edit the program again.

Softkey	Description

PARAMETER SIMULATION	The following functions are assigned to the softkeys:
	
	• CHANNEL+ or CHANNEL–: Select channel to be simulated.
	
	• COLORS: Set colors for tool paths according to tool (turning technology).
	
	• TIMES: Set time period for tool changes and M/H functions for workpiece program runtime.
	
	• 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.
	
	• M/H FUNCTIONS: Call a screen in which PLC functions can be parameterized. The following parameters can be set:
	  – Tool change identifier
	    This setting is required for the SEARCH function and tool change time.
	  – Main spindle active
	    Sets the main spindle by means of an M/H function to which the feedrate is referred (not on power tools).
	  – 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).
	  – 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.
	  – 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.
	
	• 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.

Horizontal softkeys   You can call the following softkeys once you have activated START:

Softkey	Description
STOP	Stops the current part program.
$\rightarrow$ N	Suppresses the screen displays until the next block number is reached.
$\rightarrow$ TOOL	Suppresses the screen display until the next tool change is carried out.
$\rightarrow$:	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.
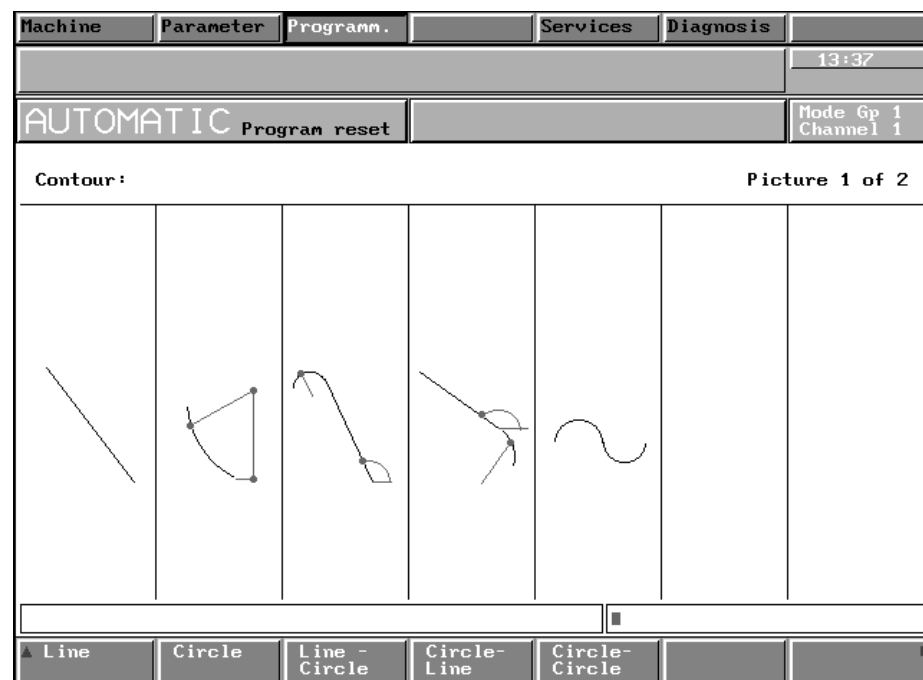
Fig. 5.6   Contour definition programming in the PROGRAMMING area

Selecting the contour definition
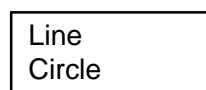
| Line Circle |

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.
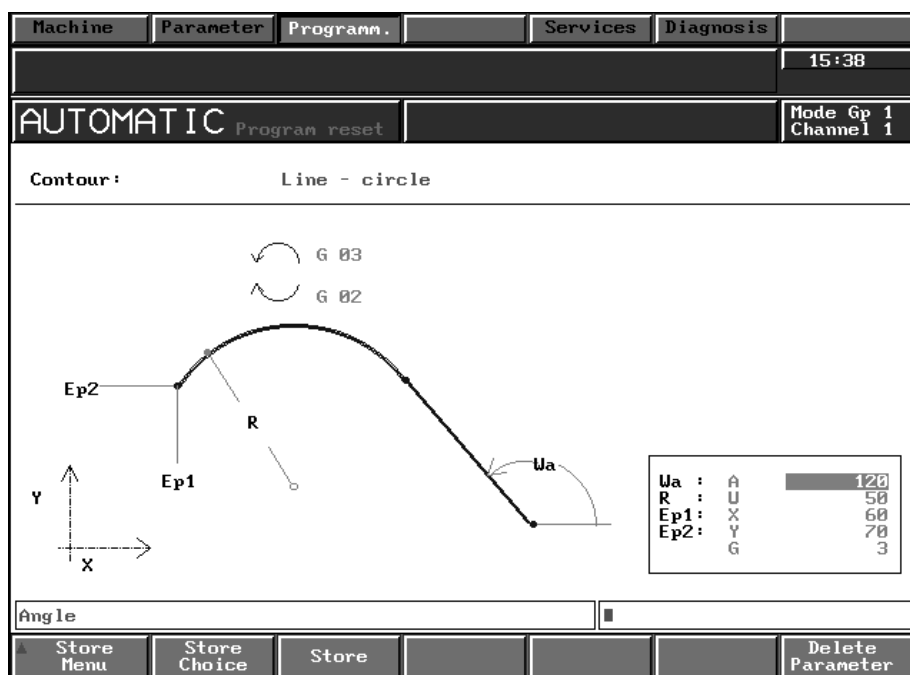
Fig. 5.7   Line – circle geometric contour programming
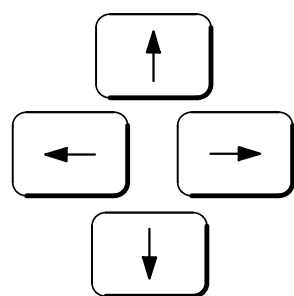
Using the cursor keys, you can choose any input field, for example if you wish to subsequently modify geometric values for the selected contour.

Note
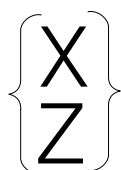
Watch out for input fields in which several values are marked in reverse video in curly brackets on the screen ("X" and "Z" in the example).

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

Drilling and
milling

Press the softkey for the cycle required (e.g. turning).

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.
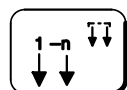
Switch over to the desired channel.

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.

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)

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.

Precondition The PROGRAMMING/EDIT NC area is selected. The main program or subroutine overview display has been called up.

Operating sequence

| %LF | ... |

Enter the name of the program whose access rights you want to change (or select the program in directory NCK with the cursor) and

| +' | X |

Modify rights

insert an access right with +X (executable program), or

| −^ | X |
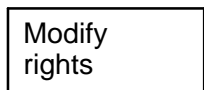
Modify rights

remove an access right with −X (executable program).
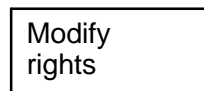
Example A program has the following access rights:

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.

> The access rights are **not** 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.

5−19

5.1.8.2 Copy program

Operating sequence

Copy

Select the source program with the cursor and press the softkey COPY to copy a program.

The following screen form appears:

Fig. 5.8 Copy MPF/SPF 01

The directory and the selected program (the program to be copied) are automatically entered in the line "Copy:".

In the example: Program MPF1 is to be copied as MPF11.

The cursor is placed on directory MPF.
This selects the directory and the new program number can be entered.

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.

5.1.8.4   Delete program

Deleting with the data selector

Operating sequence   Select the program that you wish to delete with the cursor

| Machine | Parameter | Programm. | | Services | Diagnosis | |

```
                                                                    13:47

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       LengthMark      Comment in the 1st block/date
  ..
  MPF1          part          60       -rwx   Test
  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:                685500   characters  1371   sectors
  Delete data ?                                     ▮
  ▲ Edit      Modify     Copy      Set block   Rename      Delete    Save
  ┋           rights               number                            to disk
```

Fig. 5.10   Deleting NCK programs

| Delete |

Press the softkey DELETE.

The following message appears: "Delete data?"

If you press the softkey DELETE again the program that you have selected is deleted.

Deleting by entering the program number

Operating sequence

%LF   ...   The program with the number entered is deleted by the control.
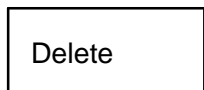
Delete   or

%LF   ...   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
Delete   deleted

                   or
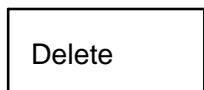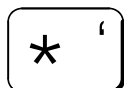
%LF   ...   you can use the question mark to represent an individual character in
                   the program name.
                   e.g. MPF?
⇧   6?   All of the main programs with a single-digit program number are deleted,
                   i.e. MPF0 to MPF9
                   MPF2??
                   The programs MPF 200 to 299 are deleted.
Delete
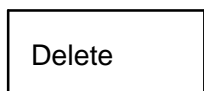
                   You can also delete areas of a program by entering the start and end of
                   the area and pressing the delete softkey. For example, if you enter

                   % 700␣%710 or
                   %700=%710

                   all of the main programs between %700 and %710 are deleted.

                   ┃▶ Cycles must be enabled before deletion. ◀┃
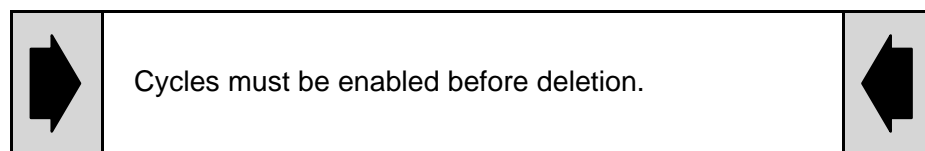
Note   As long as a part program has been selected in the editor (EDIT NC), it
                   cannot be deleted. The program can be released for deletion by
                   selecting another program.

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).

Load cycles   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").

Move cycles   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.

Move subroutine   A subroutine can be loaded as a cycle in a local or global workpiece.
to cycle   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 **not** 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.

Note   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.

Creation
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:

Generation
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.

Import
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.

Starting the converter
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 **always** 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.

Call

You can call the program converter by making the following inputs:
In the PROGRAM\WORKPIECES area, select 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.

Loading in
AUTOMATIC

Loading a part program in AUTOMATIC mode
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.

Loading Services

Loading programs in Services/NC
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.

Display

Displaying conversion errors/conversion warnings
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 <wkst>" 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".

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 l 1 2   16002=1   Message "Conversion warnings in work-piece

                                                                         <wkst>"

Entry l 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 KONVF0.INI

Parameter	Description
;The file KONVF0.ini is divided into three indi- ;vidual 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 KONVF0.ini is divided into three indi- vidual 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, Lan- ;guage, 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
;0 Pocket cal. type dec. point input 1000. –> 1000/1000 –> 1000 ;1 Former type decimal point input 1000 .–> 1000/1000 –> InputWithPoint=1	See input format in FANUC program
;Selection of Inputsystem SIN 840C (Default) ;0 SIN 840C inputsystem Metric ;1 SIN 840C inputsystem Inch MD5002_4=0	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. Calculation of setting for parameter MD 5002.4: Edit the following program sequence in MDA mode and then execute: @302 R0 K5002 K4 Now read out the value of R parameter R0 and enter it in parameter MD 5002.4.
;Selection of Inputsystem FANUC (Default) ;0 FANUC inputsystem Metric ;1 FANUC inputsystem Inch Inch=0	
;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	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. Calculation of value for "Diameter" parameter: Edit the following program instruction in MDA mode and then execute: @302 R0 K5011 K7 @302 R1 K5011 K5 Now read out the values of R parameters R0 and R1 and calculate the entry for "Diameter": 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 transfer-ring the source code of the FANUC program as a comment into the converted SINUMERIK 840C program. These comments are not dis-played in the alarm display text field during part program processing. **Example:** Parameter semicolon= 0 FANUC SINUMERIK 840C G50 S1250 G92 S1250 **Example:** 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 implemen-ted by FANUC 0T codes G70 to G73. It is sto-red in the main program after the call for ma-chining cycle L95.
;Languages ;0 Deutsch ;1 English ;2 Español ;3 Français ;4 Italiano Language=1	Converter messages are output in the selec-ted language.
Minimum subprogram number MinUN_V=101	Lowest subprogram number for contour defi-nition 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 defi-nition 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 (increm.) N735=0 ;G30 - 2nd reference position Z axis (increm.) N736=0 ;G30 P3 - 3rd reference pos. X axis (increm.) N780=0 ;G30 P3 - 3rd reference pos. Z axis (increm.) N781=0 ;G30 P4 - 4th reference pos. X axis (increm.) N784=0 ;G30 P4 - 4th reference pos. Z axis (increm.) 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 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 se-conds	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$_F$ @714 L$_F$ @361 R200 K @361 R201 K2 L$_F$ @171 R200 Xref1 Kn @121 R201 Zref1 K1 @100 Kn+1 L$_F$ N1100 (G27: reference point not reached) L$_F$ @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.

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+∆ref2 (3,4)	0	
16–1	End cutting (delete distance to go)	G31			0	
	Thread cutting lon-gitudinal, face, co-nical	G33 Wxxxx Fffff G33 Uxxxx Fffff G33 Xxxx Zxxx Fffff		G33 Zxxxx Kffff G33 Xxxxx Iffff G33 Xxxx Zxxx Kffff	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 Wwwww	yes	G58 Xxxxx Zzzzz G58 G91 Uuuuu Wwwww	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

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 allowed in block	SINUMERIK 840C	G group, modal, basic setting	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	

Page	Function	FANUC 0T G CODE A / B / C G functions listed singly below are valid for all 3 axes.	Only these words allowed in block	SINUMERIK 840C	G group, modal, basic setting	Comment
	Feed mm/rev.	G95		G95	5,M,G	
	Absolute dimensions	G90		G90	3,M,G	
	Incremental dimensions	G91		G91	3,M,G	
	Return to start plane	G98			11	
	Return to retraction plane	– / G99 / G99			11	
	Cylinder interpolation	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 degrees. The spindle is positioned 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 degrees. The spindle is positioned 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 interpolation	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 cutting rate	Sxxxx		Sxxxxx		
	Tool selection	Txxyy 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 PppppIIII	no	Xxx Zxx LIIII Ppppp		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

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		Hxxxxxxx		Value range for Fanuc: –99999999 to +99999999
	Comment	(*Comment*)		(*Comment*)		Comment text in round parenthesis
	Insert chamfer Z–>X	G1 Zxxxx Iyyyy		G1 Zxxxx B–yyyy		Sign of yyyy specifies direction with Fanuc
	Insert chamfer X–>Z	G1 Xxxxx Kyyyy		G1 Xxxxx B–yyyy		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

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.

> The menu storage function is always active on the
> SINUMERIK 840C, i.e. the menu or directory last
> selected in an area is displayed when the area is
> reselected. If you are not in the LOCAL/GLOBAL
> directory after you have pressed the PROGRAMMING
> softkey, switch over to this directory first. Position the
> cursor on the double dot 💬 and press the INPUT key
> until you have reached the basic display. The double dot
> 💬 is then replaced by a hyphen.

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.
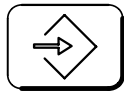
Press the INPUT key.

Press the NEW softkey. You are requested to enter a name. Enter the
name for your workpiece using the alphanumeric keyboard.

