SINUMERIK 840C
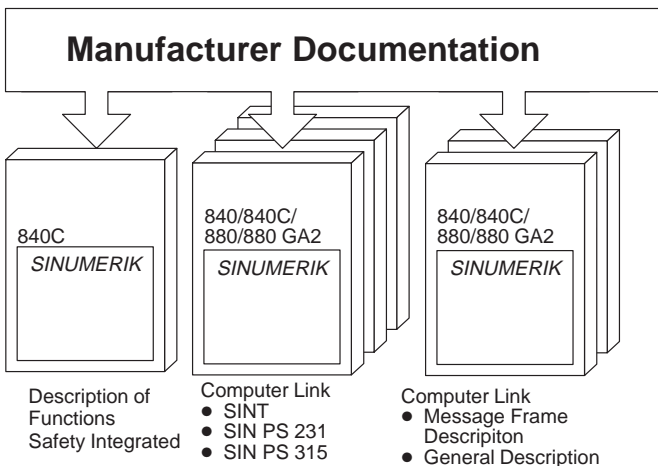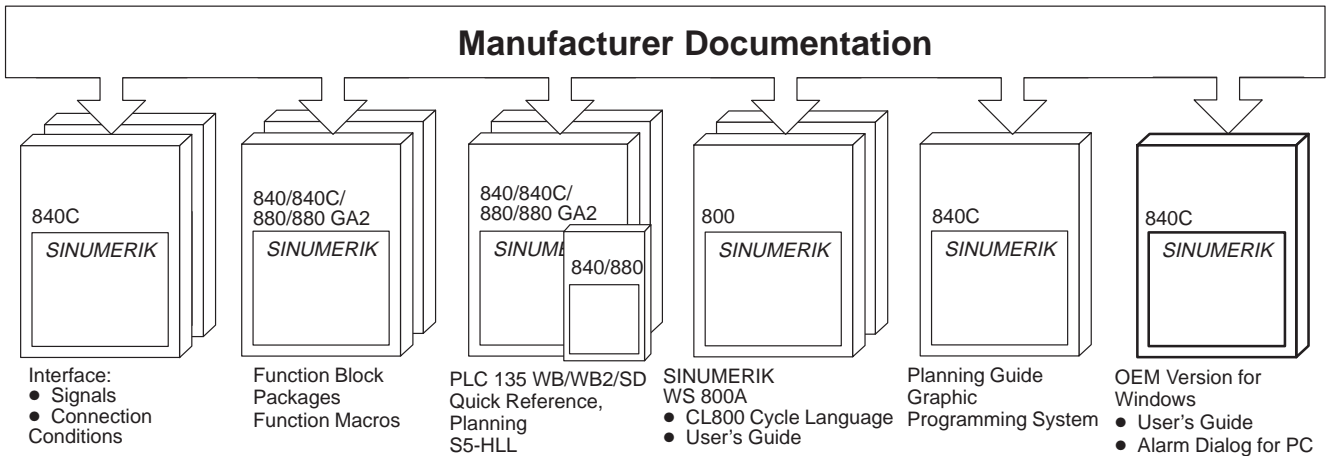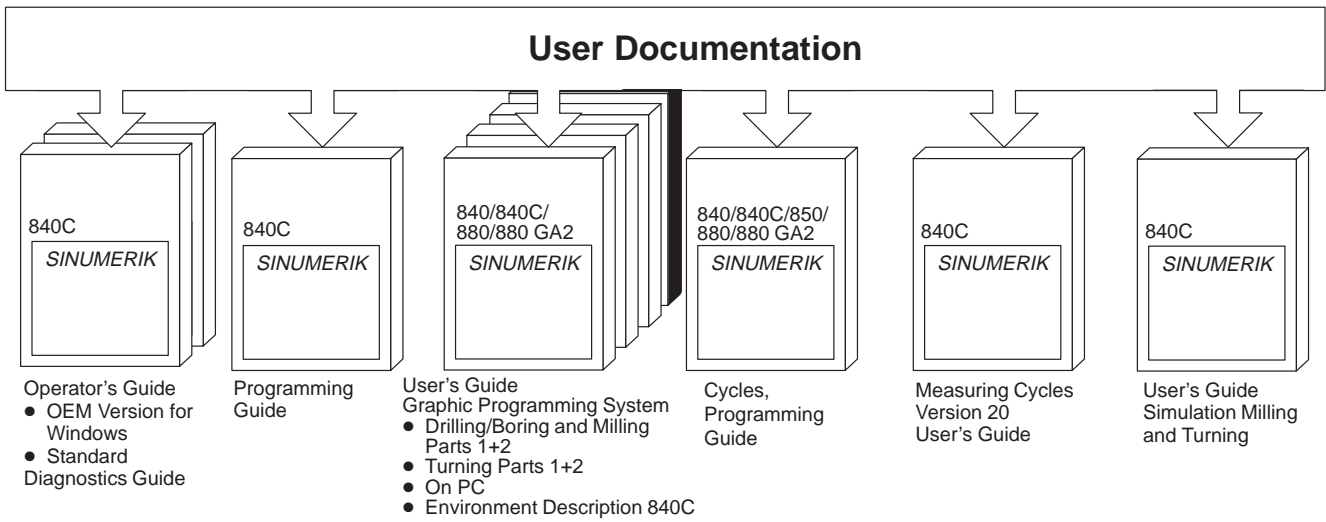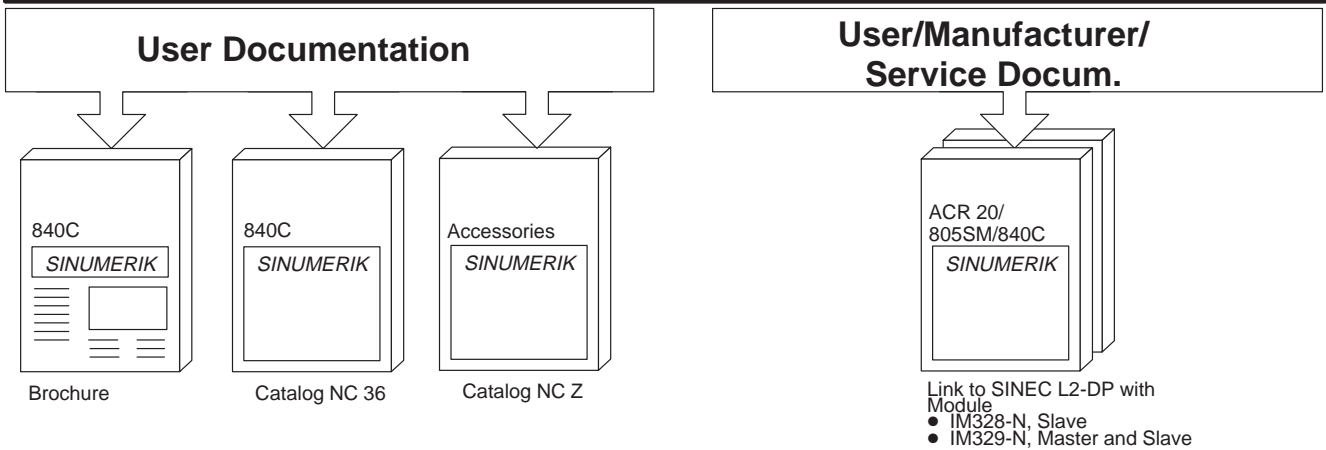# Software Version 6
OEM Version for Windows™

Operator's Guide   07.97 Edition

User Documentation

SINUMERIK 840C / OEM Version for Windows

User Documentation

840C SINUMERIK	840C SINUMERIK	Accessories SINUMERIK
Brochure	Catalog NC 36	Catalog NC Z

User/Manufacturer/Service Docum.

ACR 20/ 805SM/840C SINUMERIK

Link to SINEC L2-DP with Module
- IM328-N, Slave
- IM329-N, Master and Slave

User Documentation

840C SINUMERIK	840C SINUMERIK	840/840C/ 880/880 GA2 SINUMERIK	840/840C/850/ 880/880 GA2 SINUMERIK	840C SINUMERIK	840C SINUMERIK

Operator's Guide
- OEM Version for Windows
- Standard Diagnostics Guide

Programming Guide

User's Guide Graphic Programming System
- Drilling/Boring and Milling Parts 1+2
- Turning Parts 1+2
- On PC
- Environment Description 840C

Cycles, Programming Guide

Measuring Cycles Version 20 User's Guide

User's Guide Simulation Milling and Turning

Manufacturer Documentation

840C SINUMERIK	840/840C/ 880/880 GA2 SINUMERIK	840/840C/ 880/880 GA2 SINUMERIK 840/880	800 SINUMERIK	840C SINUMERIK	840C SINUMERIK

Interface:
- Signals
- Connection Conditions

Function Block Packages Function Macros

PLC 135 WB/WB2/SD Quick Reference, Planning S5-HLL

SINUMERIK WS 800A
- CL800 Cycle Language
- User's Guide

Planning Guide Graphic Programming System

OEM Version for Windows
- User's Guide
- Alarm Dialog for PC

Manufacturer Documentation

840C SINUMERIK	840/840C/ 880/880 GA2 SINUMERIK	840/840C/ 880/880 GA2 SINUMERIK

Description of Functions Safety Integrated

Computer Link
- SINT
- SIN PS 231
- SIN PS 315

Computer Link
- Message Frame Descripiton
- General Description

Service Documentation

840/840C/ 880/880 GA2 SINUMERIK	840/840C/ 880/880 GA2 SINUMERIK	840C SINUMERIK

Installation Guide
- Instructions
- Lists
- Difference Description Windows

Measuring Cycles Version 20 Start-up Guide

Spare Parts List

SIEMENS

SINUMERIK 840C

Software Version 6

Operator's Guide
OEM Version for Windows™

User Documentation

Valid for

Control   *Software Version*
SINUMERIK 840C/CE   6

(Standard/Export Version)

SIEMENS

07.97 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.

Edition	Order No.	Remarks
06.94	6FC5198-3AA60-0BP0	**A**
11.94	6FC5198-4AA60-0BP0	**C**
09.95	6FC5198-5AA60-0BP0	**C**
04.96	6FC5198-5AA60-0BP1	**C**
07.97	6FC5198-6AA60-0BP0	**C**

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

Edition	Order No.	Remark
09.97	6FC5198-6CA00-0BG0 (READ)	**C**
09.97	6FC5198-6CB00-0BG0 (PRINT)	**C**
09.97	6FC5198-6CC00-0BG0 (NET)	**C**

For more information please refer to the Internet:
http://www.aut.siemens.de/sinumerik

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

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

Subject to change without prior notice.

Preliminary Remarks

How the documentation is organized

The SINUMERIK documentation is organized in four parts:

- General documentation
- User documentation
- Manufacturer / service documentation
- OEM documentation

Target group

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

Standard scope

This Operator's Guide only describes functions that are part of the standard scope. Options or modifications carried out by the machine-tool manufacturer are documented by the machine-tool manufacturer.

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 oblication to supply such functions with a new control or when servicing.

Help

Note

For better orientation, the following information is provided in the appendix:

- List of Abbbreviations
- Glossary

!

Important

This Operator's Guide applies to:

SINUMERIK 840C OEM Version for Windows™ Software Version 5 (5.4) and 6.

⚙

Machine manufacturer

For safety reasons, some of the functions are protected from access by unauthorized persons. Please consult the information provided by the machine manufacturer.

Notes

The following symbols with special significance are used in the documentation:
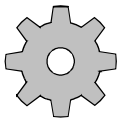
Note

The "Note" symbol always appears in this documentation to draw your attention to information relevant to the subject at hand.

!

Important / Caution

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

▯▮

Ordering data option

Occasionally you will encounter in this documentation the symbol shown here with a note referring to an option. The function described is only executable if the option has been ordered with the control.

⚙

Machine manufacturer

The symbol shown here appears in this documentation whenever the machine manufacturer can influence or modify the described function. Please observe the machine manufacturer's specifications.