Machine	Parameter	Programm.	Simulation	Services	Diagnosis	
						15:11

New: Select data type

/PC/User/LOCAL
Data type:
Workpiece

Name: SHAFT1

| | | | | | | OK |
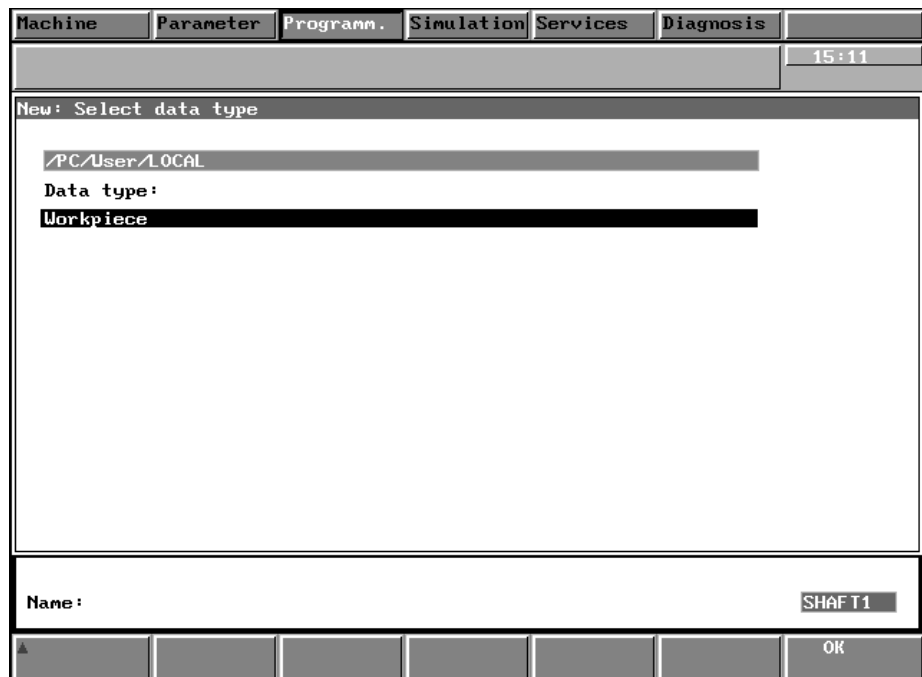
Fig. 5.11   Creating a workpiece
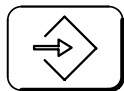
and conclude with the INPUT key.
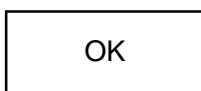
Press the OK softkey to transfer the name into the directory.

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:

```
┌──────────┬───────────┬──────────┬──────────┬──────────┬──────────┬────────┐
│ Machine  │ Parameter │ Programm.│          │ Services │ Diagnosis│        │
├──────────┴───────────┴──────────┴──────────┴──────────┴──────────┴────────┤
│                                                                    13:49   │
├───────────────────────────────────────────────────────────────────────────┤
│ New: Select data type                                                      │
│                                                                            │
│    /PC/User/LOCAL                                                          │
│    Data type:                                                             │
│    Workpiece                                                              │
│                                                                            │
│                                                                            │
│                                                                            │
│                                                                            │
│                                                                            │
│                                                                            │
│                                                                            │
│                                                                            │
├───────────────────────────────────────────────────────────────────────────┤
│ Name:                                                              PART_1  │
├──────────┬───────────┬──────────┬──────────┬──────────┬──────────┬────────┤
│ ▲        │           │          │          │          │          │  OK    │
└──────────┴───────────┴──────────┴──────────┴──────────┴──────────┴────────┘
```
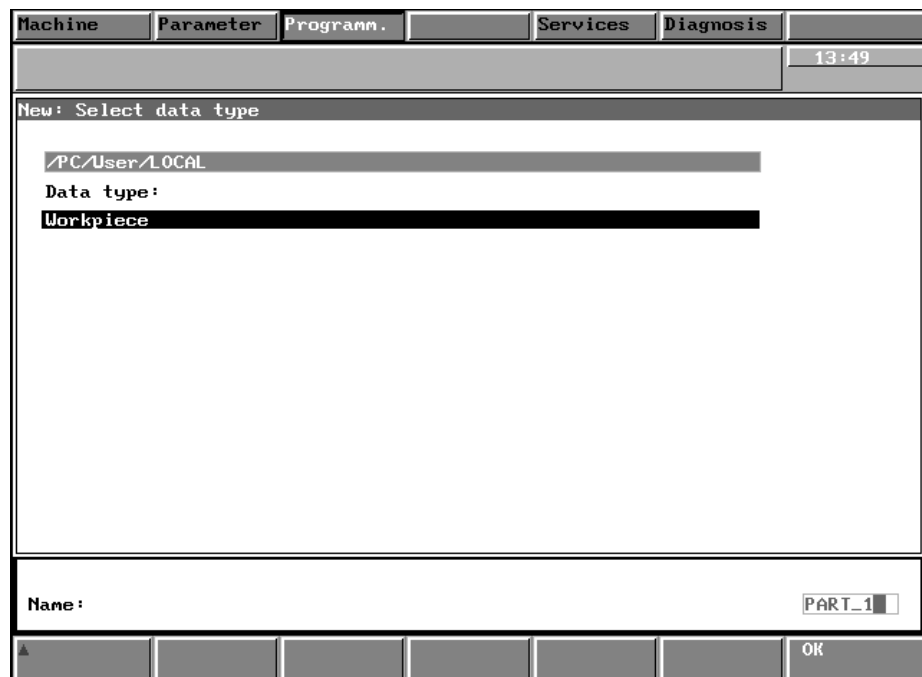
Fig. 5.12   Selecting the file types for editing in the PROGRAMMING area
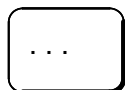
The data type "part program" is preselected; enter the file name with the alphanumeric keyboard in the input screenform and
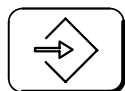
acknowledge with the INPUT key.
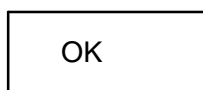
Store the file in the directory by pressing the OK softkey. The ASCII editor is displayed on the CRT. The part program can be edited immediately (see Section "Description of the ASCII editor").

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.

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
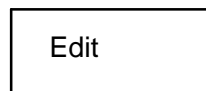
Edit   press the EDIT PC softkey.
A working field is displayed in which you can enter your part program via the keyboard. Operation of the ASCII editor is described in more detail in the "Description of the ASCII editor" section.
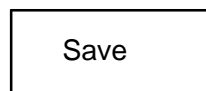
Save   After the program has been entered press the SAVE softkey and the edited program is stored on the hard disk.

If you do **not** save and press the RECALL key instead, a warning message is output: "File has been edited. Lose the changes?".
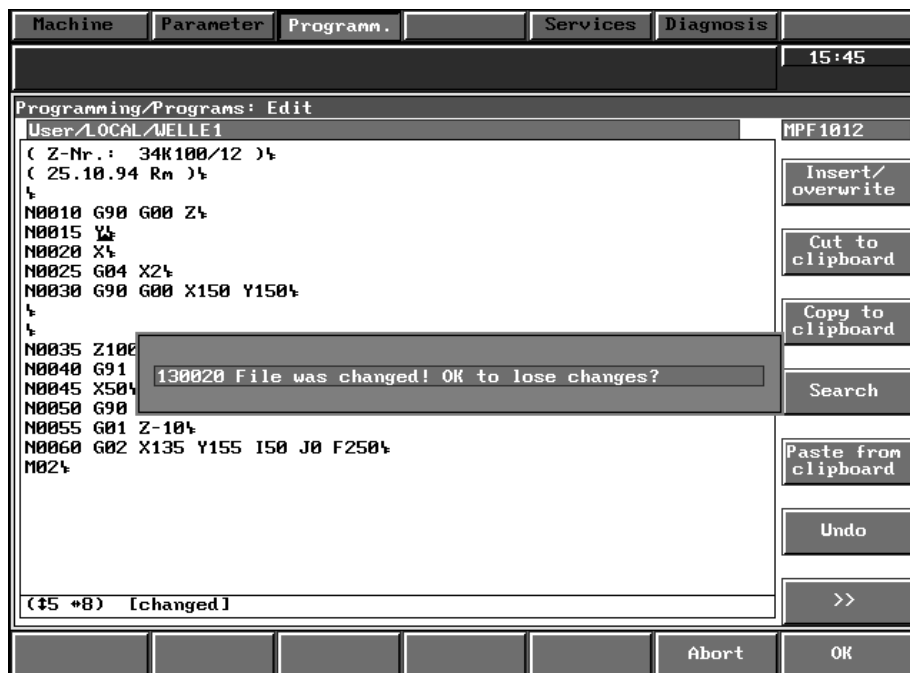
Fig. 5.13   Edit MMC, abort with RECALL

You can now select one of the two softkeys ABORT or OK.

OK
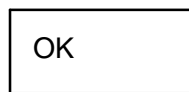
Press the softkey OK to exit the edit mode and the changes you have made are **lost**.
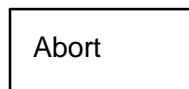
Abort

If you press the ABORT softkey, you remain in the Edit mode. You can carry out further modifications or store the file.

> If you have created or modified a program, it must be loaded into the processing memory if you want to use it for machining.
> The following functions are used for this purpose:
>
> • Select workpiece (MACHINE/AUTOMATIC area)
>
> • Load NC (SERVICES/NC/LOAD area)
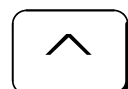
Press the RECALL key to return to the directory.

Note

Proceed as follows when editing R parameters, setting data, tool offsets and zero offsets:
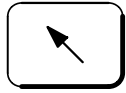
After pressing the NEW softkey switch to the "Data type" window.

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.

| New |

Press the NEW softkey; the following display appears:

```
┌──────────┬───────────┬──────────┬─────────────┬──────────┬───────────┬──────┐
│ Machine  │ Parameter │ Programm.│             │ Services │ Diagnosis │      │
├──────────┴───────────┴──────────┴─────────────┴──────────┴───────────┴──────┤
│                                                              ┌──────────┐    │
│                                                              │  13:49   │    │
│ New: Select data type                                        └──────────┘    │
│                                                                              │
│   ┌────────────────────────────────────────────────────────────────────┐    │
│   │ /PC/User/LOCAL/PART_1                                               │    │
│   └────────────────────────────────────────────────────────────────────┘    │
│   Data type:                                                                 │
│  ┌───────────────────────────────────────────────────────────────────────┐  │
│  │ Part program                                                          │  │
│  └───────────────────────────────────────────────────────────────────────┘  │
│    Job list                                                                  │
│    Comments                                                                  │
│    Work schedule                                                             │
│    Tool list                                                                 │
│    R parameters (RPA)                                                        │
│    Setting data (SEA)                                                        │
│    Cycle setting data (SEA4)                                                 │
│    Zero offsets (ZOA)                                                        │
│    Tool offsets (TOA)                                                        │
│    IKA number                                                                │
│    Curve pointer                                                             │
│    Curve points                                                             │
│                                                                              │
│                                                                              │
│                                                                              │
│   Name:                                           ┌──────────────────┐       │
│                                                   └──────────────────┘       │
├────────┬────────┬────────┬────────┬────────┬────────┬────────┬──────────────┤
│ ▲      │        │        │        │        │        │        │     OK       │
└────────┴────────┴────────┴────────┴────────┴────────┴────────┴──────────────┘
```
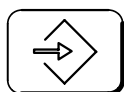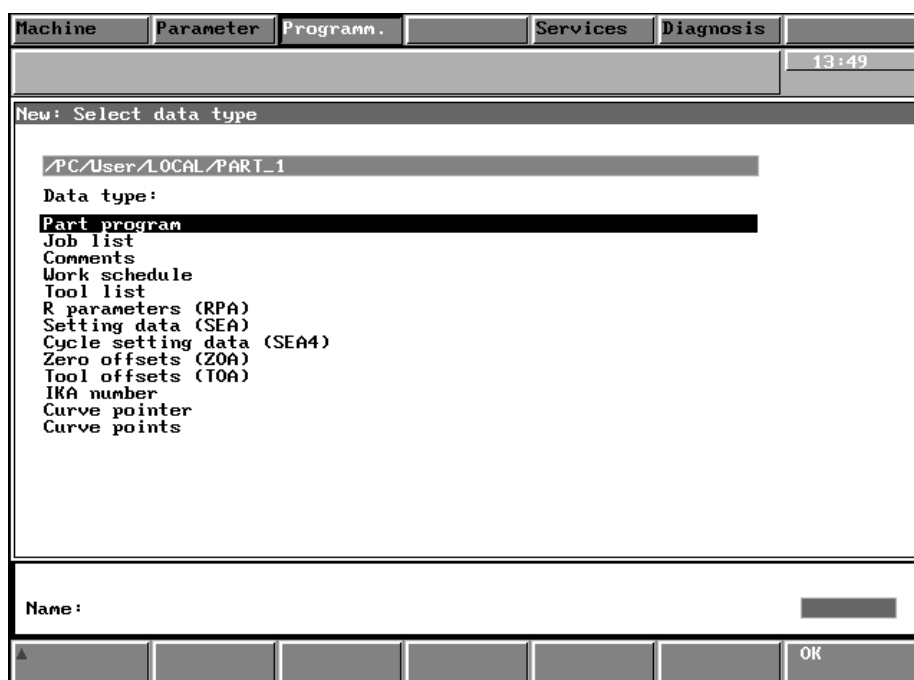
Fig. 5.14   Selecting the file types for editing in the PROGRAMMING area
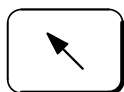
Press the window switchover key to switch to the "Data type" window. The active window is marked (yellow frame).
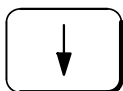
Select the data type "Job list" with the cursor keys.
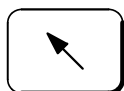
Switch to the input screenform and

enter the name with the alphanumeric keyboard.
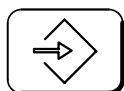
Accept the name by pressing the INPUT key and the

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.

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.
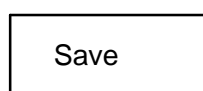
Press the SAVE softkey to store the edited job list on the hard disk.

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

| Edit |

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.

```
  Machine     Parameter  Programm.             Services  Diagnosis
                                                            09:22

Programming/Programs: Edit
User/LOCAL/WELLE1                                          JOB
( Loeschanweisungen fuer NCK-Speicher: )▲               Insert/
( CLEAR *           loescht alle MPF, SPF        )       overwrite
( CLEAR MPF[0,9999]   loescht alle MPF's von MPF0 bis MPF9999 )
( CLEAR SPF[123,4711] loescht die SPF's von SPF123 bis SPF4711)
( CLEAR MPF...        loescht ein MPF            )       Cut to
( CLEAR SPF...        loescht ein SPF            )       clipboard
(CLEAR *▌

( Ladeanweisungen: )▌                                   Copy to
▌                                                       clipboard
( LOAD . *          laedt alle Dateien(MPF,SPF,TOA,ZOA,RPA,SEA))
(                   unter dem Werkstueck zu dem die Jobliste  )  Search
(                   gehoert                      )
(                   Entspricht LADEN OHNE JOBLISTE !!!  )
LOAD . *▌
▌                                                       Paste from
( LOAD pfad name  | pfad: wo die Daten liegen    )      clipboard
(                 |       (z.B. local/werkst1, global/cycle, )
(                 |          global/data )
(                 | name: Name der Datei(MPF,SPF,TOA, ..)  )   Undo
(                 |       oder *(laedt alles unter dem  )
(                 |       angegebenen Werkstueck )  )
(LOAD GLOBAL/WELLE1 MPF2▌
(‡1 ∗1)                                        ▼            >>

▲ Save    Save and                     Wait mark  Block
          load                                    number
```

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.
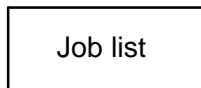
Job list

Press this softkey and the following display appears:
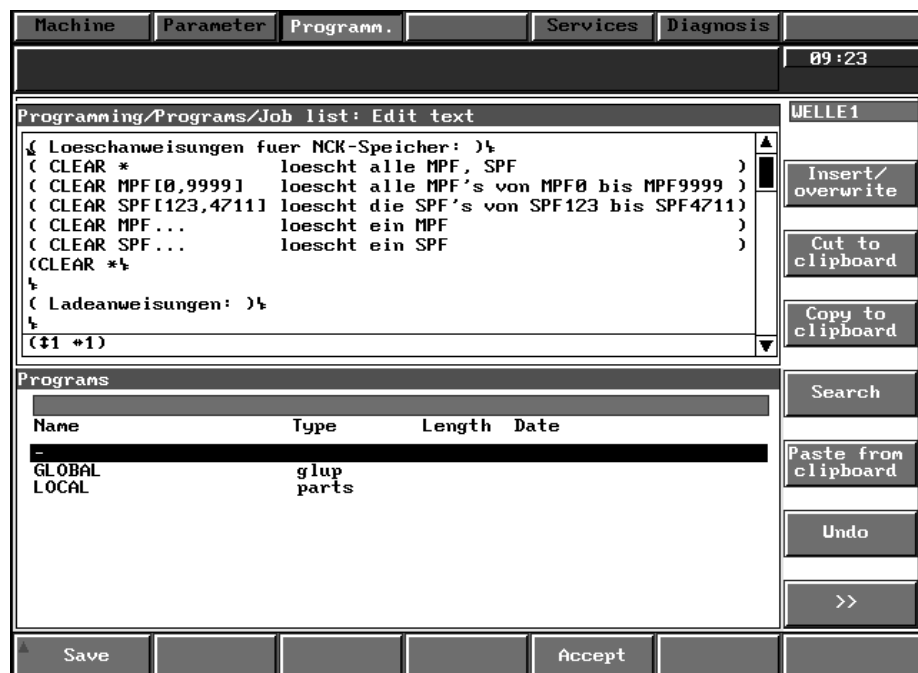
Fig. 5.17   Job list function via job list softkey
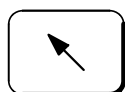
With the window switchover key you can switch between the two windows. The active window is always marked.