Warning notes

The following warning notes with graded degrees of importance are used in this documentation.

Danger

This symbol appears whenever death, serious personal injury or substantial material damage **will** occur if the appropriate precautions are not taken.

Caution

This symbol appears whenever minor personal injury or substantial material damage **can** occur if the appropriate precautions are taken.

Warning

This symbol appears whenever death, serious personal injury or substantial material damage **can** occur if the appropriate precautions are not taken.

MS-DOS®	is a registered trademark of Microsoft Corporation
MS®	is a registered trademark of Microsoft Corporation
Microsoft®	is a registered trademark of Microsoft Corporation
Windows™	is a trademark of Microsoft Corporation

Contents

Introduction - SINUMERIK 840 C and its Operation

1

6FC5198-6AA60-0BP0

1.1   Overview

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

The CNC implements the following basic functions (in conjunction with a machine tool or suchlike):

- 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.

1.2   Design of the control

Structurally, the SINUMERIK 840C CNC can be divided into the following three components:
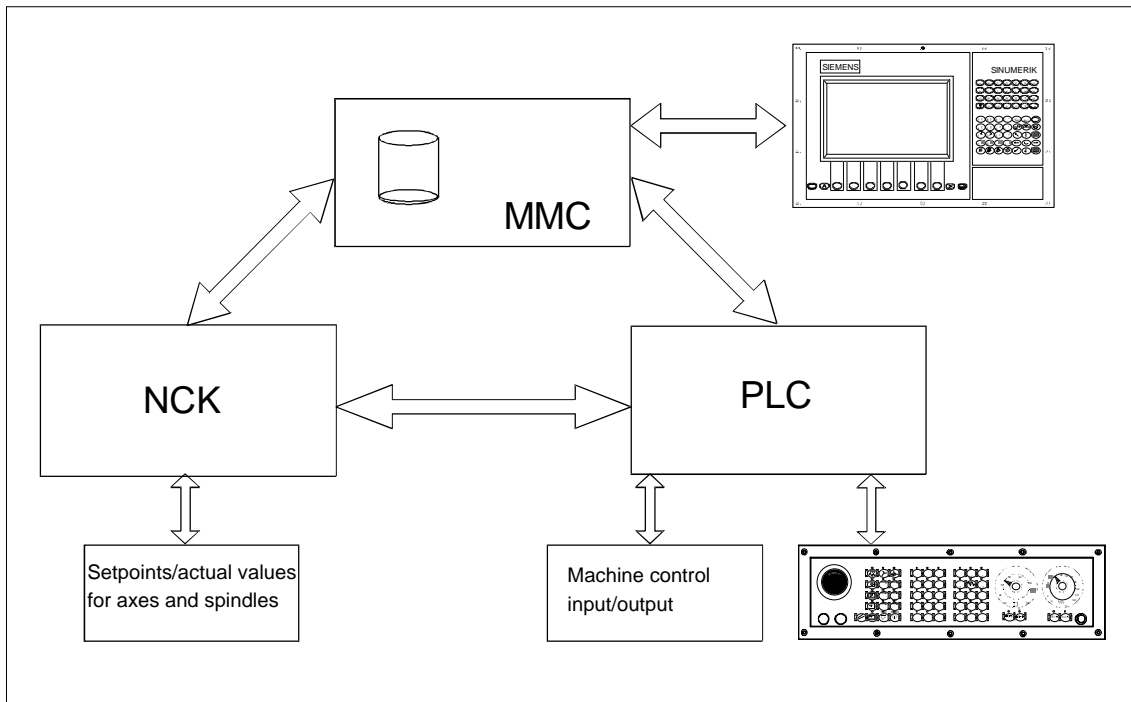
Fig. 1-1   Structure of SINUMERIK 840C

The three components perform the following functions:

NCK

The main task of the NCK (NC kernel) is to convert the program blocks of a part program into the traversing movements on the axes. Allowance must be made for all the required compensating values such as tool offsets, zero offsets, etc.

PLC

The PLC mainly performs simple control and monitoring tasks. The PLC program must assure the smooth execution of all machine functions without endangering man or machine.

MMC=PC

A PC has been integrated in the SINUMERIK 840C CNC for MMC (**M**an **M**achine **C**ommunication). In addition to data management tasks, this PC performs functions related mainly to the transfer and visualization of data exchanged between the NCK, PLC, MMC and operator. This PC runs under the MS-WINDOWS operating system.

1.3 Switching the control on and off
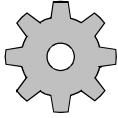
Machine manufacturer

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

Switching on the control

Please note:

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

After the control has been switched on, different messages appear on the screen for a few seconds. The control software is loaded, i.e. the MMC, NCK and PLC operating system software is loaded from the hard disk onto the various components.

The PLC user program is **not** loaded when the control is powered up, but is loaded into the battery-backed RAM memory during start-up.

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

1. The operating system software is loaded

2. In the next step, the user data are transferred from the hard disk of the MMC into the NCK memory. The machine manufacturer can configure which individual data are to be transferred.
   Both the Siemens and user cycles can be loaded in this way. A cycle disable feature provides protection against unauthorized reading and editing of data both in the NCK memory and on the MMC side.
   The machine data (TEA1, TEA2, etc.) are **not** normally loaded during power-up. These data are stored in the battery-backed static RAM of the NCK.

3. The system now loads all of the data stored in the STANDARD workpiece. The machine manufacturer can also 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 in the NCK memory 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.
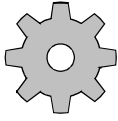
Machine manufacturer

Via machine data the machine manufacturer can configure whether, after restart of the control, the workpiece last active is to be loaded again when the control is switched on. However, before switching off the control a workpiece must have been loaded and selected. On restart of the control the workpieces are loaded one after the other starting with the workpiece in channel 1 to channel n. Generally, the STANDARD workpiece is loaded first.

Switching off the control

Before switching off the power supply the control must always be "powered down" in order to assure the integrity of the file system on the hard disk and avoid loss of data.

The control is powered down by selecting the **EXIT** command in the menu.
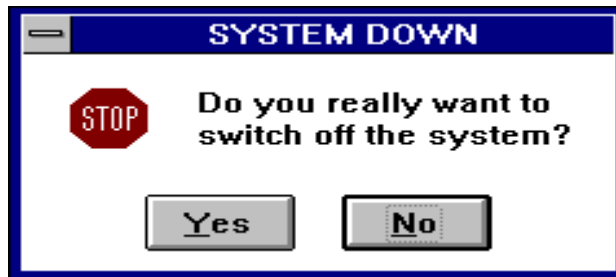
Fig. 1-2   SYSTEM DOWN query

The power supply may only be switched off when the above safety prompt has been confirmed with "Yes" and the "SYSTEM DOWN" screen has appeared.

!

Important

Never switch the power supply off during normal operation. The control must always be "powered down" by selecting the **End** command in the **Diagnosis** menu before it is switched off.

1.4 Structure of the user interface

MS-Windows   The SINUMERIK 840C is a CNC control with an MMC based on the MS
                      Windows operating system. You will therefore find on the screen the same
                      familiar elements that you have encountered when working with other MS-
                      Windows applications.

Area Switchover   The **Area Switchover** is the central program on the SINUMERIK 840C OEM
                      Version for Windows. You can use it to select 5 hierarchically organized
                      functional areas. The Area Switchover is activated by the following key:
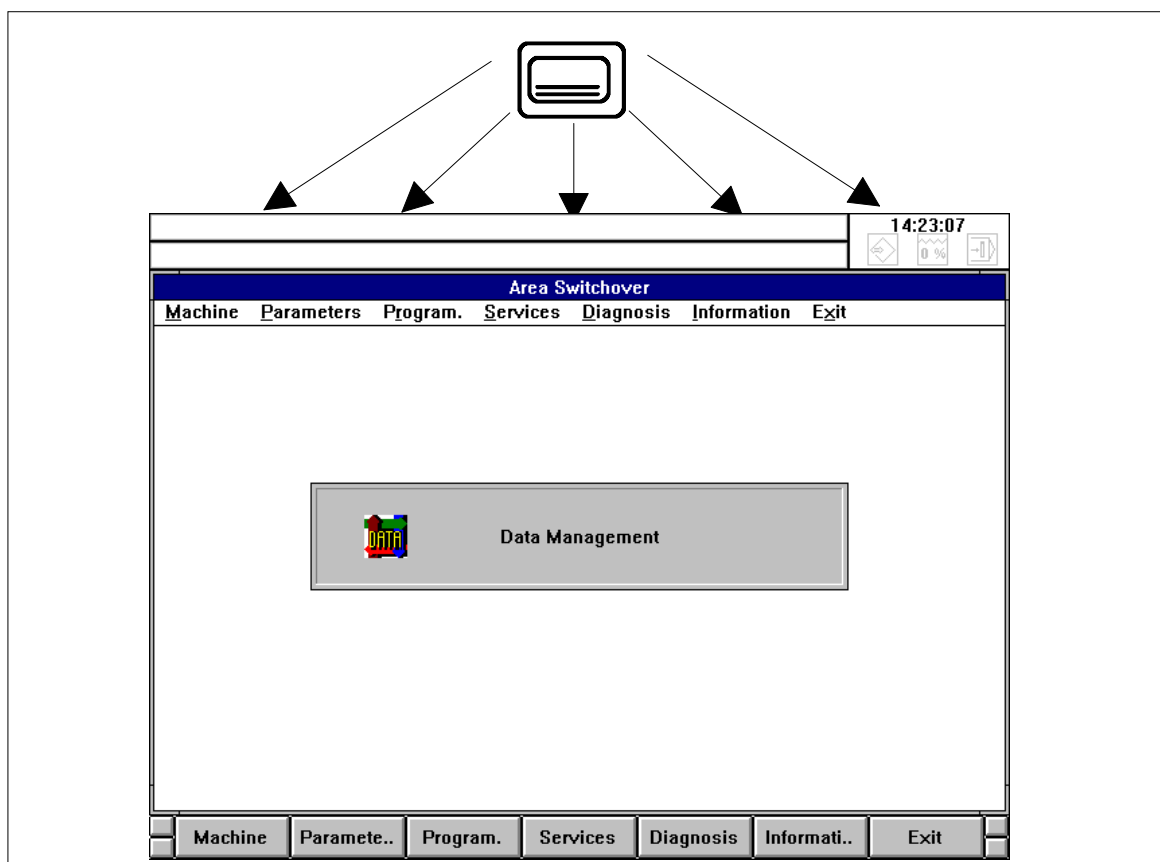
Fig. 1-3   Area Switchover

You can subsequently open the individual menus by selecting the items on the
menu or softkey bars and activate the various user areas from there.

1.5 NCK and MMC areas

Most of the individual user areas on the SINUMERIK 840C are in two main groups:

- NCK areas

- MMC areas

The operating principle differs slightly for each of the groups. While the SYSTEM 800 operating principle is used in the NCK, the user interface in the MMC areas is oriented to the MS-Windows operating system.

NCK areas

The NC screen contains "machine displays" which present the user with information communicated from the NCK or PLC. Reciprocally, all keyboard input on the operator panel is relayed to the NCK or PLC.

The selection of an NCK area is indicated by the appearance of the mode in the display.

Fig. 1-4   NCK area: Machine

       6FC5198-6AA60-0BP0

The following list provides an overview of all the NCK areas:

- Machine

- Parameters

- Programming

    - Edit NC

- Diagnosis

    - PLC Status

    - NC Service

    - NC/PLC Startup

    - Machine Data

- Information

    - NC Information

Softkeys

Within the NCK areas you will find a hierarchical menu structure operated exclusively using softkeys.