- 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.
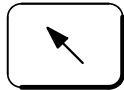
Switch to the upper window, if required. Move the cursor to the position in the file where the name is to be entered, e.g.
LOAD▲ ....
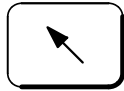
Switch to the lower window and page with the cursor keys and the INPUT key through the directories until you have reached the desired part program file.
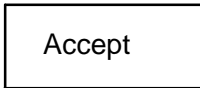
Accept

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
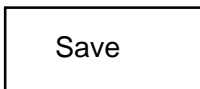
Save

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$_{\sqcup}$name L$_F$

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$_{\sqcup}$MPF7

Deleting the SPF 1 to 999:
CLEAR$_{\sqcup}$SPF[1, 999]

Deletion of all MPF,SPF:
CLEAR$_{\sqcup}$ *

Caution! The syntax is mandatory. Observe the position of the blanks $_{\sqcup}$.

Load instructions   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.

Command syntax   LOAD␣pfad␣name L$_F$

                           "path" comprises:

                           • Path name and

                           • Workpiece name.

                           Use the following command set for "path":

                           • LOCAL/workpiece name

                           • GLOBAL/workpiece name

                           . (full stop) means that the data of the current workpiece from the job list
                           selected for processing is transferred.

                           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 of a workpiece are loaded into the part
                           program memory (NCK).

                           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
                           (TOA<u>1</u>).

Examples   Load all files of workpiece SHAFT1 from the LOCAL directory:

                           LOAD␣LOCAL/WELLE1␣ *

                           Load file WKZK (type toa) of workpiece WELLE1:

                           LOAD␣LOCAL/WELLE1␣ WKZK␣CH=1

                           Load files SPF7 to SPF19 of workpiece WKST1 from the LOCAL
                           directory:

                           LOAD␣LOCAL/WKST1␣SPF[7,19]

                           Load file SPF1 of workpiece ZYC1 from the global directory:

                           LOAD␣GLOBAL/ZYC1␣SPF1

                           Load all files of the current workpiece:

                           LOAD␣.␣*

                           Caution! The syntax is mandatory. Observe the position of the blanks ␣.

> ⮕　Avoid the following combinations:
> 　　LOAD GLOBAL / <dir.>　　MPFx
> or　　LOAD GLOBAL / <dir.>　　SPFx, programs
> MPFx or SPFx in directory <dir.> under GLOBAL
> also exist in another (different) workpiece under
> LOCAL. See also the explanation below.　　⬅

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$_F$

path:	USER or STANDARD
name:	Cycle name
	SPF[n, m]　　(cycles SPFn...SPFm)
	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$_{\sqcup}$name$_{\sqcup}$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$_{\sqcup}$MPF7$_{\sqcup}$CH = 1 L$_F$

Select function "Execution of a part program" from the hard disk:

SELECT$_{\sqcup}$name$_{\sqcup}$CH=nr DISK L$_F$

Select function "Execution of part program MPF123" from the hard disk:

SELECT$_{\sqcup}$MPF123$_{\sqcup}$CH=1$_{\sqcup}$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 $_{\sqcup}$.

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.
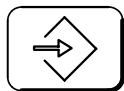
Select a workpiece.
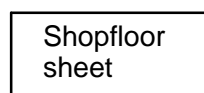
| Shopfloor sheet | Press the SHOPFLOOR SHEET softkey; an overview of all existing part programs under the selected workpiece is displayed. |

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.
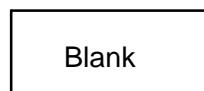
| Blank | Press the following softkeys BLANK or |

| Work schedule | WORK SCHEDULE or |

| Comments editor | COMMENTS EDITOR or |

| Create TOA file | CREATE TOA FILE. |
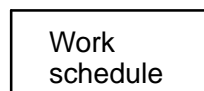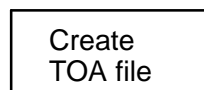
If you press the BLANK softkey, the data of the selected workpiece blank is displayed.

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.).

> In the case of a DIN type program, the data under the softkeys BLANK and WORK SCHEDULE is not displayed. A TOA file can only be created for a program produced graphically.
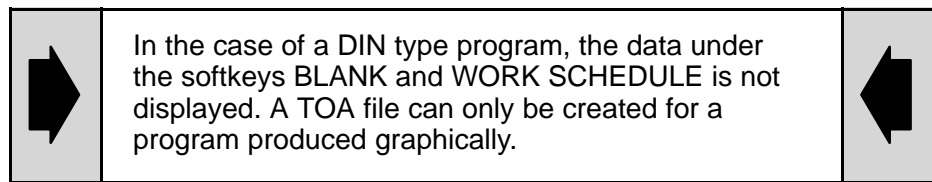
If you press the COMMENTS EDITOR softkey, you are provided with the ASCII editor with which you can create a comments file to the workpiece displayed.

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.
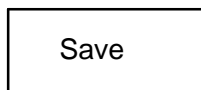
| Save |

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
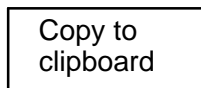
| Copy to clipboard |

By means of these softkey functions you can copy workpieces and files from the hard disk data management and paste it from another position.

| Paste from clipboard |
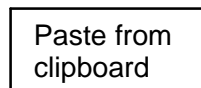
Use the INPUT key to page to the source file. If you press the COPY TO CLIPBOARD softkey, the selected file/directory is entered in a buffer. Then select a target directory. If you press the PASTE FROM CLIPBOARD softkey an input screenform is displayed in which you can enter a new name for the workpiece 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.

| Erase |
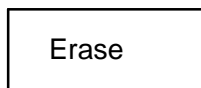
With the cursor keys or the INPUT key select the workpiece or the file to be deleted. After you have pressed the softkey you are asked by an interactive form whether you really wish to delete. Press the OK softkey to confirm or the RECALL key to abandon.

5.2.6 Description of the ASCII editor
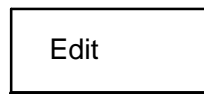
Edit

The following display is provided after you have called up the ASCII editor by pressing the EDIT softkey in directories LOCAL and GLOBAL:

```
┌──────────┬──────────┬──────────┬─────────┬──────────┬──────────┬────────┐
│ Machine  │ Parameter│ Programm.│         │ Services │ Diagnosis│        │
├──────────┴──────────┴──────────┴─────────┴──────────┴──────────┴────────┤
│                                                              ┌─────────┐ │
│                                                              │ 09:25   │ │
├──────────────────────────────────────────────────────────────────────────┤
│ Programming/Programs: Edit                                              │
│ User/LOCAL/STANDARD                                          MPF13      │
│ ( MPF   13   )↳                                      ▲    ┌──────────┐  │
│ ( Zeichnungsnr.:   145/23B  )↳                            │ Insert/  │  │
│ ( Erstellungsdatum:▲)↳                                    │ overwrite│  │
│ ( Maschine:        FZ 12  810M/T )↳                       └──────────┘  │
│ ( Bearbeiter:         )↳                                  ┌──────────┐  │
│ (   )↳                                                    │ Cut to   │  │
│ N10 G17 G54 G00 G90 G80↳                                  │ clipboard│  │
│ ( Stations-Nr. 1 )↳                                       └──────────┘  │
│ N15 ( Text/DIN-Code bei WZW V1951 )↳                      ┌──────────┐  │
│ N20 M09 M05 D0 S0↳                                        │ Copy to  │  │
│ N25 G53 Z0↳                                               │ clipboard│  │
│ :30 T1 M06↳                                               └──────────┘  │
│ N35 G54 D1↳                                               ┌──────────┐  │
│ N40 F10 S1000 M03 M08↳                                    │ Search   │  │
│ ( Anfahren Bahnbearbeitung )↳                             └──────────┘  │
│ N45 G00 X16 Y-3↳                                          ┌──────────┐  │
│ N50 Z8↳                                                   │Paste from│  │
│ N55 Z-15↳                                                 │ clipboard│  │
│ N60 Z-20↳                                                 └──────────┘  │
│ N65 G01 G64 X13 F100↳                                     ┌──────────┐  │
│ N70 G02 X10 Y0 I0 J3↳                                     │ Undo     │  │
│ N75 G03 X10 Y0 I-10 J0↳                                   └──────────┘  │
│ N80 G02 X13.25 Y3.25 I3.25 J0↳                       ▼   ┌──────────┐   │
│ (↕3 ↔21)  [changed]                                      │   >>     │   │
├──────────┬──────────┬──────────┬─────────┬──────────┬──────────┬────────┤
│ Save     │ Save and │          │         │ Wait mark│ Block    │        │
│          │ load     │          │         │          │ number   │        │
└──────────┴──────────┴──────────┴─────────┴──────────┴──────────┴────────┘
```

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

↑ ← → ↓
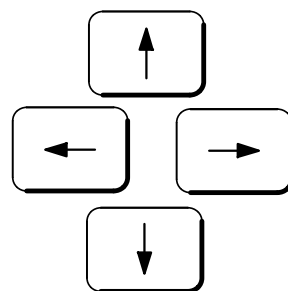
With the cursor keys you move the cursor in the text.

The current line z and column s are displayed using symbols ↕z ↔s below the text window (e.g. ↕10 ↔5 = line 10, column 5).

This lower line is called the status line.

Status line	Other information displayed in the status line:
[Changed]	Displayed if the file has been changed.
Read xx%	These messages show the status
Saved xx%	for reading, saving and replacing
Replace xx%	a file in percent
Load	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.
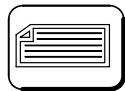
Move the cursor to the beginning/end of the file.

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.
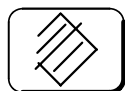
Insert mode: The character to the right of the cursor is deleted.
Overwrite mode: The character under the cursor is deleted.
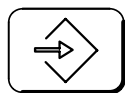
The INPUT key is used to conclude. LF is generated.

Use the vertical softkeys to select functions for text editing.
There is a main menu and a continuation menu.

Main menu	Continuation menu	Confirmation
Insert/ overwrite	Format	
Cut to clipboard	Line number	
Copy to clipboard	Search and replace	
Search		
Paste from clipboard		
Undo		OK
>>	<<	Abort

Change
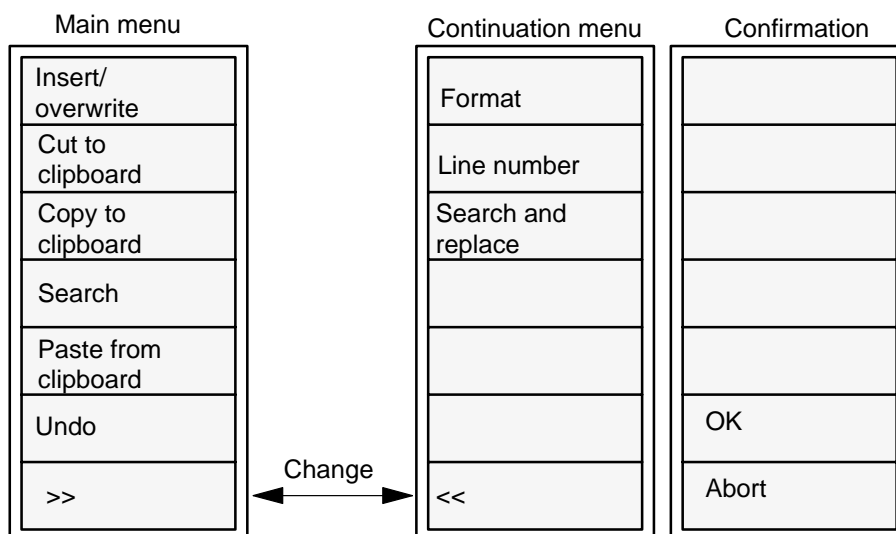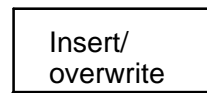
| Insert/
overwrite | switches over between insert and overwrite mode
(Cursor: △ or   ■) |

| Undo | Cancels the last function performed on the text |

| Block number | Inserts a block number at the cursor position |

| Wait mark | If you press the WAIT MARK softkey, the following screenform is displayed: |
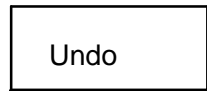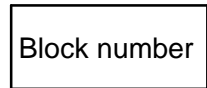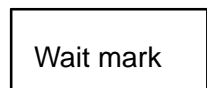
Type	m		Number	1
Chan.	Slide 1	Slide 2	Slide 3	Loader
	_	_		

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).
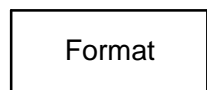
| Format | Displays the form for format parameter specification |

Format:

Font:	small	Block numbers:	no
Foreground color:	red	Steps:	5
Background color:	white	Start 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.
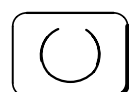
In the selection fields Font, Foreground color, Background color and Block numbers, you can page through the possibilities specified by the operating system.

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.

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.

| Cut to clipboard | Copies selected text into buffer and deletes it subsequently in the file. |

| Copy to clipboard | Copies selected text into buffer. |

| Paste from clipboard | Inserts the deleted/copied text at the cursor position. |
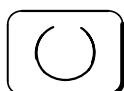
You select the 1st character of the character string you want to cut or copy.
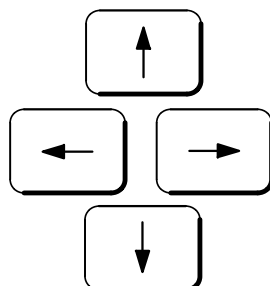
Now you select the required character string with the cursor keys. The selected text is highlighted.

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.
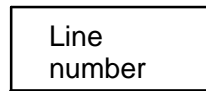
Line number

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.

OK
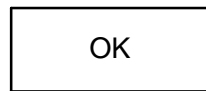
Stores the input parameters and executes the function

Abort

Aborts the function.

Save

After completion of the editing functions press the SAVE softkey to store the edited text on the hard disk.
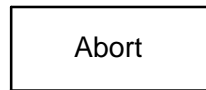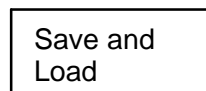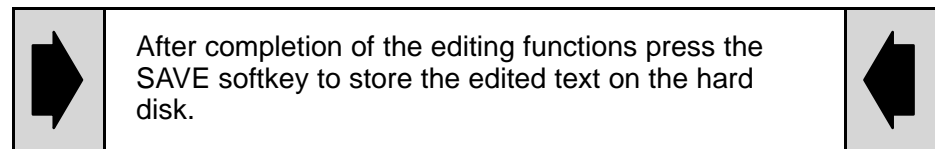
Save and Load

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.
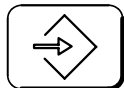
The input is stored in the directory by pressing the INPUT key or

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.
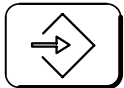
Press the INPUT key to select the next field or

use the arrow keys.
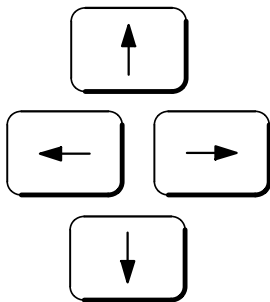
5.2.6.4   Double ASCII editor

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	jobl	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	jobl	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.
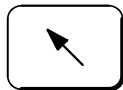
You can switch between the two files with the window switchover key. The active window is marked (yellow frame). You select a file in **both** directories by pressing the cursor keys or the INPUT key.