The softkeys are located below the screen. Various functions are assigned to the softkeys. The function performed when the key is pressed is displayed in the softkey bar.
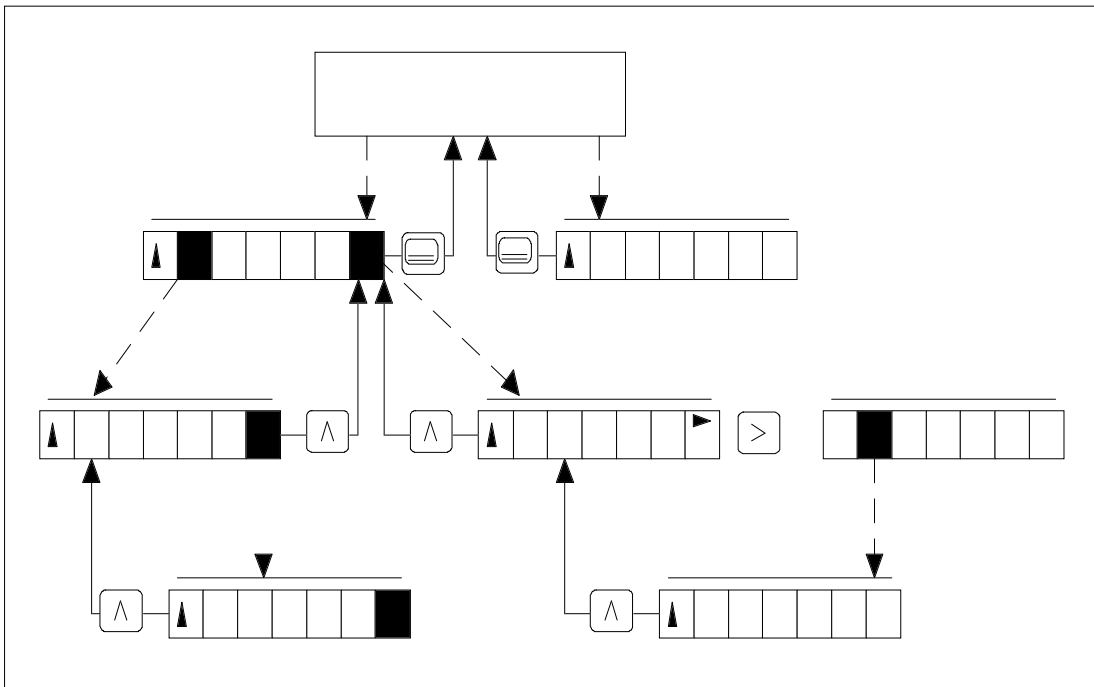
Fig. 1-5   Operating structure in NCK areas

SINUMERIK 840C, OEM Version for Windows (BA)

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

A red arrow in the extreme left softkey window indicates a return to the next higher level:
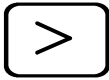
With the ETC key, you can expand the softkey menu if an extension is available.

If there is an arrow in the 7th softkey window (on the right),

this key is active.

You can press the Machine Area key to change directly from any area to the **Machine** area.

Menu storage

A menu storage function has also been implemented in the NCK areas. The last active menu of any NCK area (e.g. R parameter display) is automatically saved.

If you then select a different NCK area (e.g. Machine), you can use the key combination illustrated on the left to return to the same point in the starting menu (R parameter display).

6FC5198-6AA60-0BP0
SINUMERIK 840C, OEM Version for Windows (BA)   1-9

MMC areas

These areas provide the user with information on data in the MMC. The mode is not displayed in the MMC areas.

MS-Windows

The user interface in these areas of the SINUMERIK 840C is based on MS-WINDOWS. You will probably be familiar with the windowing technology used here.

In MS-Windows, there is a separate window for each MMC area. If another MMC area is selected, the original window is overlaid.

Task manager

You can use the task manager to switch between the different MMC areas (windows). If you call an MMC area that you have already deselected, the last display for that area is automatically displayed again. Menu storage as featured in the NCK area is therefore not necessary.
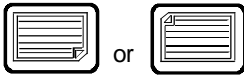
You can call the task manager using the "Page" keys in **Area switchover**.
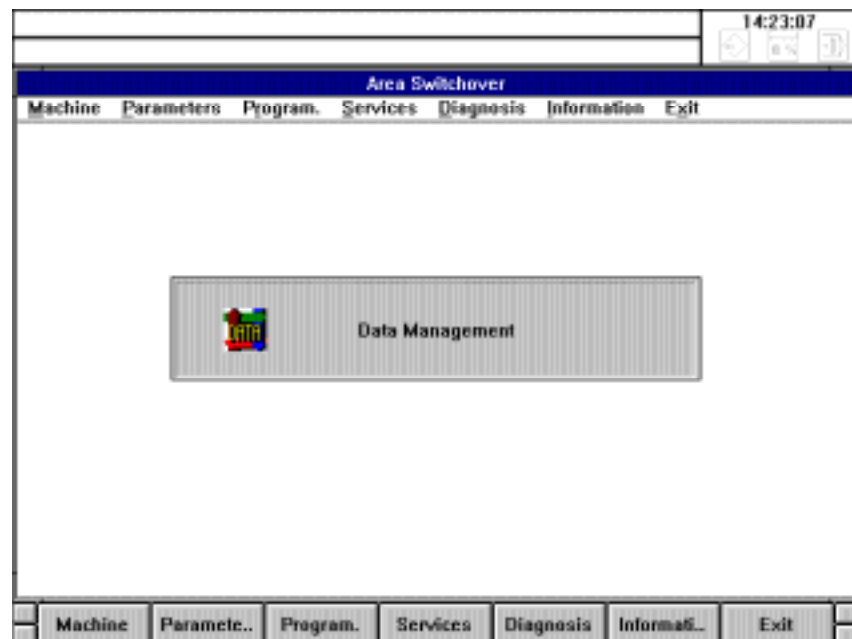
Fig. 1-6   Area switchover with active task manager

With SINUMERIK 840C OEM Version for Windows, the applications can be controlled using both the standard MS-WINDOWS operating elements and softkeys.

1.6 Data management

The SINUMERIK 840C user interface also provides its own data manager with integrated editor.
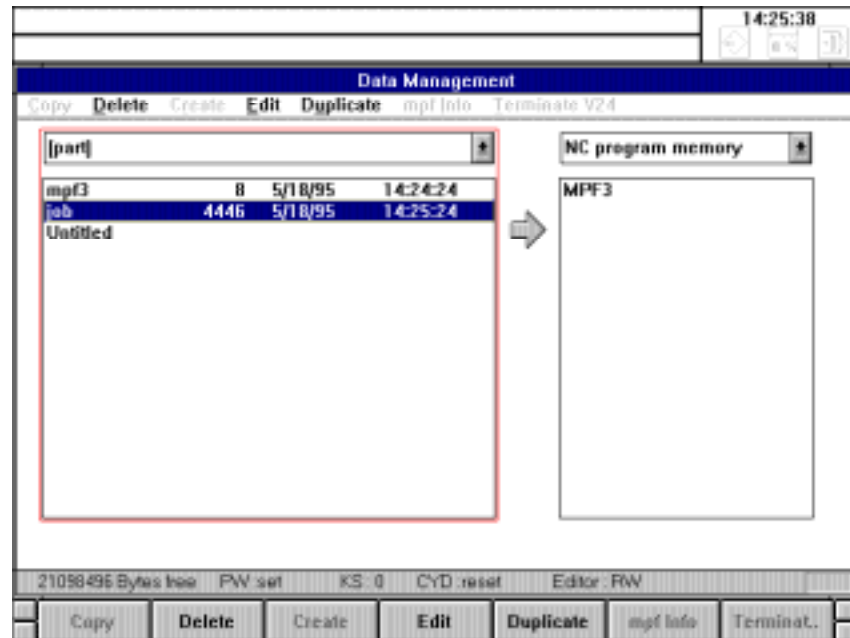
Fig. 1-7   NC Data Management

The **Data Management** area can be used as a convenient and efficient means of managing and editing the data on the hard disk. The following functions can also be performed:

- Transfer of data between the NCK and MMC
- Transfer of data between the MMC and a peripheral device

Directories

To help the user keep track of the large volumes of data stored on the hard disk, files are organized into "directories". A directory is similar to a file binder in a filing cabinet. All of the data belonging to a particular project are stored in the same directory.

Workpieces

Similarly, the SINUMERIK 840C OEM Version for Windows allows all of the data belonging to a particular workpiece to be stored in a separate directory. This type of directory is referred to here as a workpiece.

The main advantage of workpiece-oriented data management is the ability to transfer all of the data required for a workpiece from the hard disk to the NCK or peripheral device with a single operator action.

1.7 File manager and control panel

The **File Manager** and the **Control Panel** of MS-WINDOWS are available on the SINUMERIK 840C for global file manipulation and system settings within the MMC areas.
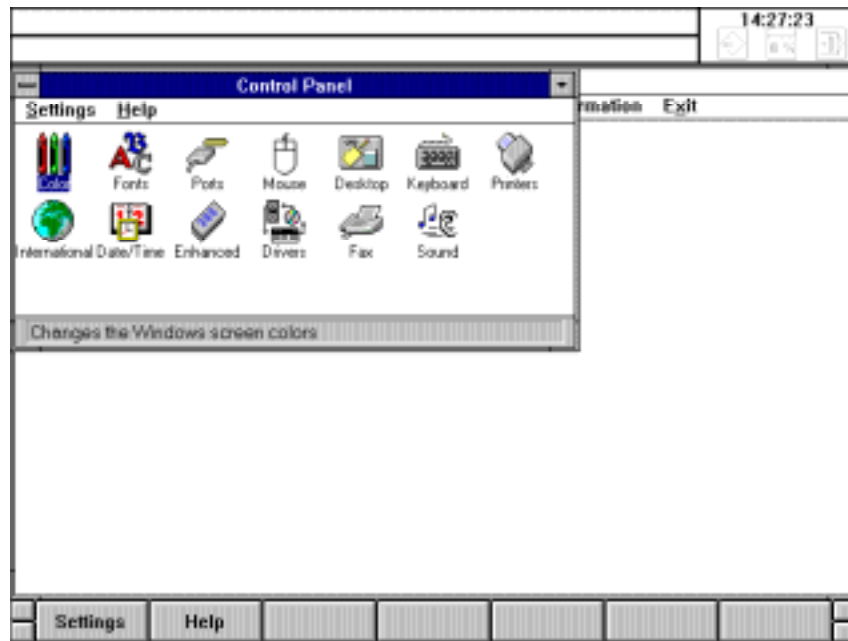
Fig. 1-8   MS-WINDOWS Control Panel

The **Control Panel** is used particularly for the following settings:

- Screen colors
- Time
- Installation and configuration of a printer

The **File Manager** and **Control Panel** can be selected from the **Diagnosis** menu in the Area Switchover.

Selection of these menu items is password-protected as standard.

1.8   Data backup

Note

The SINUMERIK 840C allows the storage of large volumes of data on the
hard disk. Siemens recommends you strongly to keep a copy of all relevant
data on an external data device (such as a programming unit or PC, etc.), as
the data can be irrecoverably lost in the event of a hard disk failure.

1.8.1   RS-232C (V.24) output

You can use the RS-232C (V.24)/20mA interface on the MMC CPU to output
your user data from the hard disk in binary or punchtape format. The interface
switches automatically RS-232C (V.24) between and 20mA mode according to
the connected device and cable used.

1.8.2   Data management/PCIN

The output of NCK data in punchtape format is performed using the data
manager. Other user data can be input or output in binary format with the aid of
the PCIN data backup program.

1.8.3   VALITEK streamer

A complete copy of all the data on the hard disk can also be made using the
VALITEK streamer. The Valitek streamer is connected to the parallel
interface.

❗   **Important**

Remember to make regular backups of your data so that you have a recent
copy of data which you can reload onto the control in the event of an error.

You can print your data on a printer connected to the parallel interface.

The FD-E2 diskette drive can also be used for data backup (optional).

1.8.4   CD-ROM access via PC link software (SW 6 and higher)

PC link

Data can be stored on an external PC by using the user interface "PC link".

Installation procedure

In software version 6 and higher, it is possible to update the software via a PC link. The software is delivered on a CD.