Press the EDIT softkey. The double ASCII editor is displayed.
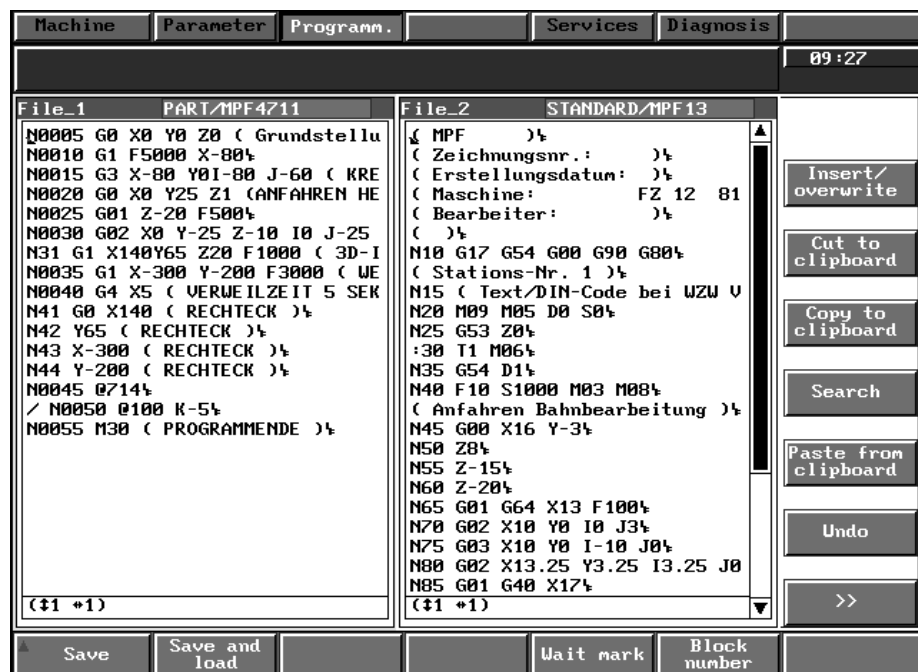
Edit

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

IKA data can be viewed in the PROGRAMMING area. To do this, press softkey VIEW IKA DATA after you have selected file of "**t**ype" IKA..3 with the data selector.

View
IKA data

> You are strongly advised against editing (= processing) IKA data with the EDIT softkey. Without "recalculation" these data will be inconsistent!
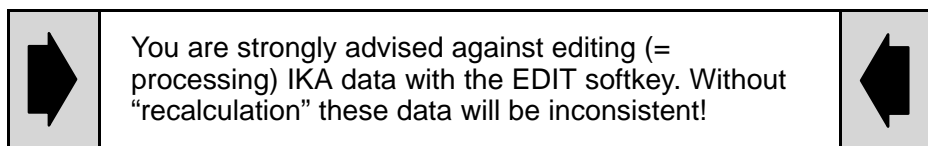
IKA data

The abbreviation IKA stands for **I**nterpolation and **K**ompensation with **T**ables. 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 ma-chine-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

SINUMERIK 840C (BA)

6 Services

| Machine | Parameter | Programm | | Services | Diagnosis | |

Data exchange between NCK and hard disk

Data In/Out to I/O devices

File management in the data management area on the hard disk

6.1 Load NC	**6.2 Save NC**	**6.3 Data input and output**	**6.10 Data management**
6.1.1 Loading work-pieces and programs	6.2.1 Saving NC data	6.3 Data input from an external device	6.10.1 Copy, paste
6.1.2 Loading data	6.2.2 Saving part programs ...	6.4 Data output to an external device	6.10.2 Delete
		6.5 Data input and output with FD−E2	
		6.6 Data input and output with FD−E1	
		6.7 Printing out files	
		6.8 Archive lists	
		6.9 Device settings	

Fig. 6.1   Services

Services

Press this softkey in the area menu bar to select the SERVICES area.

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.

SINUMERIK 840C (BA)

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

NC
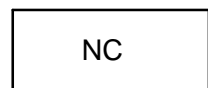
Press the NC softkey and the following display appears:
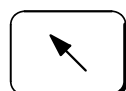
Machine	Parameter	Programm.		Services	Diagnosis	

```
                                                          10:12

NC: User

  User
  Name                Type      Length   Date
  _
  GLOBAL              glup
  LOCAL               parts
  NC
```

▲		Shopfloor sheet		Save Start	Load Start	Error log

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 (seePROGRAMMING 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).

```
NC: User
User
Name            Type          Length        Date
–
NC
LOCAL           parts
GLOBAL          glup
```

```
NC: User
User/LOCAL
Name            Type          Length        Date
..
PART            part
STANDARD        part
PART1           part
PART2           part
SHAFT1          part
TEST            part
```
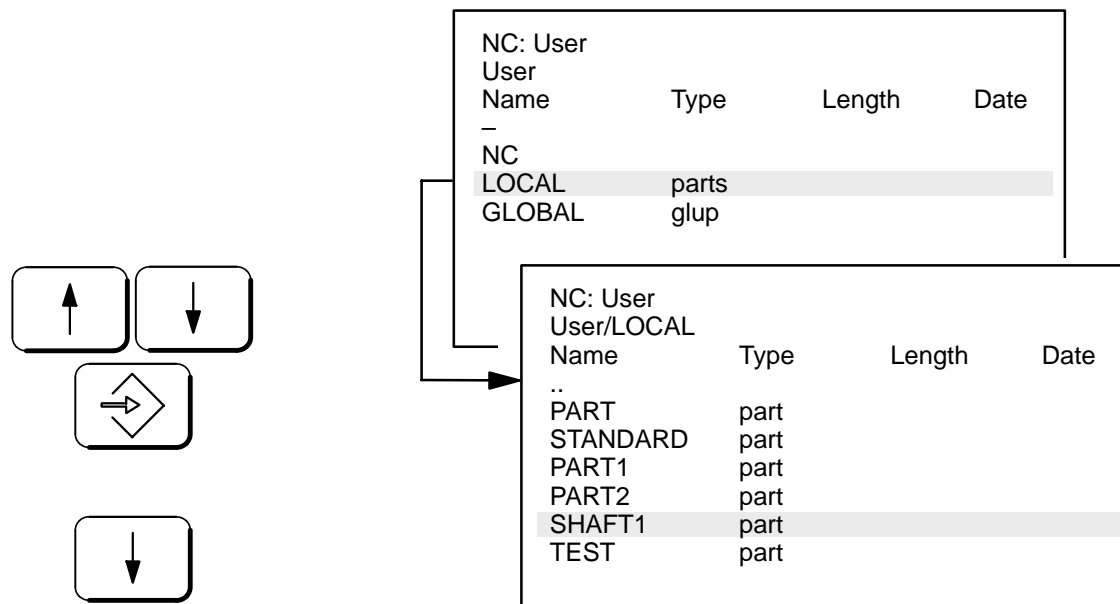
Load
Start

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.

```
NC: User
User
Name          Type          Length        Date
–
GLOBAL
LOCAL
NC
```

```
NC: User
User/LOCAL
Name          Type          Length        Date
..
BOH           part
GEW           part
FLANSCH       part
WELLE1        part
WELLE2        part
```

```
NC: User
User/LOCAL/FLANSCH
Name          Type          Length        Date
..
JOB           jobl          91            09.09.94
MPF123        parp          103           09.09.94
MPF1          parp          5938          01.09.94
MPF2          parp          8010          15.08.94
```
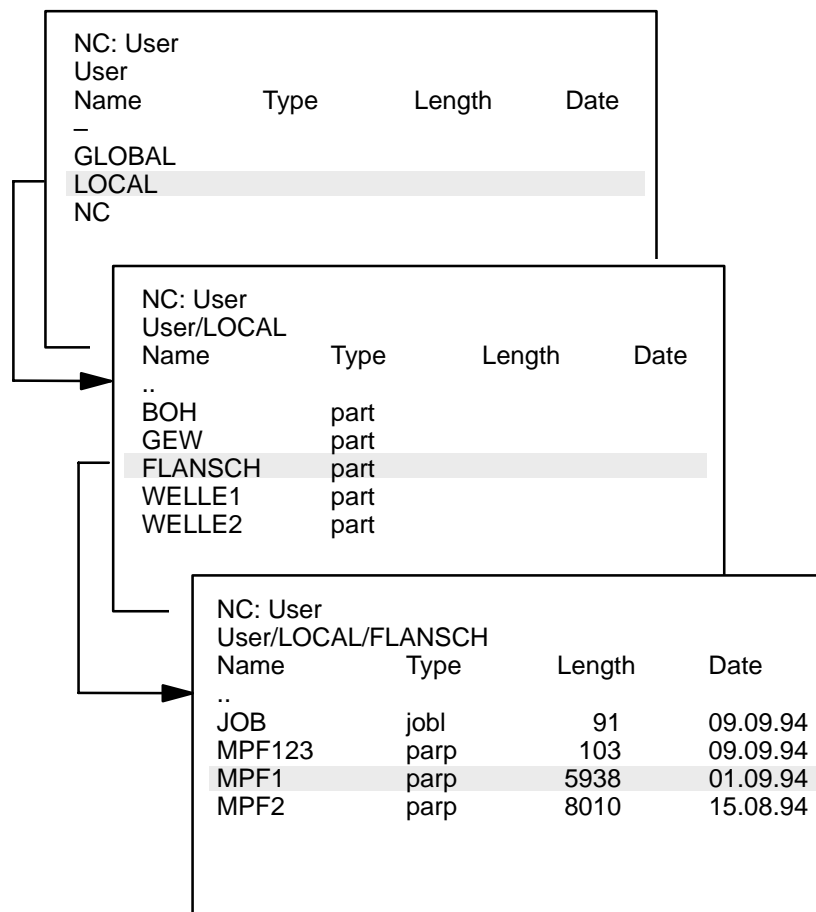
Notes   Workpieces in the MACHINE/AUTOMATIC area can be loaded into the
                 NCK memory by means of the SELECT WORKPIECE function **or** by
                 means of the LOAD NC function in the SERVICES area. If the workpiece
                 has no job list, all workpiece data stored are 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.

> When loading workpieces the following priorities for
> channel interrogation are scanned by the control
> (also with automatic loading of the STANDARD
> workpiece)
>
> 1. Channel specification in job list
>
> 2. Channel specification in file name (last character of
>   the file name
>
> 3. Current NC channel.

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:

```
NC: User
User
Name          Type          Length          Date
..
NC
LOCAL         parts
GLOBAL        glup
```

```
NC: User
User/NC
Name          Type          Length          Date
..
Data
```

```
NC: User
User/NC/Data
Name          Type          Length        Date
..
SEA1          sea           10429         08.04.93
TOA1          toa           13673         08.04.93
RPA0          rpa           12327         08.04.93
RPA1          rpa           10211         08.04.93
```
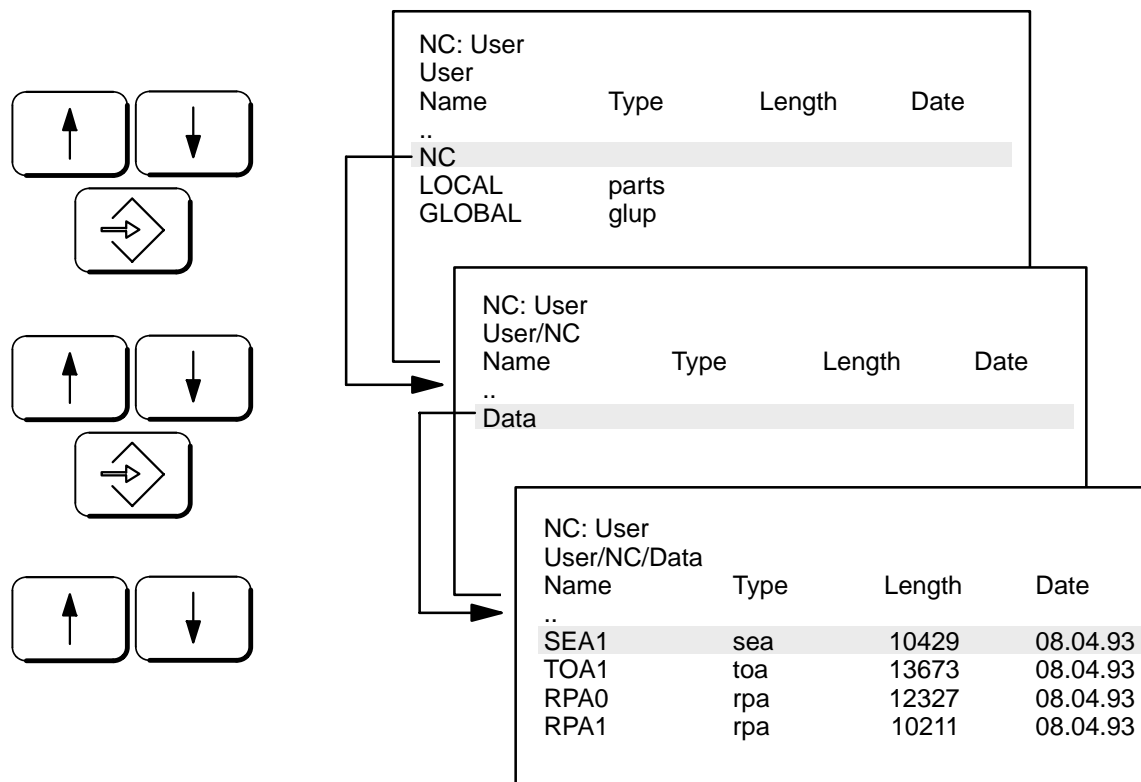
Use the arrow keys to select the file to be loaded into the NCK memory and start data transfer by

pressing the START LOAD softkey (see also "Loading workpieces").

Start
load

If an error occurs during data transfer, the following message appears in a dialog box:

```
110019  Observe error log
```

Press the OK softkey to acknowledge the message.

OK
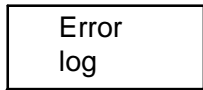
| Error log | If you press this softkey, the faulty directory path and the faulty file are displayed in the error log. |

Press RECALL to exit from the error log

> The SHOPFLOOR SHEET softkey is described in the PROGRAMMING section. Unlike the PROGRAMMING area where the comment can be edited, in the SERVICES area the comment can only be viewed.

> The files identified with TOA, ZOA, SEA, TEA and RPA may not contain any comments. Comments are only possible with MPF and SPF.
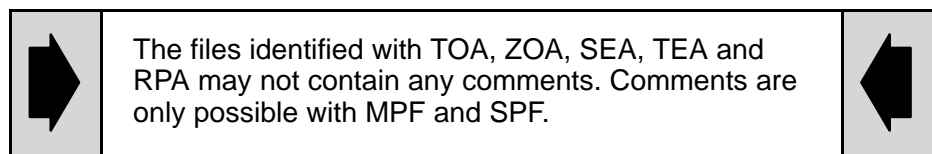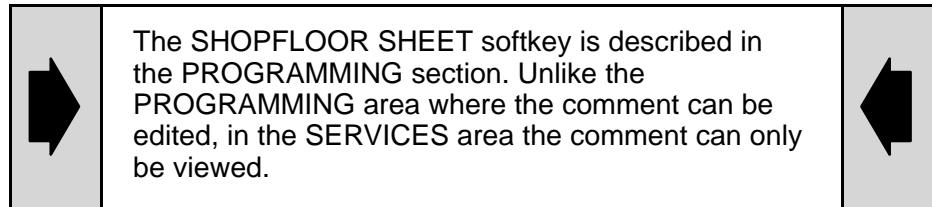
Press RECALL to switch back to the basic display of the SERVICES area.

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

NC
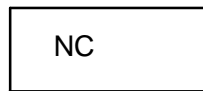
Press the NC softkey. The following selection display appears on the screen:

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




 ▲              Shopfloor        Save    Load    Error
                sheet            Start   Start   log
```

Fig. 6.4   NC save/load
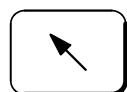
Press the HOME key to switch to the bottom window (NC: User). The active window is highlighted by a yellow frame.

6.2.1   Saving NC data

Operating sequence   Page through the directories in the user branch as follows:

```
NC: User
User
Name            Type          Length        Date
_
GLOBAL
LOCAL           parts
NC              glup
```

```
NC: User
User/NC
Name            Type          Length        Date
..
Data
```

```
NC: User
User/NC/Data
Name          Type          Length        Date
..
IKA1          ika1            4289
GIA           gia            34853
```
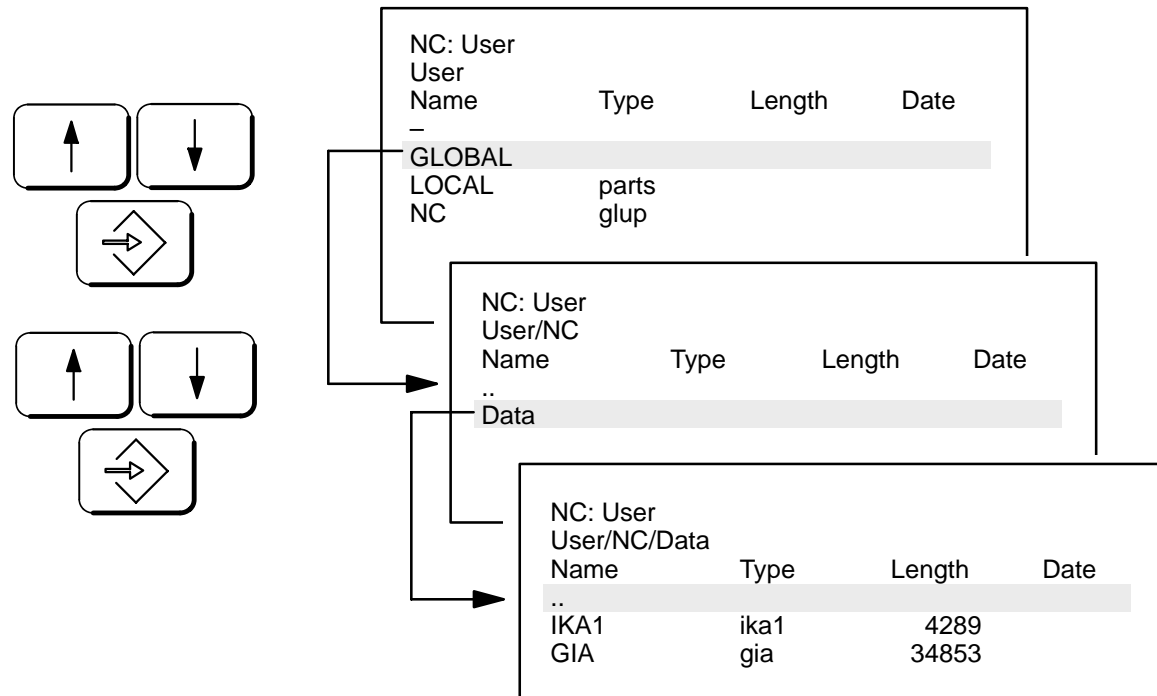
Save
Start

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./TO area    0

    Name PC data           IKA3

    Note :
     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. 6.5   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)

If you try to select other data types offered (TOA, SEA, SEA4, ZOA, RPA, MPF, SPF), the error message "... saving of <...> not allowed here".

These data can only be stored in a workpiece in directory GLOBAL or LOCAL.

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.
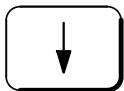
After the data have been selected, press this cursor key to switch to the next input window. Enter the range of selected data to be saved (e.g. IKA3: 0 ... 100) in the FROM/TO input window and conclude with the IN-PUT key.

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.

OK

The following dialog text is displayed:

Transfer from NC to PC in progress!	WKZK1
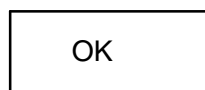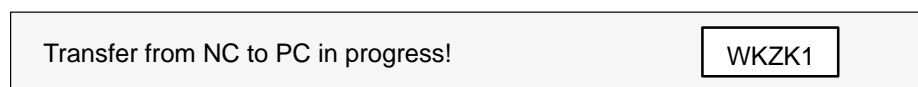
If the data has been stored already in the PC area under the same name, you are asked:

PC data already exists. Overwrite?	TOA 1

Yes
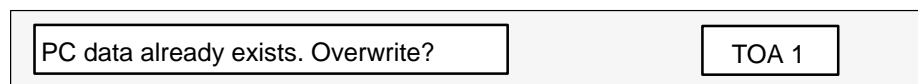
With these softkeys you decide whether existing data on the hard disk is to be overwritten.

No
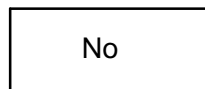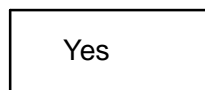
Note

For saving NC data in the NC/Data directory, the password must be set.

Abort

If you press the ABORT softkey during transfer, the transfer is aborted.

> If you interrupt data transfer by means of the ABORT softkey function, incomplete files are stored. It is advisable to delete these files by means of the SERVICES/MANAGEMENT function.

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.
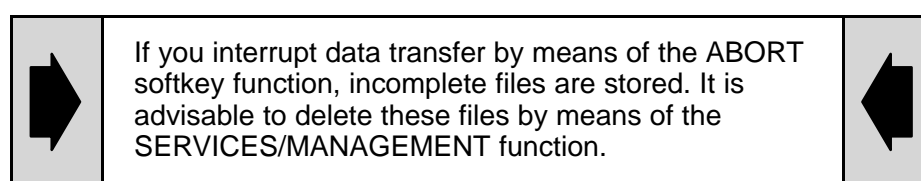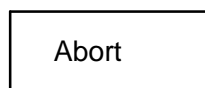
6.2.2   Saving part programs or NCK data in a workpiece in the local or global directory

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
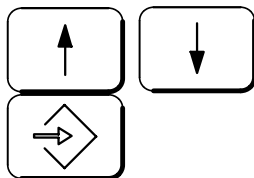
With the cursor keys, position the cursor on LOCAL or GLOBAL and

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

| ▲ | | Shopfloor sheet | | Save Start | Load Start | Error log |

Fig. 6.6   Directory of local workpieces in the SERVICES/SAVE NC area

Using the cursor keys, select the workpiece in which you wish to store the data and
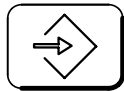
confirm with the INPUT key.
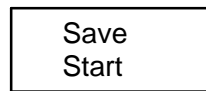
Save
Start

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               TOA tool offsets

    from/to                 0          100

    Channel no./TO area     1

    Name PC data            TOA

     Note :
      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)

> Gearbox interpolation data (GIA) can only be stored in directory NC/Data.

SINUMERIK 840C (BA)
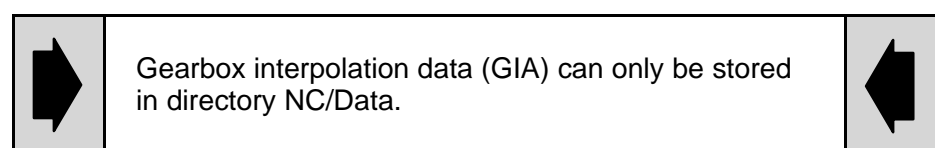
IKA data can be stored both here and under NC/ Data. Please heed the note in Section "Saving NC data" (in NCK/Data directory).
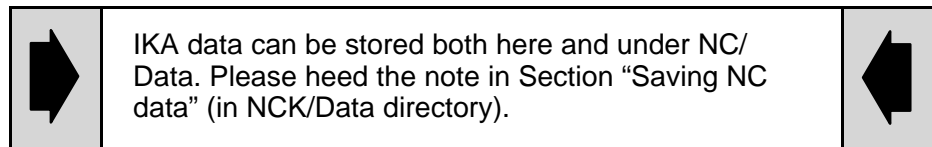
Several part programs can be transferred at the same time.

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.

The STANDARD workpiece, created by the operating system, is in the local directory. The data for this STANDARD workpiece is automatically loaded into the NCK memory on powering up.
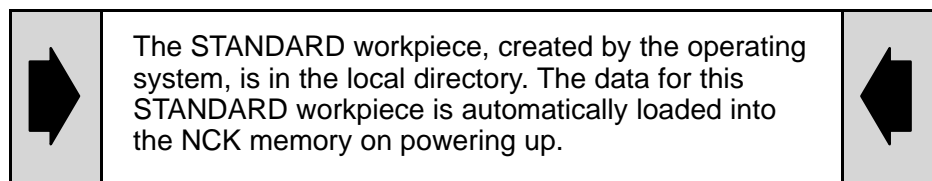
Note   NCK data and part programs can only be stored into the file directories provided for this purpose. If the SAVE NC function is started in another directory level a dialog form with the following message appears on the screen:

110107 NC data/part program cannot be created here
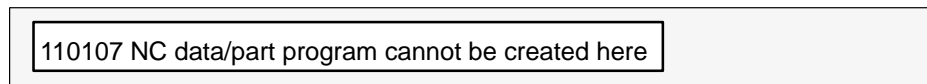
Acknowledge the form with the OK softkey.

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
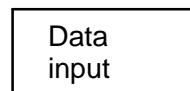
| Data
input |

Press the DATA INPUT softkey and the following interactive screenform is displayed:

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).
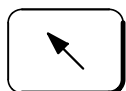
The screenform in the upper window shows the currently parameterized interface and provides a toggle field (selection field).

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:

```
File1              %TOA1
%TOA1              File1
File2              File2
%TOA1
```
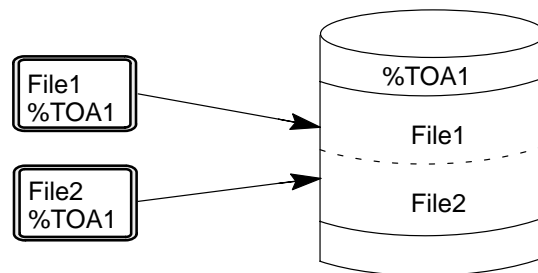
Notes   Data in PC format can be generated and edited externally with PCIN
                 V3.1 and higher, i.e. you can only read-in data in PC format if it has been
                 output from the SINUMERIK 840C. When the data is output, information
                 stating from which directory and 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:

```
Data input: target path
PC
Name          Type        Length      Date
–
User
Buffer
```

```
Data input: target path
PC/User
Name          Type        Length      Date
..
GLOBAL        glup
Startup
LOCAL         parts
NC
```