1. Install the PC link on the external PC by using the "install.bat" file.

2. Connect the control to the external PC by means of a parallel cable.

Note

The PC link connection required for the installation and start-up has not been EMC tested as required for normal operation, and may therefore only be used for servicing purposes (parallel transfer cable,
Order No. 6FX2 002-1AA02-1AD0).

---

3. For how to proceed with the installation from here, please refer to the "readme.txt" file in the "siepclnk" directory. The "readme.txt" file is generated with the "install.bat" file.

Note

Before initiating backup or restore on the external PC, you have to select the correct menu item on the control (Backup or Restore).

!

Important

The menu items Backup and Restore in the PC link program on the external PC are enabled according to the selection on the control (Backup or Restore).

Expansion possibilities with Backup/Restore User Data

The paths which are to be saved during backup, are stored in the "back840c" file. The control file can be selected via the "Settings/Controlfile" menu item and saved in the "pc-link.ini" file by activating "Settings/Save". This lets the operator save any of his or her own user files.

1.8.5   Computer link

Communications modules are optionally available for setting up a computer link in the NCK area of the SINUMERIK 840C.

The machine manufacturer can also implement his or her own computer link. The SINUMERIK 840C offers the following components for networking:

- Standard PC network cards. These can be plugged into the "AT box" (central unit).

- Pocket network adapters. These connect a PC to the network via the parallel interface.

Please refer to the machine manufacturer's description for further information.

1.9 Online help

The SINUMERIK 840C allows users to call up help screens for fast information in relation to a task, function or command.

NCK areas

The following features are provided for calling up help displays in these areas:

- 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.
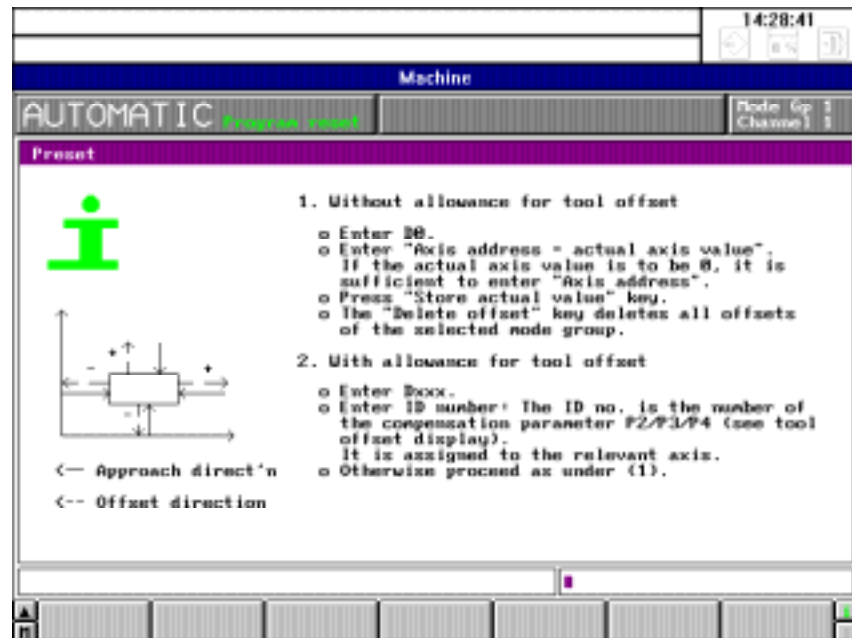
Fig. 1-9 Online help display in the machine area

Alarm/message window

In this window, the help information is based on the MS-WINDOWS help system.
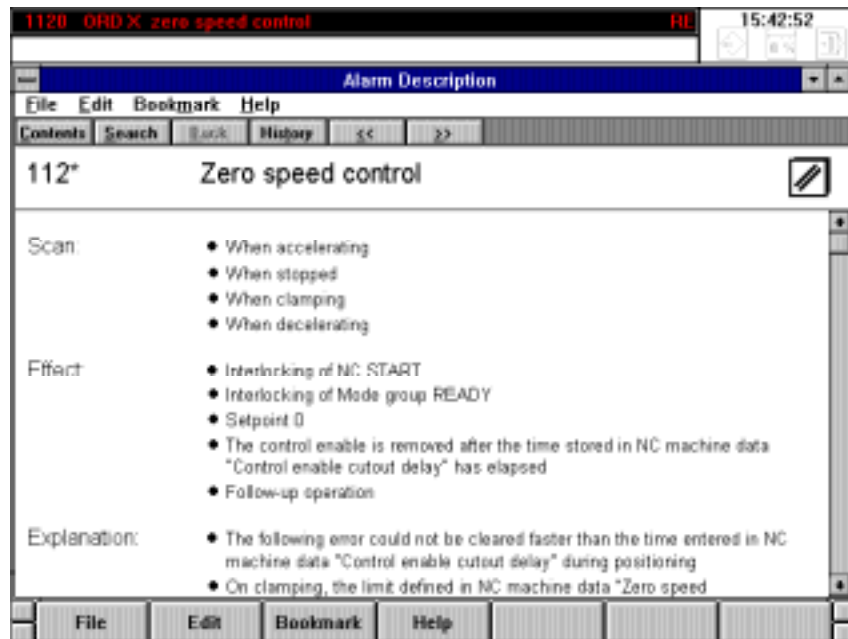
Fig. 1-10   Online help in the alarm window

- First select **Alarm Window** from the **Diagnosis** menu and position the selection bar on the desired alarm.

Press this key to display an MS-WINDOWS help window containing additional information relating to queries, causes and remedies. Use the ARROW keys to scroll within the help window. ∎

       6FC5198-6AA60-0BP0

SINUMERIK 840C, OEM Version for Windows (BA)

SINUMERIK 840C, OEM Version for Windows (BA)

Operator Interface

2

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" 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 colour LC 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
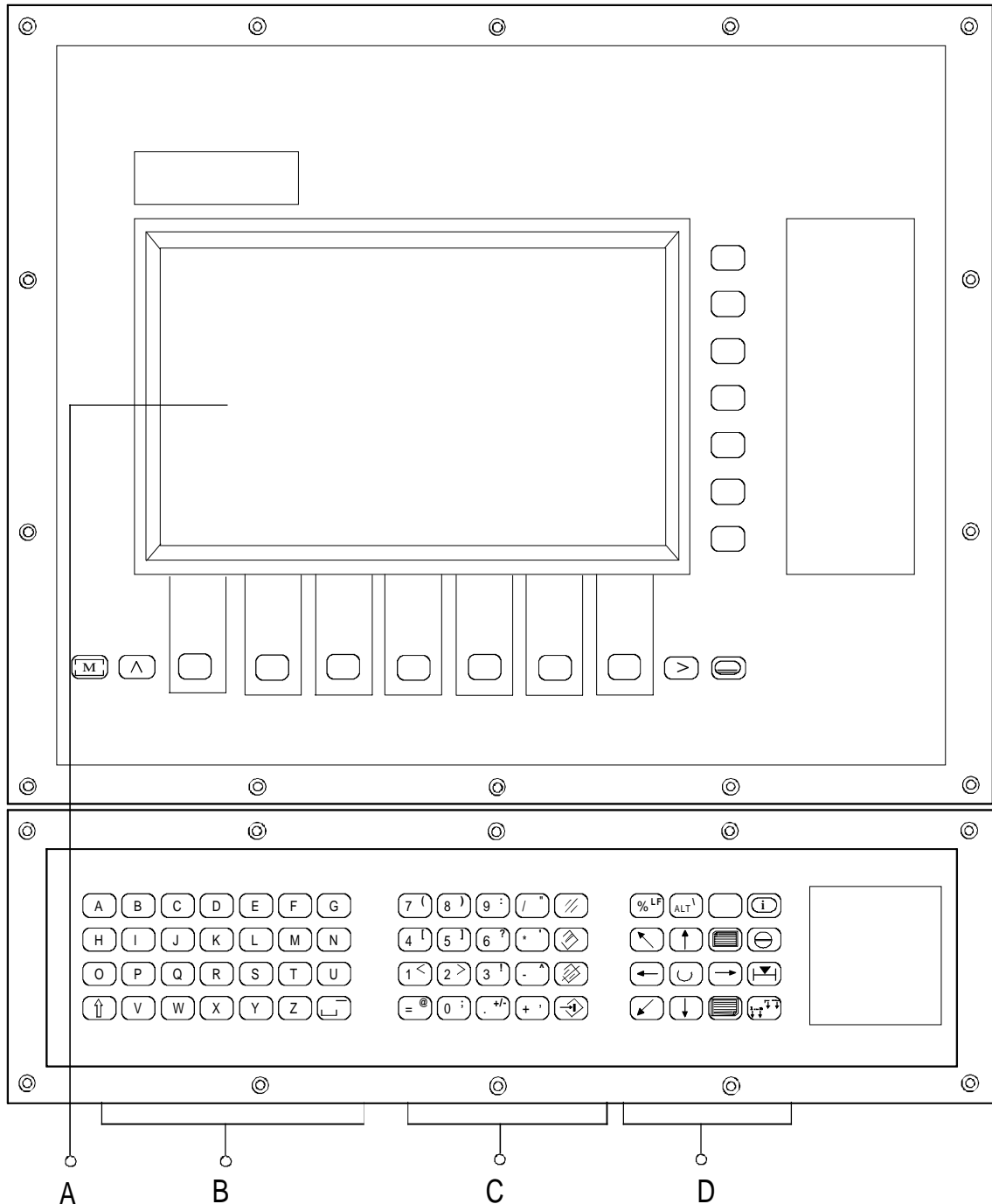
Fig. 2-1   19" operator panel with 14" colour monitor
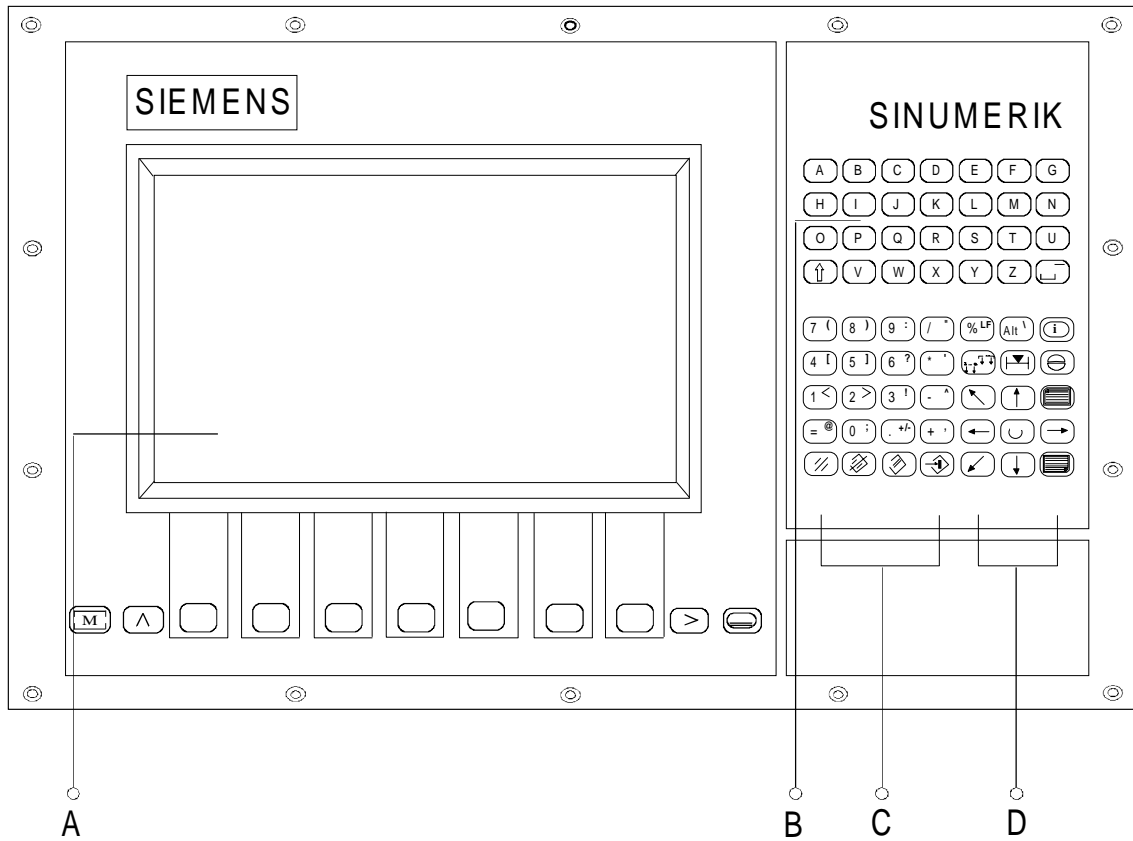
Fig. 2-2   SINUMERIK 9.5"/10" slimline operator panel

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   Full PC keyboard/MF2 keyboard

Full keyboard A standard MF2 keyboard (full keyboard) can be connected to the operator panel interface.