```
Data input: target path
PC/User/LOCAL
Name          Type        Length      Date
..
PART          part
STANDARD      part
WELLE 1       part
TEIL 3        part
```
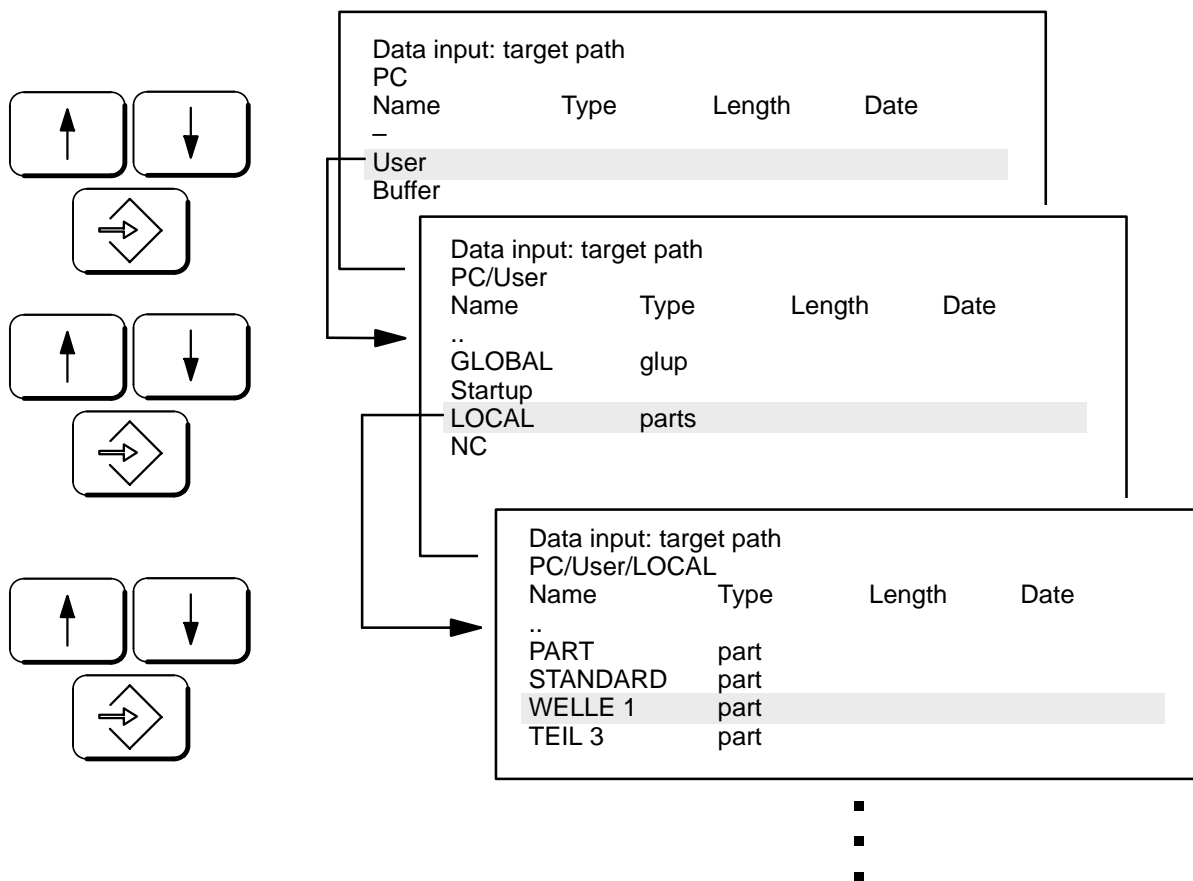
| Into new workpiece |

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 L$_F$.

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:

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.
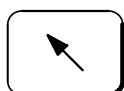
Press the HOME key to switch to the lower window CLIPBOARD/INSERT.

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.
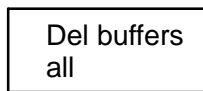
Del buffers all	Using this softkey you delete all files from the buffer. If you press this key, the following dialog text is displayed:

Caution
Function deletes all objects in clipboard
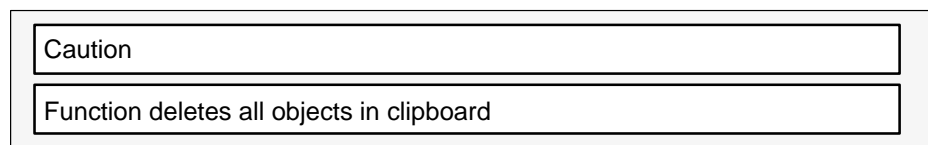
⌃	Press the RECALL key to leave the menu if you do not wish to delete the files.
OK	The function is executed if you press the OK softkey.
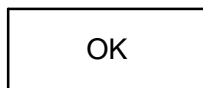
6.3.2 Continuous mode for data input via RS232C (V24) (from SW 5.6)

Continuous mode for reading in of part programs via an RS232C (V24) inter-face 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 inter-face of the 3 possible RS232C (V24) interfaces of the MMC. The correspond-ing entries must be made in the file BEDCONF (line 191) in the directory Op-eration/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


    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-      Archive              Floppy              Output      Error
 pieces     list                                     Start       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.

> PC format data can only be created and edited externally with PCIN 3.1 or higher.
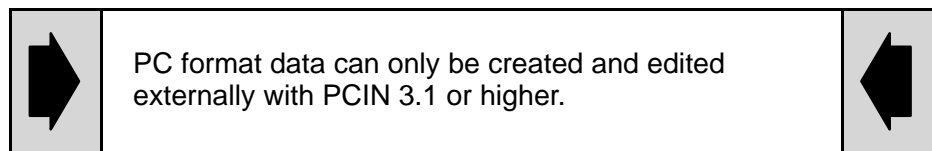
Data in PC format is written on the external device as byte stream complying with the standardized "tar Format". The PC format standard is described in:

- 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
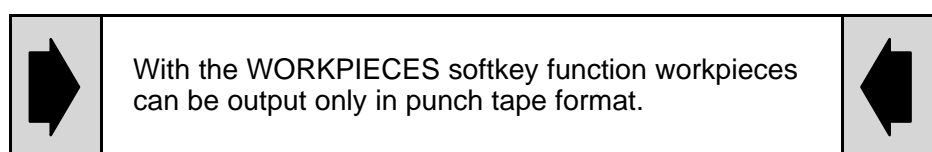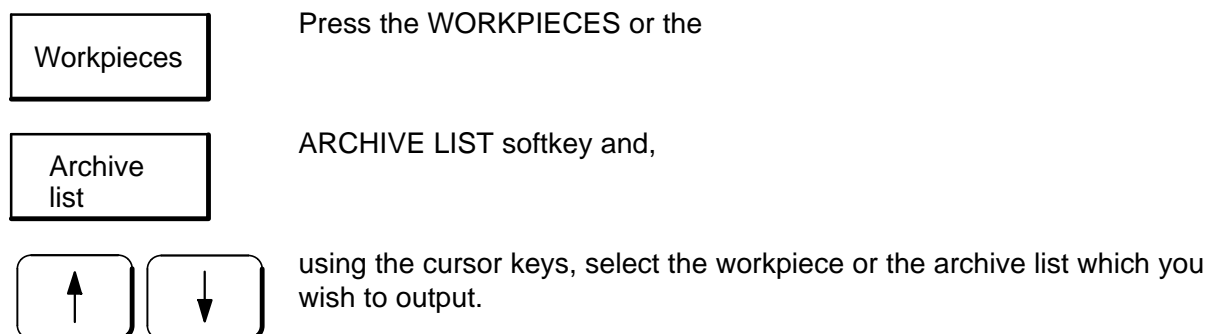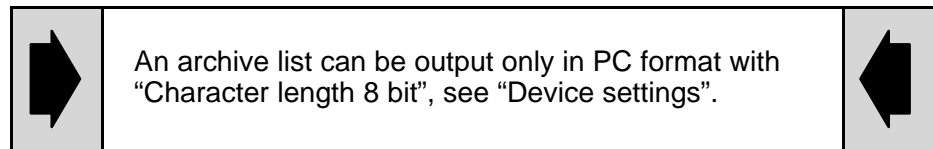
| Workpieces |

Press the WORKPIECES or the

| Archive list |

ARCHIVE LIST softkey and,

| ↑ | | ↓ |

using the cursor keys, select the workpiece or the archive list which you wish to output.

> With the WORKPIECES softkey function workpieces can be output only in punch tape format.

An archive list can be output only in PC format with "Character length 8 bit", see "Device settings".

Start

Press the START softkey to start transmission.

The following dialog screenform is displayed:

| !!! Transmission PC/external active !!! | Punch tape format | MPF 1 | 1024 |
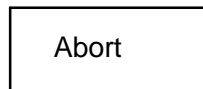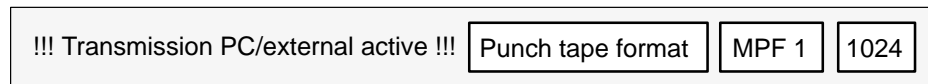
Abort

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:

180010 Workpiece archiving in punch tape format only
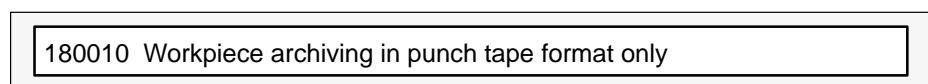
Acknowledge the dialog box by pressing the OK softkey.

If data other than loaded by means of the job lit is stored under a workpiece, it is not backed up at data output through the WORKPIECES softkey function.

If you start data transfer for a workpiece by means of the DATA softkey function, the entire workpiece with all data, irrespective of whether a job list is available or not, is transferred.
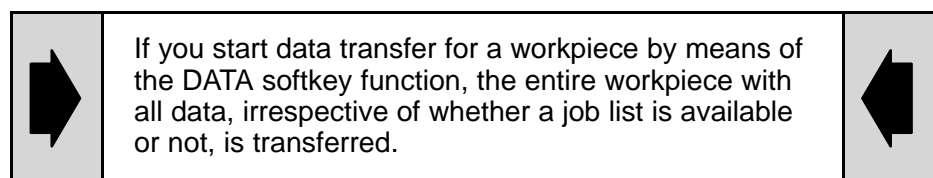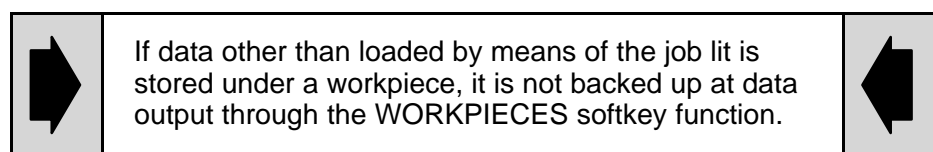
6.4.3   Data output

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:

```
User
Name            Type        Length       Date
–
Services
Data management
GLOBAL          glup
Startup
Listenbaustein
LOCAL           parts
NC
PLC
Regie
Ser
Driv     User/LOCAL
Sim      Name            Type        Length       Date
List     ..
         Part            part
         Standard        part
         Welle1          part
         Zahnrad5        part
                 User/LOCAL/Welle1
                 Name            Type        Length       Date
                 ..
                 JOB             jobl        1866        05–25–93
                 MPF1            parp        596         06–20–93
                 RPA1            rpa         470         06–20–93
                 TOA             toa         13700       06–20–93
```
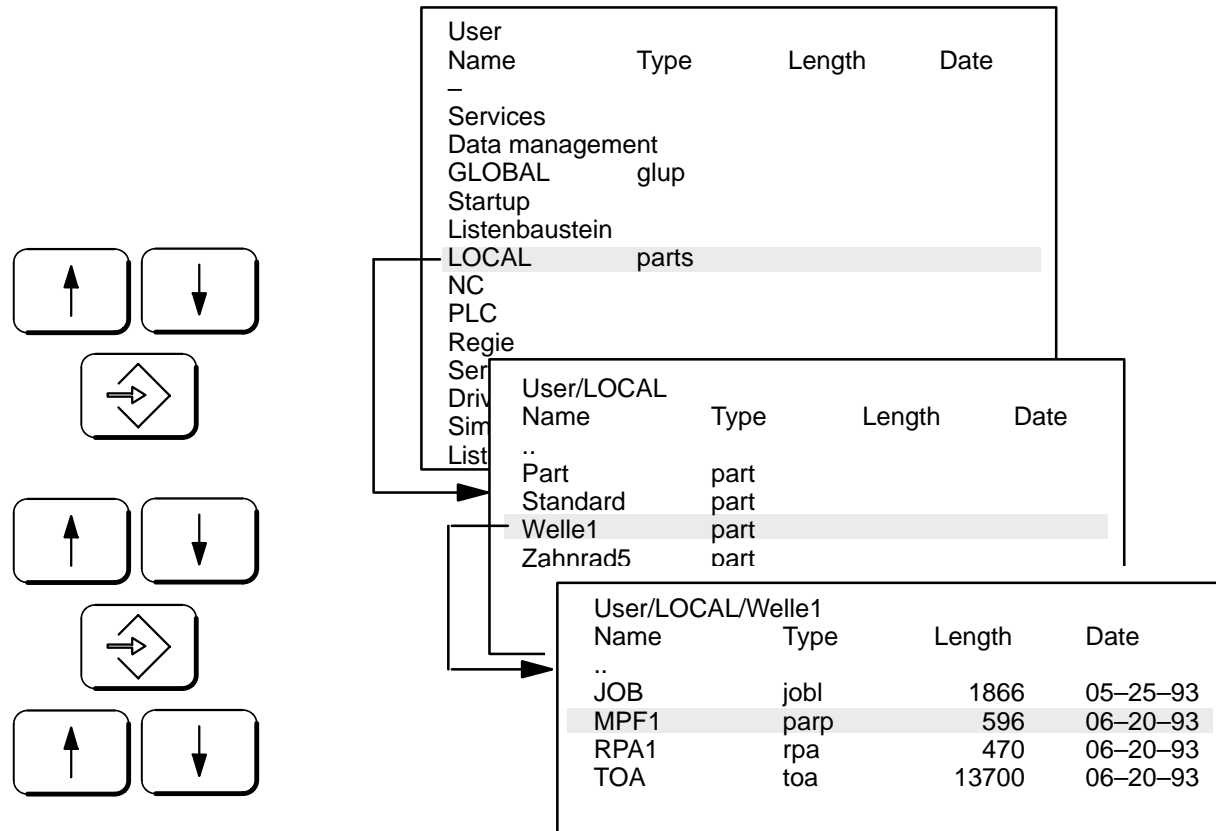
When you have reached the source directory, use the cursor keys to select the file you wish to output (you can recognize files by the entry in the LENGTH column).

Output Start

Press the OUTPUT START softkey to start transmission. The following dialog screenform is displayed:

| !!! Transmission PC/external active !!! | PC format | MPF 1 | 1024 |

Abort

Press the ABORT softkey to interrupt data transfer.

If an error occurs during data transfer, the following dialog text is displayed:

| 110019 Observe error log |

Press the OK softkey to acknowledge the dialog text.

Error log

If you press the ERROR LOG softkey, all files in which errors have occurred during data transfer are displayed.

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 (**L**ong **D**istance
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).

6–27

6.5.1   Data output with the FD–E2 floppy drive
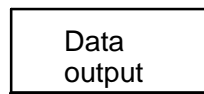
| Data output |

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.
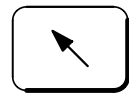
Use the HOME key to switch between the windows. The active window is highlighted by a yellow frame. The current interface parameter appears in the top window, e.g.:

> 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.
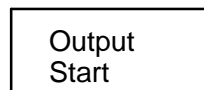
| Output Start |

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

| Floppy |

Press the FLOPPY softkey to display a screen with the boxes

- Floppy directory

- Archive name, length, date, time.
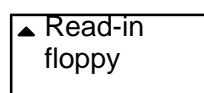
| ▲ Read-in floppy |

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.

```
┌─────────────┬──────────────┬───────────┬─────────┬──────────────┬────────────┬─────────┐
│ 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                              │
│   PLCPROG .001          22528       13.01.1994         16:02                              │
│                                                                                          │
│                                                                                          │
├──────────────┬──────────┬──────────┬──────────┬──────────┬──────────┬──────────┬─────────┤
│ Read-in      │          │          │          │          │          │ Archive  │         │
│ floppy       │          │          │          │          │          │ 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.

Delete archive

The following message appears:

Delete archive?	PART . 001

Press RECALL to cancel the process.

OK
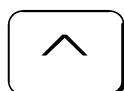
If you press the OK softkey, the archive and all files assigned to it are deleted from the hard disk.

Archive Select

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.
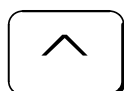
Press RECALL to switch back to the basic SERVICES display.

6.5.2   Data input from the FD–E2 floppy drive
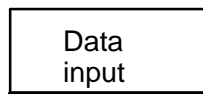
|	
Data input	Press the DATA INPUT softkey in the basic display of the SERVICES area. The screen is divided into two windows for the data input function.
↖	Use the HOME key to switch to the top window. In the "Archive name" input field enter the name of the archive file that you want to read in from the diskette.
Archive contents	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:

Fig. 6.12   Data input from diskette

The data of the selected archive are displayed.

Use the arrow keys to select a file.
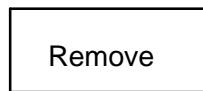
Remove

Press the REMOVE softkey to delete the selected file from the buffer, i.e. the contents of the diskette remain the same.

Display again

Press this softkey to display the contents of the diskette again.

OK
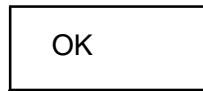
Press the OK softkey to terminate the data transfer, i.e. the files are transferred to the hard disk.
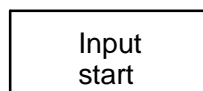
Input start

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
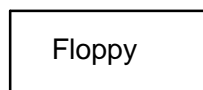
Floppy

Press the FLOPPY softkey. A display with the following fields appears:

- Floppy directory
- Archive name, length, date, time.
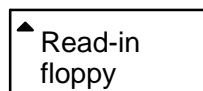
Read-in floppy

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.
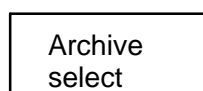
Archive select

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
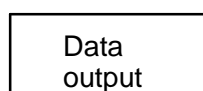
Data output

Select the DATA OUTPUT softkey in the basic SERVICES display.

Floppy

Press the FLOPPY softkey.

Format diskette	Press the softkey and the following message appears:

Please insert floppy

OK	Press the OK softkey to start formatting the diskette in the external floppy drive FD–E2 and the following message appears:

Now formatting

> You cannot abort the formatting process. Press RECALL to exit the menu and perform other operator actions.

Press RECALL to return to the basic SERVICES display.

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-pro-tected).

Operation   The operations required for data transmission with the FD–E1 are identi-cal 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
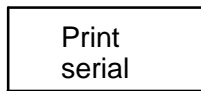
Print serial

Press the PRINT SERIAL softkey. The user and the Siemens branch are displayed.

Use the cursor keys and the
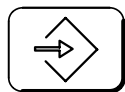
INPUT key to select the user branch. The following directory is displayed:

```
Machine     Parameter  Programm.              Services  Diagnosis

                                                              16:10

Print serial
  PC/User                                                 Serial 1

  Name                 Type      Length   Date

  ..
  WOP
  NC
  Services
  Master control
  Operation
  LOCAL                parts
  GLOBAL               glup
  Start-up




                                                          Start
              Job
              list
```
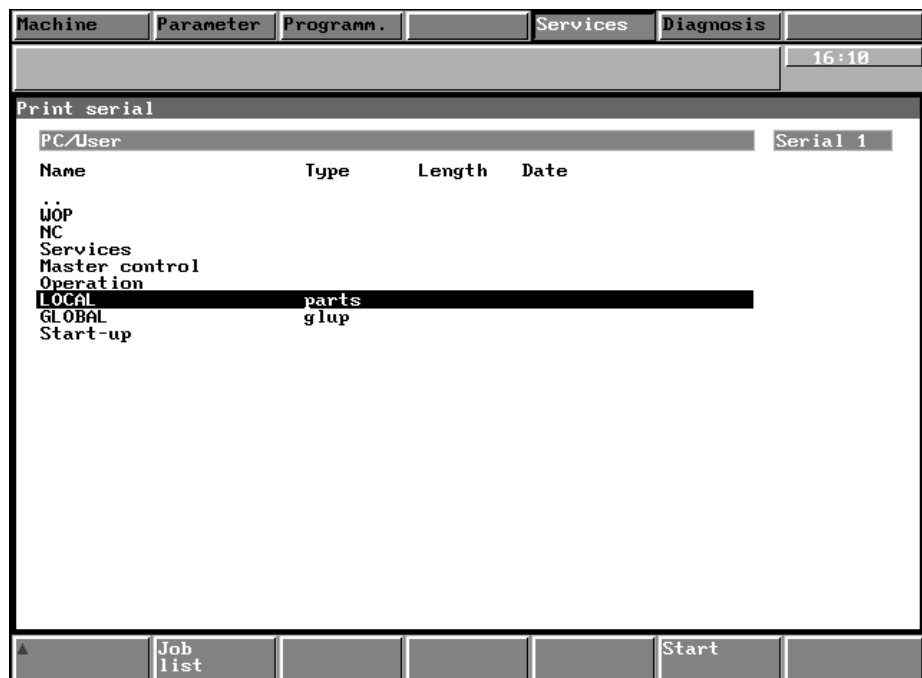
Fig. 6.13   User branch directory in the SERVICES/PRINT area

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 but not directories.
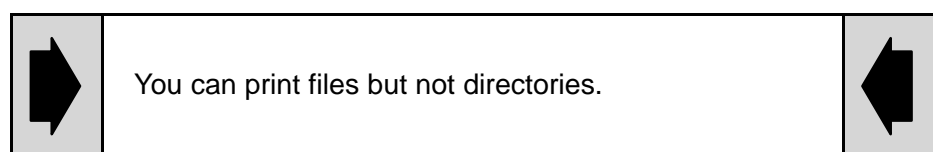
You can print files from various directories one after the other. Page with the cursor keys and the INPUT key through the corresponding directories.