A set of key caps is supplied with this standard full keyboard. These key caps correspond to the symbols on the operating panel and can be mounted on the standard full keyboard as shown in Fig. 2.3.

The functions of the machine control cannot be implemented on the MF2 keyboard.

MF2 keyboard An MF2 keyboard (American keyboard) can be connected directly to the keyboard input on the MMC module. The following restriction applies to this configuration.

Note

When connected directly to the MMC module, the MF2 keyboard does not comply with the requirements of a SINUMERIK control with respect to noise immunity. The MF2 keyboard may therefore only be used for start-up and servicing.

Fig. 2-3   Full keyboard

6FC5198-6AA60-0BP0

SINUMERIK 840C, OEM Version for Windows (BA)
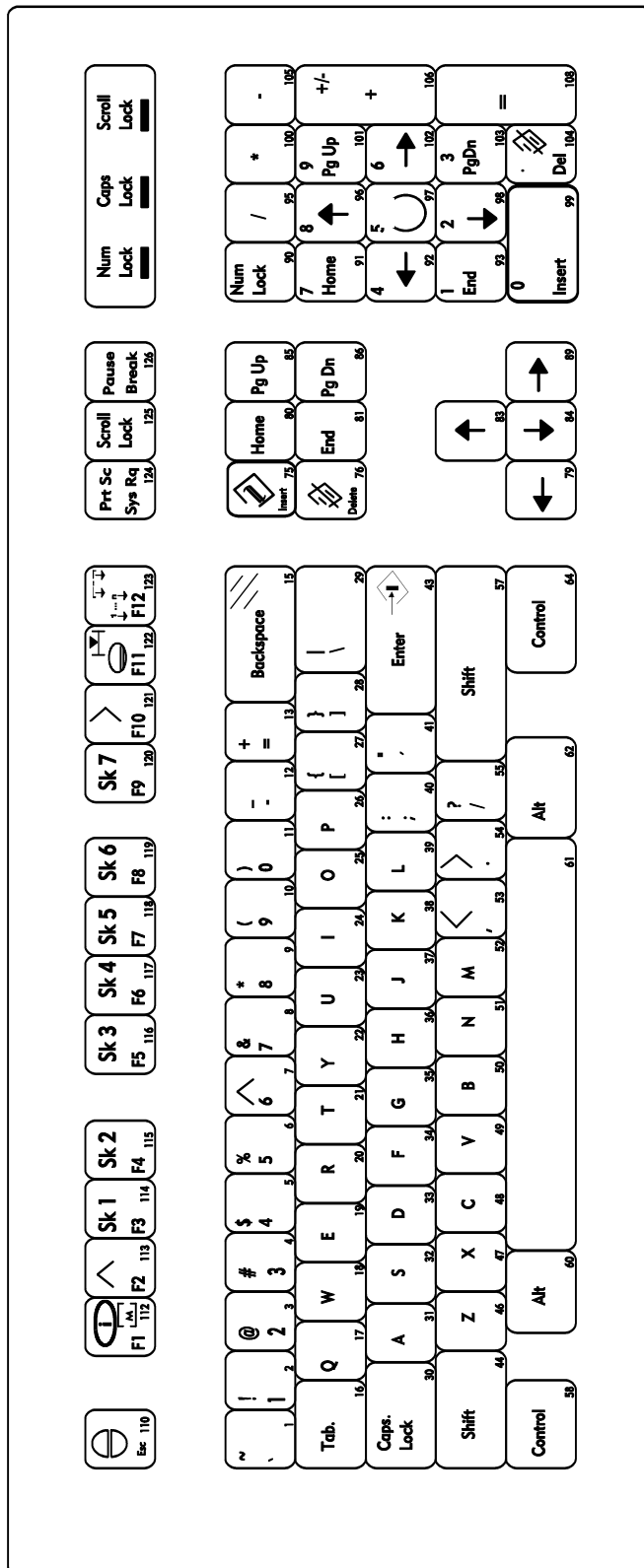
2.2.3   Operating elements of the operator panel keyboard/full keyboard

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

Horizontal softkey bar (full keyboard): standard applications on the SINUMERIK 840C are operated exclusively using the horizontal softkeys.
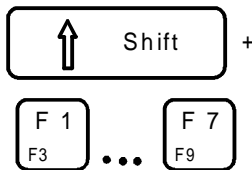
Vertical softkey bar (full keyboard): The designation F1 to F7 corresponds to the designation on the additional key caps of the standard keyboard (see also the illustration of the full keyboard). The vertical softkeys are not used in the standard version and are therefore not displayed.

MACHINE AREA key. Press this key to switch from any user area to the **Machine** area.

RECALL key: Return to the higher level menu

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

DATA AREA key: the **Area Switchover** is selected (or deselected if it is already selected) by pressing this key. When selected, it displays the **Area Switchover** on top of the currently active application-.

SHIFT key: Switching over keys with double allocation; not self-retaining, i.e. two keys must be activated.

With the combination SHIFT + DATA AREA, an NCK menu can be reselected if it was deselected by another NCK application. It is thus possible to move from an NCK menu (e.g. R parameter display) to a different application in the NCK area (e.g. Machine) and then to return to the same point in the parameter display.

A .. Z	Lower case letter a ... z, activated with SHIFT Upper case letter A ... Z
␣	Underscore SPACE, blank Press the SPACE key to select the checkbox in a dialog box.
1 <	Special character: less than Digit 1
2 >	Special character: greater than Digit 2
3 !	Exclamation mark Digit 3
4 [	Square bracket Digit 4
5]	Square bracket Digit 5
6 ?	Question mark Digit 6
7 (	Round bracket Digit 7
8)	Round bracket Digit 8
9 :	Colon Digit 9
0 ;	Semicolon Digit 0
/ "	Inverted commas Oblique stroke, division

SINUMERIK 840C, OEM Version for Windows (BA)

Key	Description
`* '`	<u>Apostrophe</u> Multiplication
`- ^`	<u>Special character</u> Hyphen, subtraction
`= @`	<u>Special character</u> Equal sign
`+ '`	<u>Comma</u> Plus, addition
`. +/-` or `Shift` + `+`	<u>Change of sign</u> Fullstop (point/period)
`//` or `←Backspace`	CLEAR key: deletes the character to the left of the input cursor. Identical to the <BACKSPACE> key on the MF2 keyboard.
CANCEL icon `DEL`	CANCEL key: deletes the character underneath the input cursor. Identical to the <DELETE> key on the MF2 keyboard.
EDIT icon or `Tabulator`	EDIT key for editing words in the NCK areas. The EDIT key can be used in the dialog boxes of the MMC areas to jump from one input box to the next. Identical to the <TAB> key on the MF2 keyboard.
INPUT icon or `Enter`	INPUT key: confirm input (save the edited value to memory). Identical to the <ENTER> key on the MF2 keyboard.
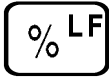
LINE FEED key: identifier for end of block
Percent sign; identifier for beginning of main program
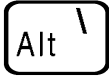
ALT/BACKSLASH key: this key is used for the selection of menus and dialog
boxes in MS-WINDOWS. It is located to the right of the LINE FEED/percent
sign key. It is not labelled in standard versions.
The ALT key is used for the selection of MMC menus and dialog boxes. When
used in combination with the SHIFT key, it generates the "\" (backslash)
character. This is useful for entering complete file paths in MMC applications.

Help
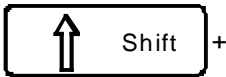
HELP key: this key can be used in machine displays to call up explanations
and information on the current operating state.
Pressing it a second time brings you back to the previous display. The "i" in the
softkey bar indicates the possibility of calling up information by pressing the
"Help" key.
You can press the HELP key in the alarm window to call up an explanation of
the active alarm.

Actual position in large
characters
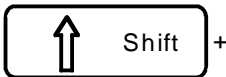
When you operate this key, the screen display of "Actual position" is shown in
double-height characters.
Operating the key again brings you back to the previous full display (with
standard size characters).
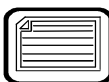
PAGE UP key: press these keys to scroll the screen one page back.
When the Area Switchover is active, the MS-WINDOWS Task Manager is
selected. You can press PAGE UP and PAGE DOWN to select the desired
application and press any other key to bring it into the foreground.

PAGE DOWN key: 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).
When the Area Switchover is active, this key activates the MS-Windows Task
Manager. You can choose an application by pressing the PAGE UP and PAGE
DOWN keys and bring it into the foreground by subsequently pressing any
other key.
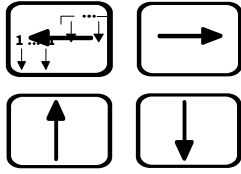
ARROW keys: with these keys, you move the cursor on the screen to the left or to the right, upwards (back) or downwards (forward).

With the ARROW keys, you can move the cursor in the input fields (toggle fields, etc.) or in the various editors (**WEdit** editor; **DIN editor**). Further information is given in the respective Sections of this Operator's Guide.
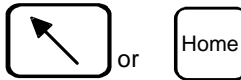
HOME key: moves the cursor to the beginning of the line in the **WEdit** editor. This key is used within a machine display to advance the input field..

END key: moves the cursor to the end of the line in the **WEdit** editor.
This key is used within a machine display to activate/deactivate the 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.

SELECTION/SEARCH key
Select toggle fields in machine displays and text in the FlexOS editor/Search text.

SHIFT + SELECTION
(reset a toggle field in a machine display)

Acknowledge alarm

ACKNOWLEDGE ALARM: press this key in the alarm window or when the Machine area is selected to acknowledge the information from the NC monitoring system displayed in the alarm line such as:

- Alarm number and alarm text for CANCEL alarms

- The machine manufacturer can configure whether this key is to act on all channel-specific CANCEL alarms or only on the CANCEL alarms of the current channel.

In MMC applications, the key has the same function as the ESC key on the MF2 keyboard.
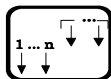
Change of mode groups
Channel switchover

Changing mode
groups

The NC channels are grouped together in mode groups. With the SINUMERIK 840C up to 2 mode groups can be selected.
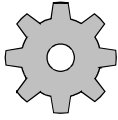
Machine manufacturer

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.