Start	Press the START softkey to transfer the files to be printed in the job list; printing of the first print job is immediately started.
Job list	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.
Erase	By pressing the ERASE softkey you can delete data in the job list which you have previously selected.
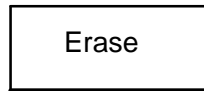
6.8   Archive lists

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.

> Only save complete workpieces in tape format and avoid the archive list. Otherwise programs of the same name will be overwritten when they are read into the clipboard.
> Only use PC format archive lists.

Precondition

The SERVICES area has been selected.

> Please note that if you wish to create, edit or delete archive lists you must assign access rights by means of the keyswitch position 2.
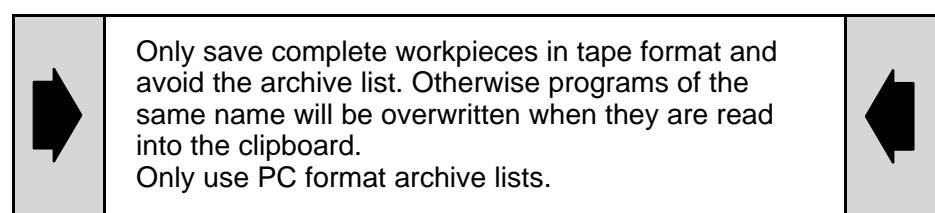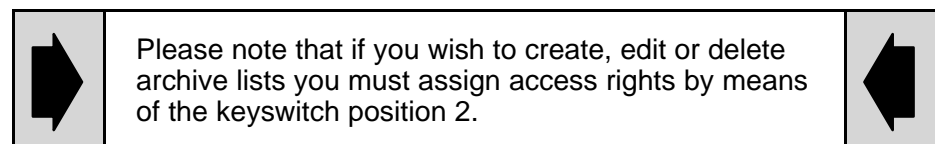
6.8.1   Creating new archive lists

Operating sequence

| Data output | Press the DATA OUTPUT softkey. |

| Archive list | When you press the ARCHIVE LIST softkey, the following display appears: |

```
 Machine     │ Parameter │ Programm. │        │  Services │ Diagnosis │
                                                                      │ 15:45
 Data output: Archive list
    Interface            Serial 1      Archive name     ARCHFILE
    Device               STANDARD      Format           PC format

    Archive lists
    Name                 Type       Length  Date
    -
    ARCH_1               arc          356   11-08-1994  15:44:02
    ARCH_2               arc          398   11-08-1994  15:44:56




 │New       │Edit      │Delete    │Copy      │          │          │          │
```

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.
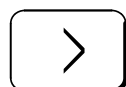
| > | Press the ETC key to extend the displayed softkey bar with the following options: |

▲ New	Edit	Delete	Copy			▶

| ▲ New | If you press the NEW softkey, the following menu is displayed: |

| Please enter name | CW1000 |

You can enter a name for the new archive list on the alphanumeric keyboard and
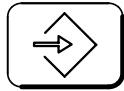
confirm the entry with the INPUT key.

OK
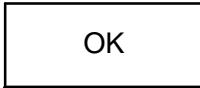
Press the OK softkey to acknowledge and the following display appears:

| Machine | Parameter | Programm. | | Services | Diagnosis | |

11:59

Data output/Edit archive list

Archivelist   Type   CW1000

/PC/User/Operation/Basic settings   bdgl
/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   jobl

User/Operation

Name   Type   Length Date

..
ANWENDE   mode
Basic_settings
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:

SINUMERIK 840C (BA)

```
User
Name            Type        Length      Date
Operation
Data management
GLOBAL          glup
Startup
```

```
User
Name            Type        Length      Date
Data management
Regie
Operation
LOCAL           parts
GLOBAL          glup
```

```
User/LOCAL
Name            Type        Length      Date
PART            part
STANDARD        part
PART1           part
SHAFT1          part
```
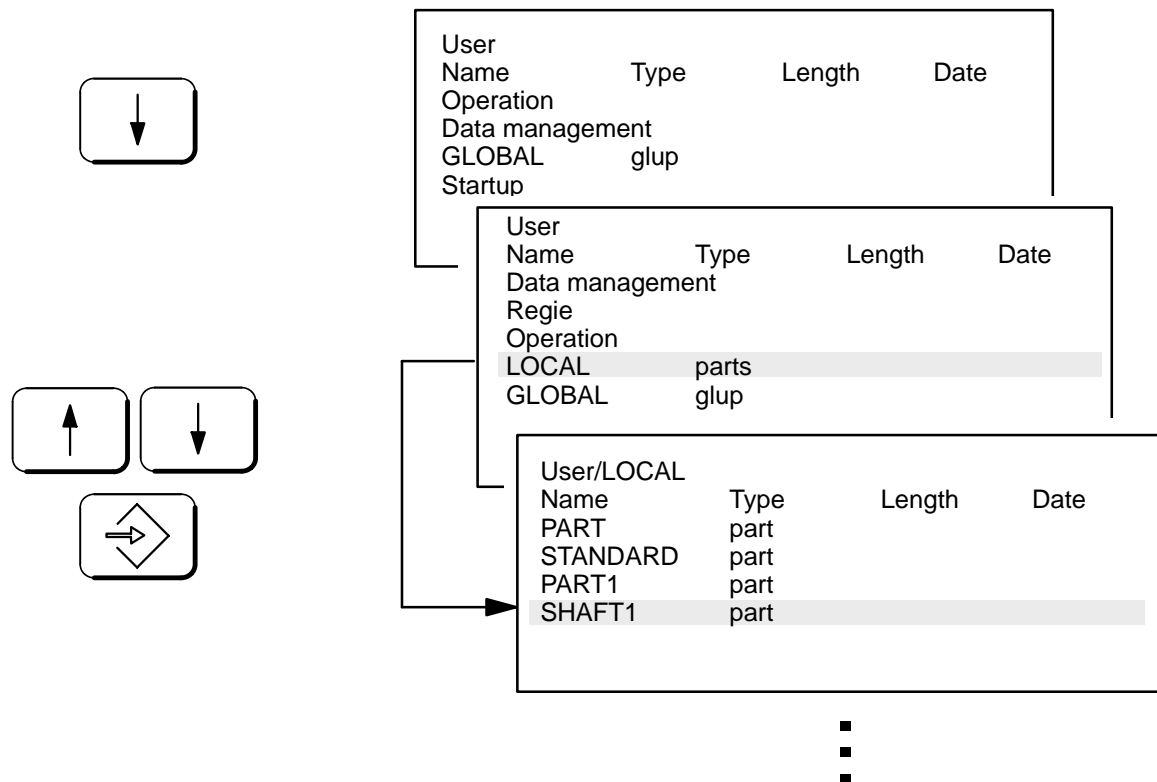
Include

Press the INCLUDE softkey to include the previously selected element (directory, workpiece, files) in the archive list.

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.

Save

Press the SAVE softkey to save the archive list. The operating system automatically inserts the correct syntax for the archive list.

Remove

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.

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

Archive list	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.

Edit	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
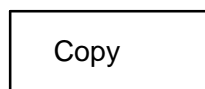
Press the COPY softkey. The following dialog text is displayed:

Please enter name	CW500
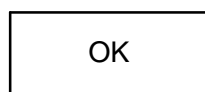
Enter the new name with the alphanumeric keyboard, acknowledge with the INPUT key and press the OK softkey. The copied archive list is stored in the directory.

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
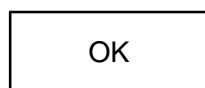
Press the OK softkey to delete the archive list.

> While data input/output is active copying is not possible.

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:

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
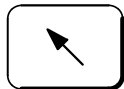
Use the HOME key to switch from the Siemens window to the user
window and vice versa. The window which is active is marked.

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.
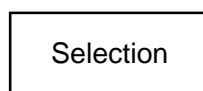
Selection   Press the SELECTION softkey and the following dialog text is displayed:

Interface being parameterized!

After parameterization has been successfully completed the text
disappears. The interface is selected and is displayed in the basic
display.

> When you select the FLOPPY device setting, the
> diskette device must be connected and switched on
> otherwise a message with the following text is
> output: "Interface <RS232C(V24)> not
> parameterized".
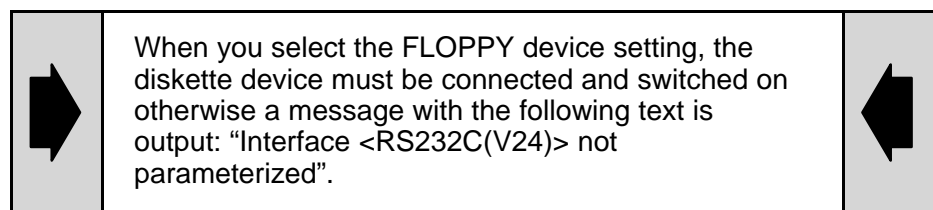
Note   The interface remains selected after switching the control on or off.

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
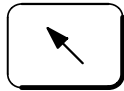
Press the HOME key to switch to the Siemens window.

Select an interface (e.g. STANDARD)
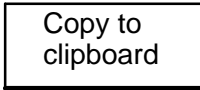
| Copy to clipboard |

and press the COPY TO CLIPBOARD softkey
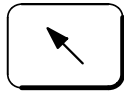
Press the HOME key to switch to the user window

| Paste from clipboard |
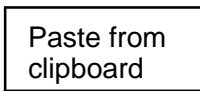
and press the PASTE FROM CLIPBOARD softkey; the following screenform is displayed:

| Please enter name | | TREADER |

Use the alphanumeric keyboard to enter a name and acknowledge with the INPUT key.

| OK |
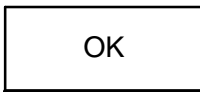
Press the OK softkey to accept the name in the user interface directory.
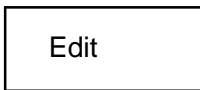
| Edit |

Use the cursor keys to select an interface in the user branch and press the EDIT softkey.

```
 Machine    Parameter  Programm.            Services  Diagnosis
                                                        12:12

Devices/Edit
                                                        PG740

    Comments      Dies ist die standardmaessige 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.

    XON char. (hex)     11                               60

    XOFF char. (hex)    13          Transmission format

    End of transmission                      PC format
       character (hex)  03

    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.
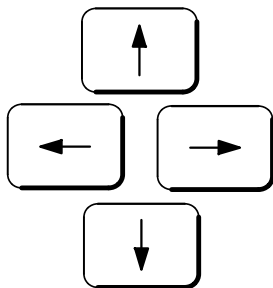
Use the cursor keys to move from one field to the next.

Store

When you have completed editing, press the STORE softkey to store the setting on the hard disk.

Delete

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.

OK
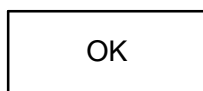
Use the OK softkey to delete the device settings from the directory.

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 modi-
fied in the SERVICES, DATA INPUT and DATA OUTPUT area. The op-
erator interface for interfaces and device settings has been changed.

The selection is made in the SERVICES area.

<div style="border:1px solid; padding:4px; width:180px; text-align:center">Select</div>

```
┌──────────┬───────────┬──────────┬─────────┬─────────────┬───────────┬──────┐
│ Machine  │ Parameter │ Programm.│         │ Services    │ Diagnosis │      │
├──────────┴───────────┴──────────┴─────────┴─────────────┴───────────┴──────┤
│                                                              │ 14:18  │    │
├────────────────────────────────────────────────────────────┴────────┴────┤
│ Service                                                                    │
│                                                                            │
│      NC                    Transfer data from/to NC                        │
│                            Display information from the shopfloor sheet     │
│      Computer              Transfer data from/to host computer             │
│      link                  transfer                                        │
│      Devices               Device settings                                 │
│      Print                 Print PC data                                   │
│      serial                                                                │
│      Data                  Read-in PC data                                 │
│      input                                                                 │
│      Data                  Output PC data                                  │
│      output                Process archive lists                           │
│      Data                  Copy and delete PC data                         │
│      management                                                            │
│                                                                            │
│      Interface      Device                    Status                       │
│                                                                            │
│      Floppy         FLOPPY                     inactiv                      │
│      Serial 1       STANDARD                   inactiv                      │
│      Serial 2       STANDARD                   inactiv                      │
│      Serial 3                                                              │
│                                                                            │
├──────────┬──────────┬──────────┬─────────┬─────────┬─────────┬────────────┤
│    NC    │ Computer │ Modify   │ Print   │ Data    │ Data    │ Data       │
│          │ link     │ devices  │ serial  │ input   │ output  │ management │
└──────────┴──────────┴──────────┴─────────┴─────────┴─────────┴────────────┘
```

Fig. 6.18

The screenshot above shows the assignment between the interfaces
(Floppy, Serial 1, Serial 2 and Serial 3) and the device (Floppy, Stan-
dard, WS800A) and, for example, a user–generated file New. The cur-
rent setting is displayed on a grey background.

<div style="border:1px solid; padding:4px; width:180px; text-align:center">Change
devices</div>

Softkey CHANGE DEVICES opens a screen in which the assignment
between the interface and the device can be changed.

The interface selected in 6.18 can be assigned to a device in screen De-
vices: User. The device is selected by means of the cursor keys.

```
┌──────────┬───────────┬──────────┬──────┬──────────┬──────────┬─────────┐
│ Machine  │ Parameter │ Programm.│      │ Services │ Diagnosis│         │
├──────────┴───────────┴──────────┴──────┴──────────┴──────────┼─────────┤
│                                                              │  14:18  │
├──────────────────────────────────────────────────────────────┴─────────┤
│ Devices: User                                                           │
│                                                                         │
│   Interface:              Floppy                                        │
│                                                                         │
│   Device:                 FLOPPY      '-' not initialized!              │
│                                                                         │
│   Name                       Length  Date                              │
│                                                                         │
│   -                                                                     │
│   FLOPPY                        158   02-11-1999   08:08:14            │
│   STANDARD                      158   02-11-1999   08:08:14            │
│   WS800A                        158   02-11-1999   08:08:14            │
│   XON_XOFF                      158   02-11-1999   08:08:16            │
│                                                                         │
│                                                                         │
├─────────┬─────────┬─────────┬─────────┬──────────┬─────────┬──────────┤
│         │         │         │         │  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 Device settings.

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/Output”. 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.

> When editing please bear in mind the access rights
> determined by the positions of the keyswitch.
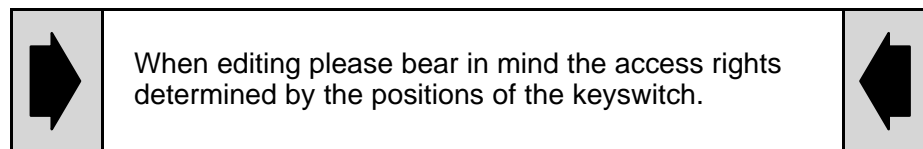
Operating sequence   The "Data management" display is divided into two windows:

- Siemens and

- User
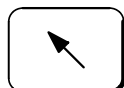
You can use the HOME key to switch between the two windows. The
active window is highlighted by a yellow frame.

Data
management

Press the DATA MANAGEMENT softkey; the following is displayed:

```
 Machine    Parameter  Programm.            Services  Diagnosis
                                                                      12:13
Data management: Standard
  Siemens
  Name                    Type      Length  Date
  -
  List_module
  NC
  WOP


Data management: User
  User
  Name                    Type      Length  Date
  -                                               ▲
  Operation
  Data_management
  GLOBAL                  glup
  Start-up
  List_module
  LOCAL                   parts
  NC
  PLC
  Master_control
  Services                                        ▼
 ▲              Delete   Copy to  Paste from            View
                         clipboard clipboard            only
```
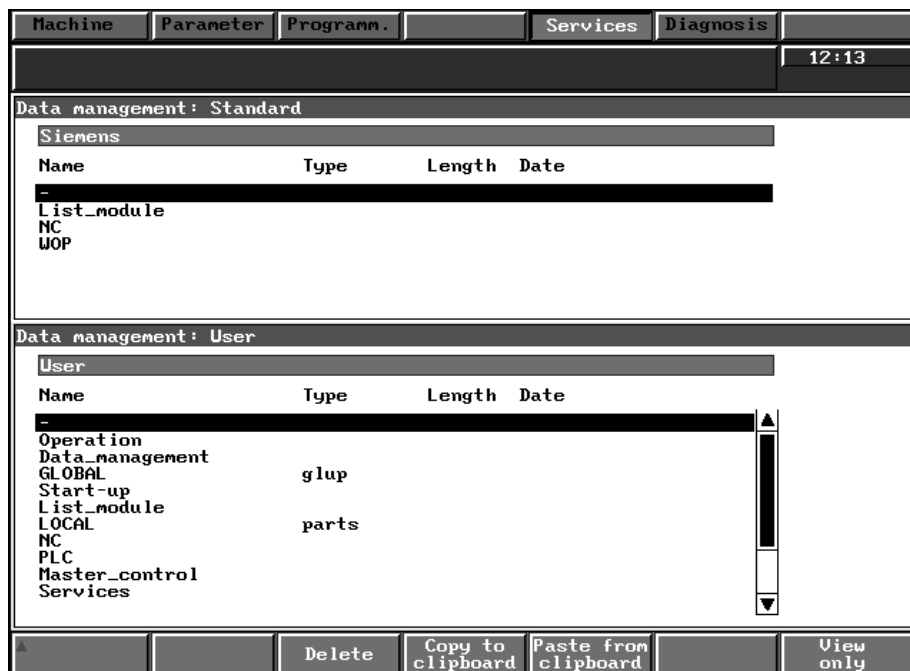
Fig. 6.21   SERVICES/MANAGEMENT area

6.10.1   Copy and paste in the SERVICES/MANAGEMENT area
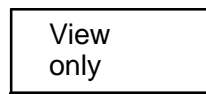
View
only

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).
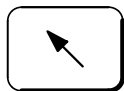
Use the HOME key to switch form the Siemens window to the user window and vice versa. The active window is marked (yellow frame).

Use the cursor keys
and
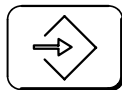
the INPUT key to page to the desired source directory: select the workpiece or the data with the cursor keys.
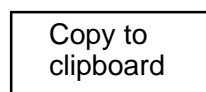
Copy to
clipboard

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.

Paste from
clipboard
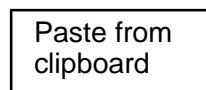
Press the PASTE FROM CLIPBOARD softkey; you are requested to enter the file name:

Please enter name		STANDARD

OK
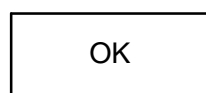
Press the OK softkey if the name of the source file and the copied file are to be identical.

. . .
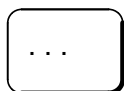
If the copied file is to be named differently from the source file, enter the new name via the alphanumeric keyboard and conclude with the INPUT key.

Press RECALL to cancel the process

OK	Press the OK softkey to copy the file into the target directory.

▶ While data input/output is active copying is not possible. ◀
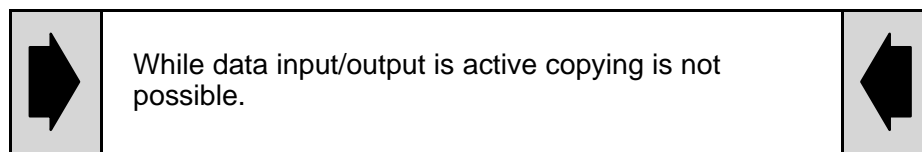
6.10.2   Delete data in the SERVICES/MANAGEMENT area

Operating sequence

↑ ↓	Use the cursor keys to select a data directory in the user branch and
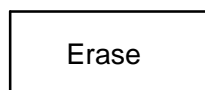
Erase	press the ERASE softkey. The following dialog text appears:

| Do you want to delete the data? | SEA4 |
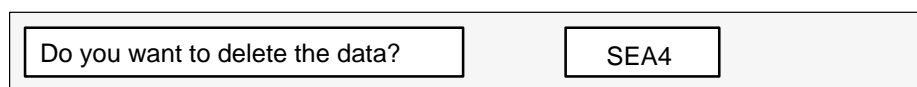
∧	Press RECALL to cancel the process.
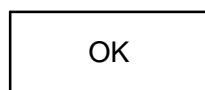
OK	If you press the OK softkey, the selected data is irretrievably deleted from the hard disk.

▶ If a workpiece is deleted or copied, all workpiece data is deleted or copied. ◀

▶ Caution when deleting data with set identifier. Important system data might be deleted. ◀

END OF SECTION

SINUMERIK 840C (BA)

7 Diagnosis

| Machine | Parameter | Programm | | Services | Diagnosis | |

Alarms and message displays, service displays

7.1 **Alarm and message display**	**7.2** **Service displays**	**Startup**
	7.2.1 **Alarm logs**	**Machine data dialog**
	7.2.2 **NC information**	**General reset**
	7.2.3 **Drift compensation**	**PC data management**
		Drive/servo startup
		Backup
		PG functions

Fig. 7.1   Diagnosis

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 | |

| Messages | Press this softkey to display a list of the current 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.
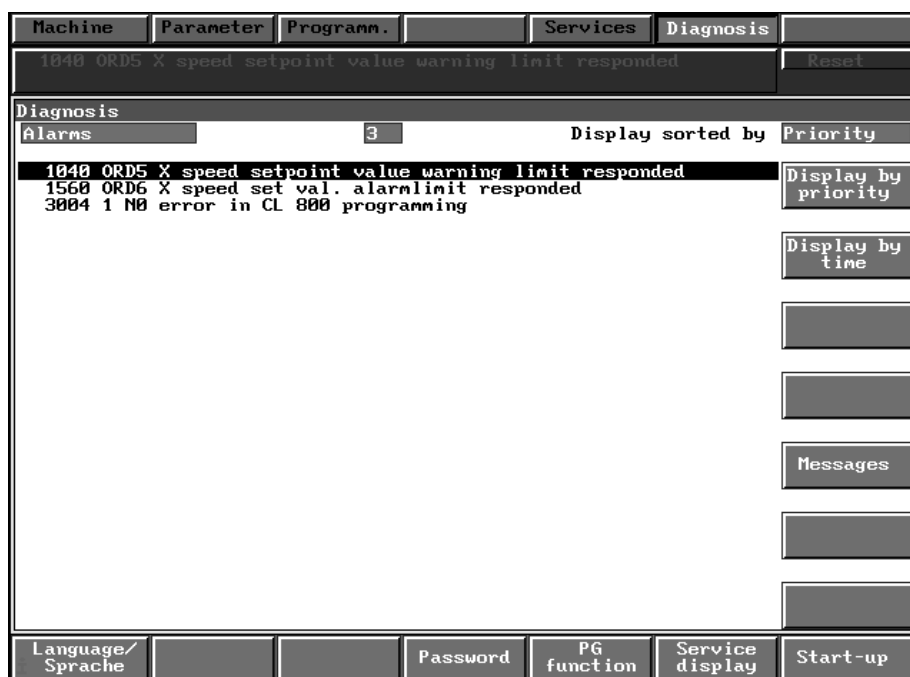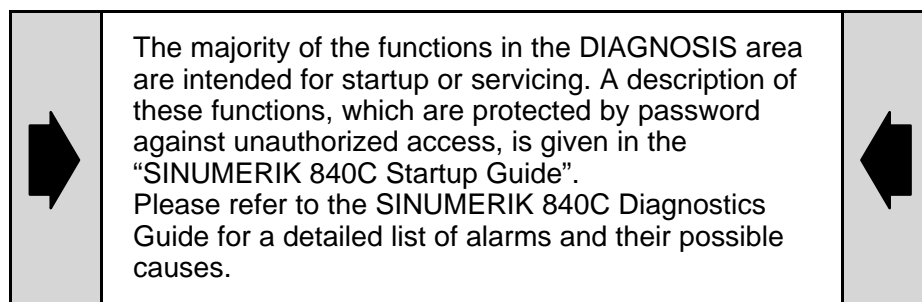
7.2   Service displays

7.2.1   Alarm logs

| Service display |

Press the SERVICE DISPLAY softkey in the basic display of the DIAGNOSIS area.

| Alarm log 1 |

| Alarm log 2 |

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.

Fig. 7.3   Alarm log 1

7.2.2   NC information

NC info

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).
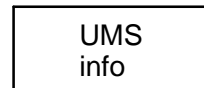
UMS info

Press the UMS INFO softkey for information on the contents of the UMS (languages, menus, cycles etc.).
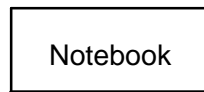
Notebook

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
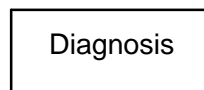
Diagnosis

DIAGNOSIS

Service display

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

Further axis/spind	Axis single	Spindle single	Drift comp axis 1	Drift comp axis 2	Drift comp axis 3	Drift comp axis 4

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).

Preconditions

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 CSB module must **not** be removed from the subrack even if a battery is in place, because the data to be battery-backed is not physically on the CSB. Removing the CSB would automatically cause all data mentioned above to be lost.
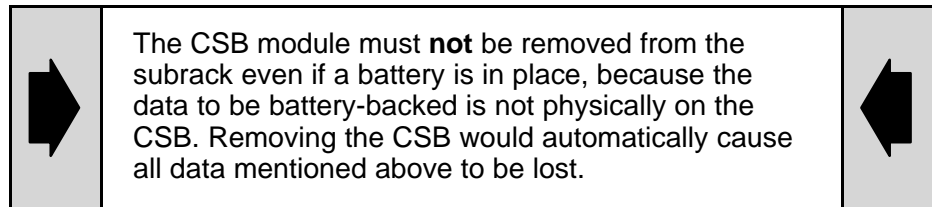
The battery backup voltage is monitored by the control.

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.

> The battery may not be replaced unless the control is switched on.
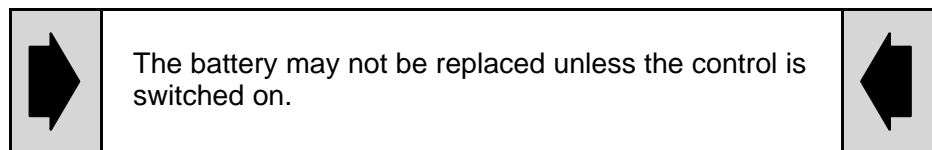
Battery replacement

- 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 **E**NDANGERED
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

Electro**m**agnetic **c**ompatibility (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. **Note**: 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 wrong-ly	• 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.

Faulty behaviour	Possible cause	Elimination of fault
Rough irregular machining surface	• Poor equipotential bonding between position encoder and control	• Increase cross section of bonding conductors (between control and position encoder), connect bonding conductor directly to the control (not to separate voltage bus)
	• High equalizing current to setpoint cable screening when screen is connected to housings on both sides	• Interrupt screen connection on the control side
	• Bonding conductor between SINUMERIK and drive control laid too far from setpoint cable	• Shorten distance between bonding conductor and setpoint cable
	• Setpoint input on the drive control does not match SINUMERIK EMC concept	• After input protective circuit on the drive control (in accordance with protective system of Siemens drive controls)

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.

> Solvents such as chlorinated hydrocarbons, benzene, esters and ethers should be avoided.
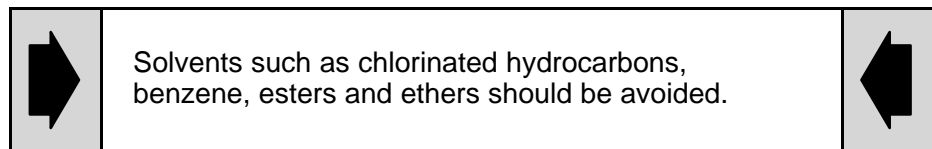
END OF SECTION

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 memeory 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

Suggestions	
Corrections	
For Publication/Manual SINUMERIK 840C Software Version 6 User Documentation	

| **From** | Operator's Guide

Order No.: 6FC5198–6AA00–0BP2
Edition: 09.2001 |
| Name _____
Company/Dept.
Address _____
_____
Telephone: _____ / _____
Telefax: _____ / _____ | Should you come across any printing errors when reading this publication, please notify us on this sheet. Suggestions for improvement are also welcome. |

Suggestions and/or corrections

Overview of SINUMERIK 840C Documentation / OEM Version for Windows

General Documentation

SINUMERIK 840C
Brochure

SINUMERIK 840C
Catalog NC 36

SINUMERIK Accessories
Catalog NC Z

User/Manufacturer/Service Documentation

SINUMERIK ACR 20/805SM/840C
Link to SINEC L2–DP with Module
– IM328–N, Slave
– IM329–N, Master and Slave

User Documentation

SINUMERIK 840C
Diagnostics Guide

User Documentation

SINUMERIK 840C
Operator's Guide
– OEM Version
   Windows
– Standard

SINUMERIK 840C
Programming Guide

SINUMERIK 840/840C/880/880 GA2
User's Guide
Graphic Programming System
– Drilling/Boring and Milling
   Parts 1 + 2
– Turning Parts 1 + 2
– On DOS PC
– Environment Description 840C

SINUMERIK 840/840C/850/880/880 GA2
Cycles,
Programming
Guide

SINUMERIK 840C
Measuring Cycles
Version 20
User's Guide

SINUMERIK 840C
User's Guide
Simulation Milling
and Turning

Manufacturer Documentation

SINUMERIK 840C
Interface:
– Signals
– Connection
   Conditions

SINUMERIK 840/840C/880/880 GA2
Function Block
Packages
Function Macros

SINUMERIK 840/840C/880/880 GA2 / **840/880**
PLC 135 WB/WB2d/WD
Quick Reference,
Planning
S5–HLL

SINUMERIK 840C
SINUMERIK
WS 800A
– CL 800
   Cycle Language
– User's Guide

SINUMERIK 840C
Planning Guide
Graphic
Programming System

SINUMERIK 840C
User's Guide
OEM Version
for Windows

Manufacturer Documentation

SINUMERIK 840C
Alarm Dialog
for PC
OEM Version
for Windows

SINUMERIK 840/840C/880/880 GA2
Computer Link
– SINT
– SIN PS 231
– SIN PS 315

SINUMERIK 840/840C/880/880 GA2
Computer Link
– Message Frame
   Description
– General
   Description

SINUMERIK SIMODRIVE Safety Integrated
Description of
Functions
SINUMERIK
Safety Integra-
ted

Service Documentation

SINUMERIK SIMODRIVE 840C 611D
Installation Guide
– Instructions
– Lists
– Difference
   Description Windows

SINUMERIK 840/840C/850/880/880 GA2
Measuring Cycles
Version 20
Start–up Guide

SINUMERIK 840C
Spare Parts
List

Electronic Documentation

SINUMERIK SIMODRIVE 840C 611D
DOC ON CD