2.2.4   The operating elements of the machine control panel

Machine control panel
for a milling machine
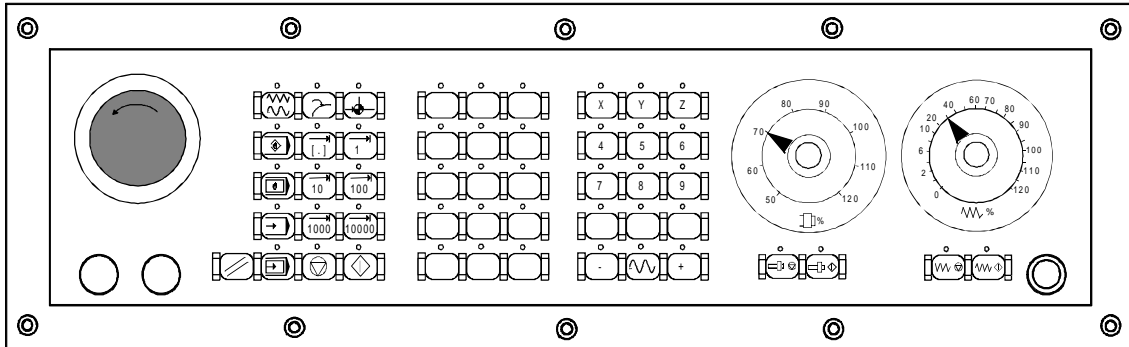
Machine control panel
for a turning machine
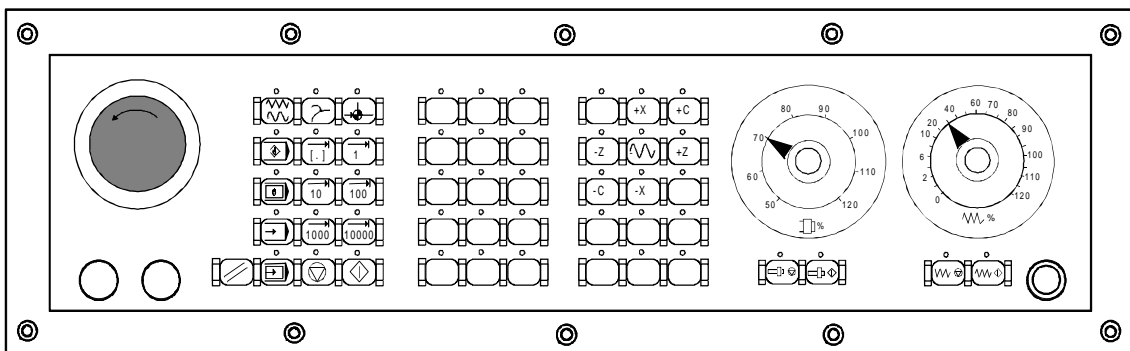
Fig. 2-4   Machine control panels

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
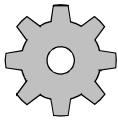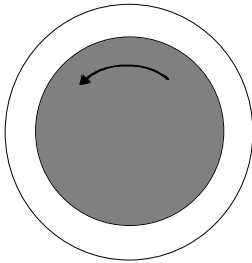
You operate the red button in emergency situations:

- When human life is in danger

- When there is a risk of the machine or workpiece being damaged.
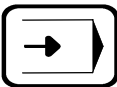
Operation of the "Emergency stop" button generally brings all drives to a stop with maximum braking torque.

Machine manufacturer

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

Operating modes

Table 2-1   Overview - Operating modes
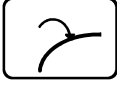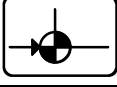
Key symbol	Definition	Designation of operating mode
	The axes are traversed continuously in JOG mode using the direction keys or incrementally using the direction keys or the handwheel.	**JOG** Set-up (**Jog**ging)
	Interactive creation of programs	**TEACH IN**
	The machine is controlled by executing a block or a series of blocks. The operator panel is used to input blocks.	**MDA** **M**anual **D**ata **A**utomatic
	The machine is controlled by automatic execution of programs.	**AUTOMATIC**

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

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

REPOS and reference point

Table 2-2   REPOS - Reference point

Key symbol	Definition	Designation of function
	Repositioning, approach contour again in JOG/TEACH IN mode	**REPOS** **Repos**ition
	Approach reference point in JOG mode	**REFPOINT** Approach reference point
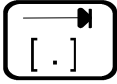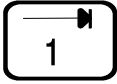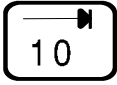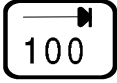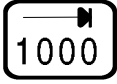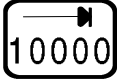
The REPOS function is only active in JOG and TEACH IN mode and can be selected only in these modes.

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 also cancelled if you change the operating mode.

INC function

Table 2-3   Overview - Submode INC

Key symbol	Definition	Designation of function
[.]	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
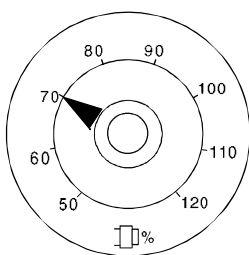
- The rotary switch with 16 notched positions enables you to reduce or increase the programmed spindle speed S (corresponds to 100%).

- A machine data determines whether the spindle override switch 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 screen.

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

Step size: 5% from position to position
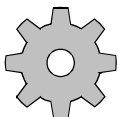
Machine manufacturer

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

Spindle stop
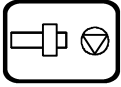
When you press the SPINDLE STOP key:

- The spindle speed is reduced down to zero

- The associated LED lights up as soon as SPINDLE STOP is accepted by the control, for example also with M05 (see Programming Guide).

Example for use of SPINDLE STOP

- To effect tool change

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

Spindle start
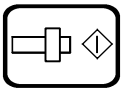
When you press the "Spindle start" key:

- 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.
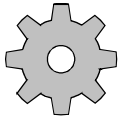
Machine manufacturer

The following is defined in the machine data or setting data:
- Max. spindle speed
- Values for the spindle speed override
(see machine-tool manufacturer's instructions)

Feedrate control

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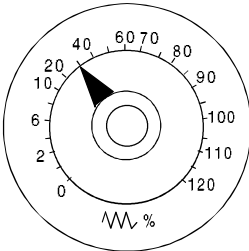
The rotary switch with 23 notched positions allows you to reduce or increase the programmed feedrate value F (corresponds to 100%).

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

Setting 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%
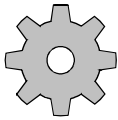
Machine manufacturer

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 hold

When you operate the FEED HOLD 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 HOLD is accepted by the control.

Examples for use of FEED HOLD:

- During application in MDA mode, a block with a fault is discovered

- To effect tool change

Feed start

When you operate the FEED START key:

- 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.
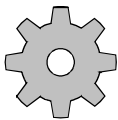
Machine manufacturer

The following is configured by the machine manufacturer:

- The feed and rapid traverse rates

- The values for feed override positions

- Whether the feed override switch is also active for rapid traverse

- The axis names (see machine-tool manufacturer's specifications)

**Direction keys,
turning machine**
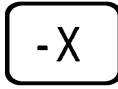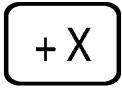
$\boxed{+ X}$ $\boxed{- X}$ You traverse the axis marked X.
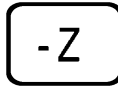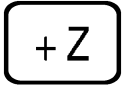
$\boxed{+ Z}$ $\boxed{- Z}$ You traverse the axis marked Z.
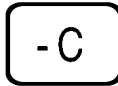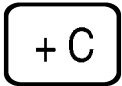
$\boxed{+ C}$ $\boxed{- C}$ You traverse the axis marked C.

**Rapid traverse
overlay**
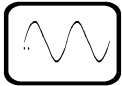
$\boxed{\sim}$ When you operate this key at the same time as any of the keys above, the axis is traversed in rapid traverse mode

**Direction keys,
milling machine**

$\boxed{X}$ $\boxed{Y}$
$\boxed{Z}$
    You select the axis marked X, Y, Z etc.
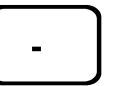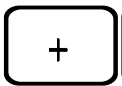
$\boxed{+}$ $\boxed{-}$ You traverse the selected axis in positive or in negative direction.

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**
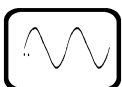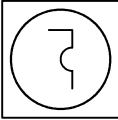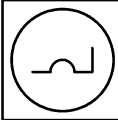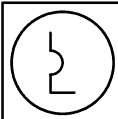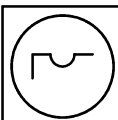
$\boxed{\sim}$ When you operate this key at the same time as any of the keys above, the axis is traversed in rapid traverse mode.

Keyswitch

The SINUMERIK 840C keyswitch has 4 positions which are evaluated by 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

Table 2-4

Position of switch	Withdraw position	Function
Position 0	-	Data is 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 and setting data
Position 3	0+1+2+3 key 3 red	The STANDARD workpiece is not loaded when the control is powered up. Use only for start-up and servicing!
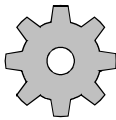
Machine manufacturer

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

When you operate the "Reset" key:

- Execution of the current part program is interrupted, if the key releases a mode group reset (= reset of all channels).

- Messages are cleared from the monitoring system unless they are POWER ON or acknowledgement 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.
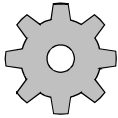
Machine manufacturer

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

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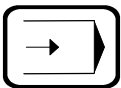
This function allows you to execute a part program on a block-by-block basis.

The "Single block" function can be activated in the AUTOMATIC, TEACH IN and MDA modes.

When SINGLE BLOCK operation is active:

- The SBL (**S**ingle **B**lock) message is shown on the CRT display,

- The current block of the part program is executed only when you press the "NC start" key,

- When the current block has been executed, processing is stopped,

- The following block can be executed by pressing the "NC start" key again.

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

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

The function can be deselected by pressing the key again.

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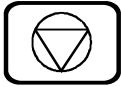
If you press the NC STOP key, processing of the current part program is interrupted. All axis movements are brought to a controlled stop. The reactions of the H and M functions in the case of NC STOP are configured by the machine manufacturer. For more information refer to the machine manufacturer's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

NC Start

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

Caution

The axis positions programmed in the current block are approached with linear interpolation after the "NC START" key has been activated. Collision danger!

2.2.5   Operating elements of the hand-held unit
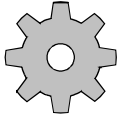
Machine manufacturer

The display and key functions can be configured by the machine manufacturer.
For further details refer to the machine manufacturer's documentation.
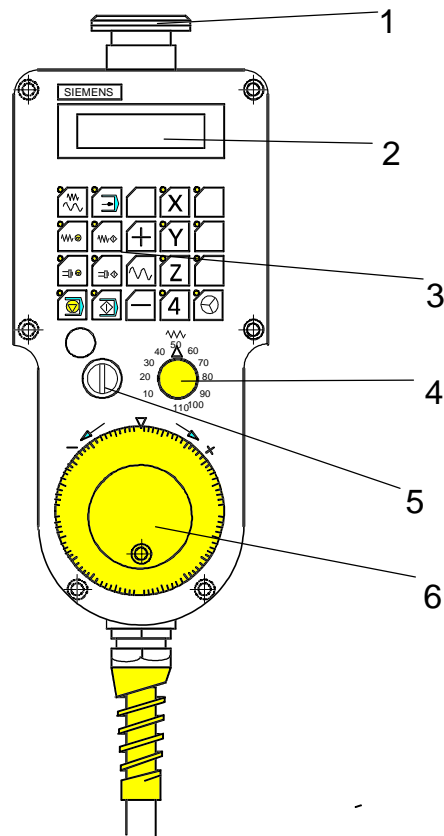
Fig. 2-5   Operating elements of the hand-held unit

Meaning of the operating elements:

1 - EMERGENCY STOP switch

2 - Two-line display

3 - Group of keys with LEDs

4 - Feed/rapid traverse/override switch

5 - Keyswitch

6 - Handwheel

2.2.6   Screen layout
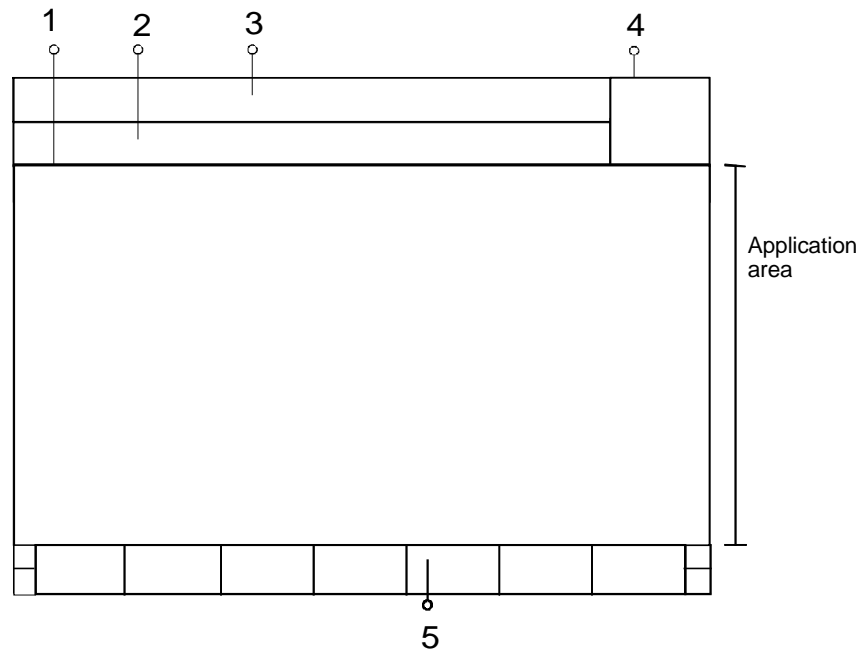
Fig. 2-6   Screen layout

1 - Application title bar

2 - Message line

3 - Alarm line

4 - Time display, general channel status display with icons

5 - Softkeys

The display is divided into various window areas. An alarm line and a message line are displayed at the top left-hand side of the screen. To the right of this area is a box displaying the current time and the icons.

The application window starts below the alarm area and extends to the softkey bar. Standard Siemens applications are operated using the horizontal softkeys. The vertical softkeys are not used and are therefore not displayed. ∎

MMC Applications

3

3.1   Introduction

Before you start working with the SINUMERIK 840C you should be familiar with some of the basic elements of MS-WINDOWS and its operation.

The control can be operated using 3 different keyboards. The operator panel keyboard and an MF2 keyboard (full keyboard) can be connected to the keyboard interface or the slimline operator panel. The operator panel interface is connected to the 1st serial interface of the MMC module via an RS 232C (V.24) cable.

A conventional MF2 keyboard can also be connected directly to the keyboard input of the MMC central unit. For reasons of noise immunity, this version may not be used for normal operation, but is intended exclusively for start-up and servicing.

When several keyboards are used together, the user should check that the <SHIFTLOCK>, <CAPSLOCK> and <ALT> keys work on all of the keyboards.

The MF2 keyboard connected to the central unit allows the operation of MS-Windows applications exactly as described in the Microsoft documentation.

The SINUMERIK 840C is operated <u>as standard</u> with the <u>operator panel keyboard</u>.

Aside from the usual key assignments for operation in the Machine area, certain keys have been redefined for MS-WINDOWS applications. In particular, key combinations such as <ALT> + <TAB> have the same effect as on the MF2 keyboard. On the operator panel keyboard, however, the two keys must be pressed consecutively (and not simultaneously).
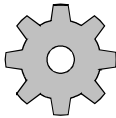
Machine manufacturer

In the new firmware version of the operator panel interface module it is possible to evaluate any key actuation on the operator panel or full keyboard connected to this in a similar way as an MF2 keyboard connected to the central unit.

Combined key inputs such as ALT + X are thus possible.
In addition , the machine manufacturer is able to plan the key allocation of the operator panel freely.

Please refer to the machine manufacturer's documentation.

The following section describes some of the basic elements of MS-WINDOWS and the operation of MMC applications using the operator panel keyboard.

Operator panel keyboard   In the following section, references to the keys on the operator panel keyboard are made in UPPER CASE CHARACTERS (e.g. CANCEL, ALT, EDIT, etc.).

MF2 keyboard   The keys on the MF2 keyboard are also enclosed in angle brackets (e.g. <ALT>, <ESC>, etc.).

Language   In general, all menus and dialog boxes, including those that directly access MS-WINDOWS resources, appear in the selected language. The language can be installed at the time of start-up and then selected in the diagnostics menu. The following languages are available as standard:

English, French, German, Spanish and Italian.

!   **Important**

If an application crashes, it can generally only be operated using the MF2 keyboard of the central unit and not the operator panel keyboard.

If no MF2 keyboard is provided, the control must be switched off and powered up again.

3.2   Components of an MS WINDOWS window

In MS-WINDOWS, you work with rectangular screen areas known as windows. All windows have certain standard elements such as a title bar and menus.

The following figure depicting an MS-WINDOWS editor illustrates the major components of a window.

Control menu bar   Titel bar   Window title   Minimize   Maximize

Menu bar

Notepad - (Untitled)

File   Edit   Search   Help

Horizontal scroll bar   Vertical scroll bar
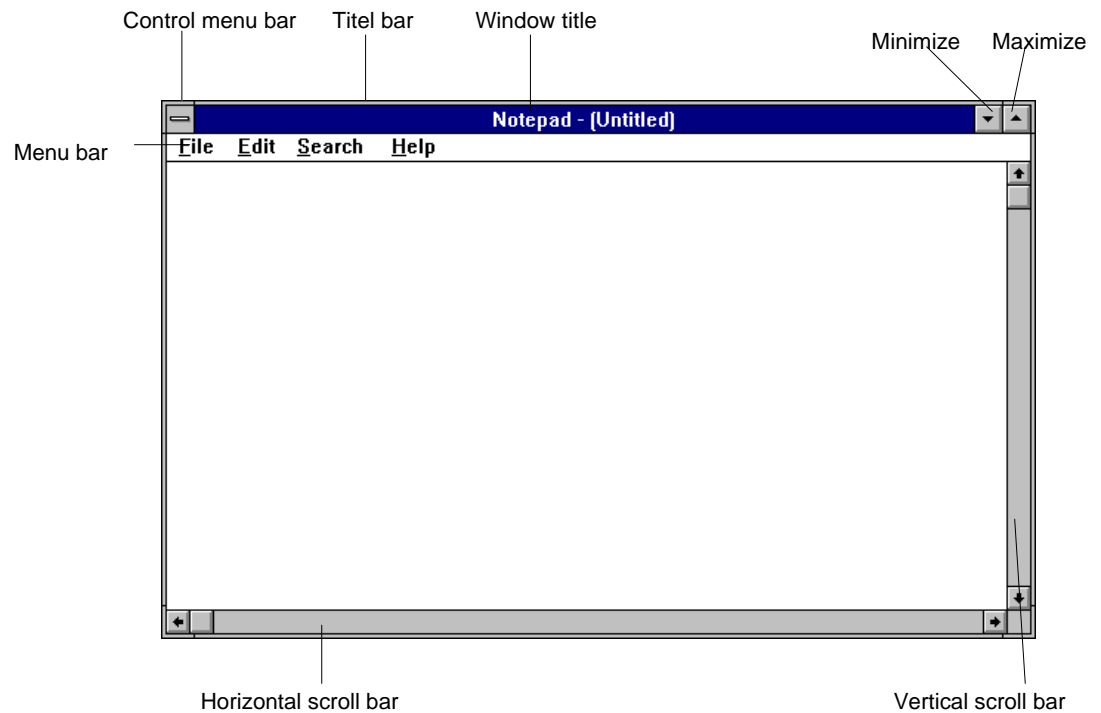
Fig. 3-1   Components of a window in MS-WINDOWS

Title bar

The title bar shows the name of the application program or the document to be edited. If several windows are open simultaneously, the title bar of the active window is displayed in a different colour or with a different intensity to all the other windows.

Menu bar

The menu bar displays the menus available in this application.

Scroll bars

The scroll bars show you the position of the currently displayed window pane in documents of more than one page in length.

Window border

The window border represents the outer limit of a window.

Note

Standard Siemens applications have no control menu and no minimize/
maximize buttons. The size and position of an application can therefore only
be altered by the machine manufacturer, not the user.

The windows in standard MS-WINDOWS applications such as the **File
Manager** or **Control Panel** also contain further elements.

Control menu box
The control menu box of an application program is located in the top left-hand
corner and can be opened by pressing the ALT and SPACE key combination or
by pressing the RECALL key twice in succession.

The control menu box of a document window belonging to an application
program can be opened by pressing the ALT and (-) key combination. Please
note that you must press the two keys consecutively (not simultaneously).
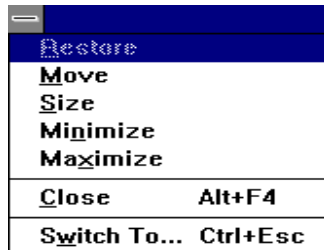
Fig. 3-2   Control menu

You can use the commands in the control menu to resize, move, maximize or
minimize a window. The **Close** command closes the window.

Minimize/Maximize
buttons
An icon is a small graphic image used to represent diverse types of application
programs. An icon appears when you start an application and subsequently
select the **Minimize** button.

Selecting the **Maximize** button enlarges a window to cover the entire area of
the screen.

3.3 Operations with windows

When you work with different applications, you will often have several windows open simultaneously on the screen. The window in which you are currently working and where you can enter data is known as the active window. You can switch between different windows (= applications) in the following way.
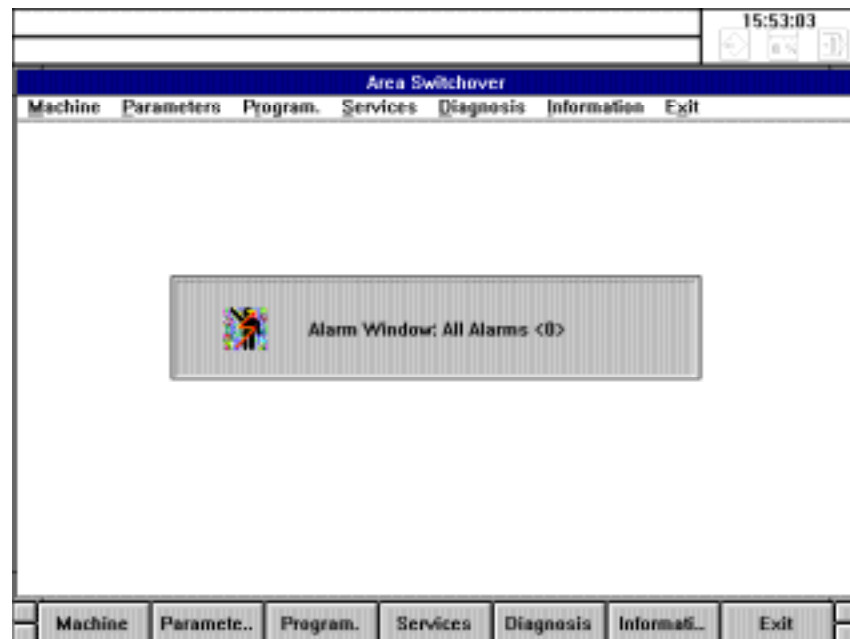
Fig. 3-3   The Task Manager of the SINUMERIK 840C

Applications in MS-WINDOWS are normally started from the **Program Manager**. On the SINUMERIK 840C, the **Area Switchover** performs this task. However, when the Area Switchover is used to select an application that is already running in the background, it simply brings that application into the foreground instead of starting a second instance. This procedure only applies to standard Siemens applications, not to general MS-Windows applications.

Task Manager

Hidden applications, i.e. applications running in the background, can be brought into the foreground using the **Task Manager**. In MS-Windows it is activated by simultaneously pressing the <ALT> and <TAB> keys on the MF2 keyboard. By holding down the <ALT> key and repeatedly pressing the <TAB> key you can select the individual applications and bring them into the foreground.

SINUMERIK 840C, OEM Version for Windows (BA)

Operator panel keyboard

You can select individual applications as follows using the operator panel keyboard:

- Press the DATA AREA key to select **Area Switchover**.

- Press the PAGE UP or PAGE DOWN key to activate the Task Manager.

- Select the desired application with PAGE UP or PAGE DOWN.

- Press any other key to deselect **Area Switchover** and activate the desired application.

Moving a window

Where MS-WINDOWS applications are provided with a control menu box, the commands listed in this box can be used to move the application window to another area of the NC screen.

- Activate the application program window that you want to move.

- Open the control menu box by pressing ALT and SPACE or ALT and (-).

- Select the command **Move** in the control menu. The pointer changes to a four-headed arrow.

- Now move the window using the ARROW keys. An outline of the window moves when you press the ARROW keys.

- Press the INPUT key when the window is in the desired position.

Changing the size of a window

You can change the size of application windows which have a control menu box.

- Activate the window you want to resize.

- Open the control menu box by pressing the key combination ALT and SPACE or (-).

- Choose the **Size** command from the control menu. The pointer changes to a four-headed arrow.

- Press one of the ARROW keys to move the pointer to the window border you want to move.

- Now press one of the ARROW keys to move the selected border in the desired direction.

- Press the INPUT key when the window is the desired size.

3.4   Working with menus

Opening menus   Almost every application based on MS-WINDOWS displays a menu bar below the title bar. The menu bar can contain the names of either menus to open or commands to execute.

Note

On the SINUMERIK 840C, the items in the menu bar are also displayed on the softkeys. The menus can be opened in various ways:
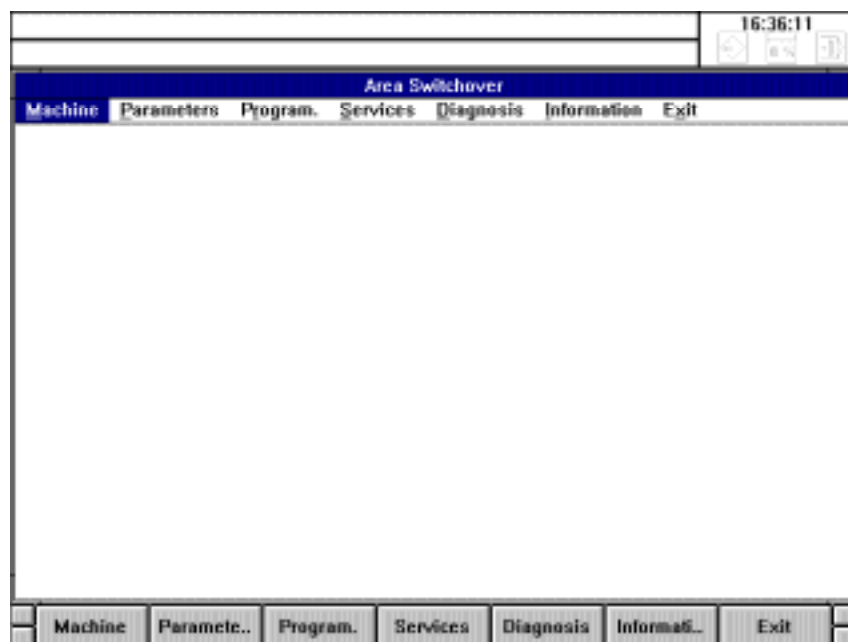
Fig. 3-4   Menu items within Area Switchover

- Press the ALT key. The selection bar highlights the first item in the menu bar. You can now select the desired menu and open it with the INPUT key.

- Press the ALT key and then enter the HOTKEY for the menu. The HOTKEY is the letter underlined in the menu.

- Simply press the softkey belonging to the item in the menu bar. The menu opens.

When a menu is opened, the first item is highlighted by the selection bar. There are two ways of selecting individual commands within a menu:
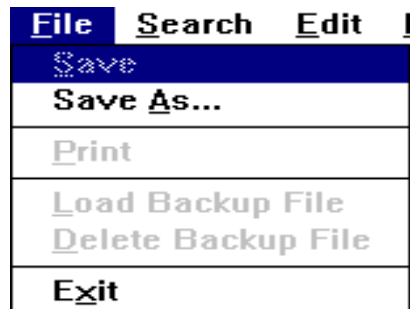
Fig. 3-5   Items in the **File** menu of the **WEdit** editor

- Use the vertical ARROW keys to select the desired menu item and then press the INPUT key.

- Press the HOTKEY. The HOTKEY is the letter underlined in the menu. The selected command is executed.

Note

In SINUMERIK 840C applications, the menu bar is displayed using the softkeys. It is therefore easy to open or execute the items in the menu bar by pressing the softkeys. The fastest way of selecting an item within a menu is to use the HOTKEY.

Closing menus

You can close a menu without selecting an item by pressing the ACKNOWLEDGE ALARM or RECALL key.

Menu conventions

The menu conventions used on the SINUMERIK 840C are identical to those of MS-WINDOWS.

Dimmed (grayed out) elements

Elements which are dimmed are not available for the active application. You may have to increase your access rights or select other elements before these commands can be executed.

Omission dots (...)

Omission dots following a menu item indicates that a dialog box with options is displayed. The options must be selected before the command is executed.

Triangle (▶„)

A triangle on the immediate right hand side of an item indicates another overlapping menu containing a list of additional commands.

3.5   Working with dialog boxes

SINUMERIK 840C applications display a dialog box when additional
information is required from the user in order to perform a certain task (e.g.
parameters for initializing the serial interface).

Omission dots (...) following a menu item indicates that a dialog box is to
follow. The illustration below shows all the components that can appear in a
dialog box.
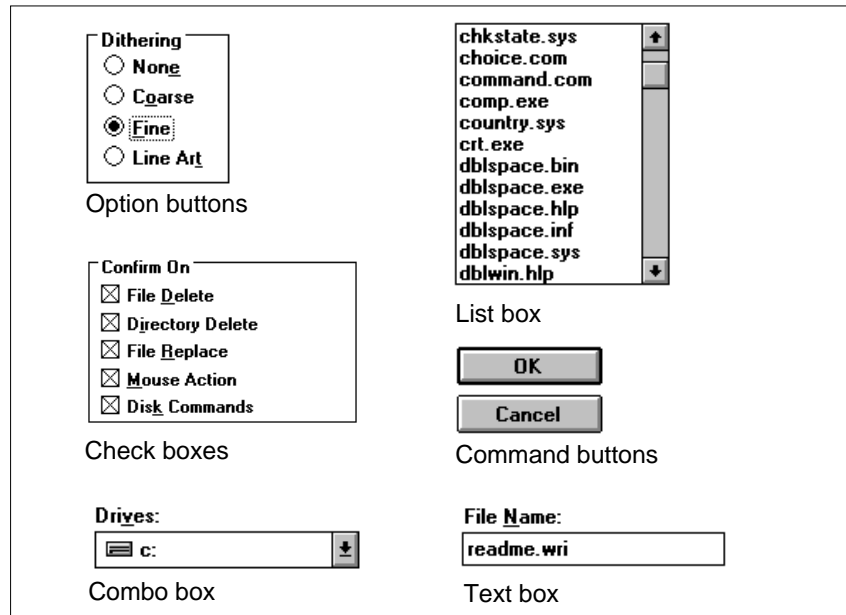
Fig. 3-6   Elements of a dialog box

Moving in dialog boxes

You generally have to move the cursor within a dialog box in order to select
the various options or to make entries. The currently selected input box is
highlighted optically or surrounded by a dotted frame.

You can change between the various input boxes in the following way using
the operator panel keyboard:

- Press the EDIT key to change between the individual input boxes,
  or, if the element has been assigned a HOTKEY (underlined letter),
  press the ALT key followed by the appropriate letter.

- Within a group of options you can jump from one option to another using
  the ARROW keys.

Option buttons   Option buttons are used to select mutually exclusive options, i.e. you can only
                       select one of the options. The selected option button is identified by a black
                       dot.
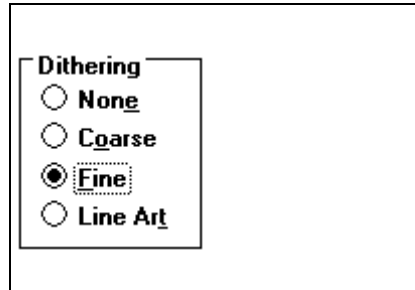
Fig. 3-7   Option buttons

You can select an option button using one of the following methods:

- Press the EDIT key to move to the desired group of options or press the
  corresponding HOTKEY.

- Use the ARROW keys to select the desired option.

Check boxes   A check box beside an option indicates that you can activate or deactivate the
                       option. In contrast to option buttons, any number of check boxes can be
                       activated simultaneously. Activated options are identified by an X.
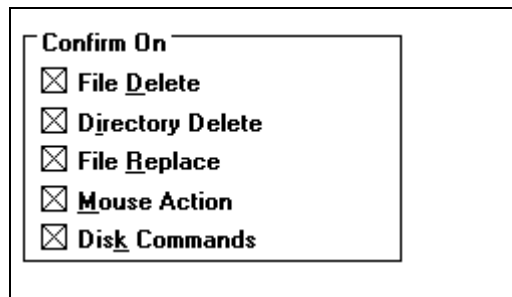
Fig. 3-8   Check boxes

A check box can be activated or deactivated in one of the following ways:

- Press the EDIT key to move to the desired group of options or press the
  corresponding HOTKEY.

- Use the ARROW keys to select the desired option.

- Use the SPACE key to select or deselect the box.

If the check box has been assigned a HOTKEY (underlined letter), you can use
the HOTKEY to activate the box.

Text boxes

Text boxes are used to enter information such as file names, etc. In empty text boxes, the insertion point appears at the far left-hand side. If the box already contains text, the text is selected and is automatically overwritten by newly entered text.

File Name:
readme.wri
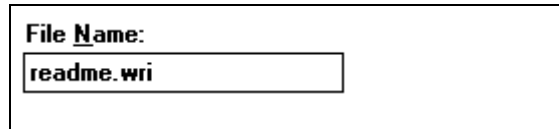
Fig. 3-9   Text box

You can access a text box in the following way:

- Press the EDIT key until the desired text box is selected.

If the text box has been assigned a HOTKEY (underlined letter), you can use the HOTKEY to select the box.

You can delete existing text in a text box using the CANCEL or CLEAR key.

List boxes

List boxes contain a list of selection options. If more options appear in the list than can be displayed in the box at one time, scroll bars are displayed at the edge of the list box.

chkstate.sys
choice.com
command.com
comp.exe
country.sys
crt.exe
dblspace.bin
dblspace.exe
dblspace.hlp
dblspace.inf
dblspace.sys
dblwin.hlp
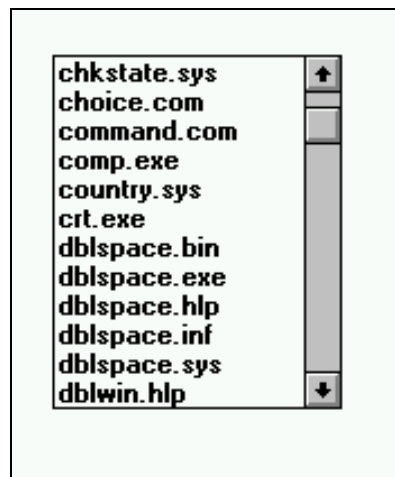
Fig. 3-10 List box

You can generally only select one item from the list.

- Select the desired item using the ARROW keys
- Press the INPUT key to confirm your selection.

You can search for an item in a list by entering the first letter.

Combo boxes
Combo boxes are single-line list boxes displaying the current selection.
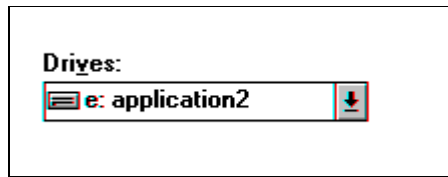
Fig. 3-11   Closed combo box

A combo box can be opened in the following way:

- Press the RIGHT ARROW key. The combo box is expanded to display a standard list box.
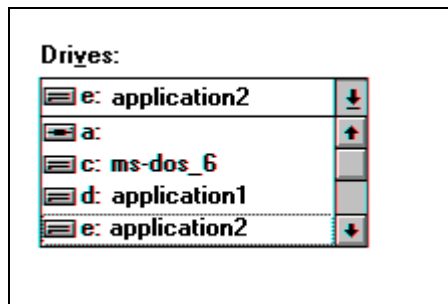
Fig. 3-12   Open combo box

The items in an open combo box are selected in the same way as in a standard list box.

Press the ACKNOWLEDGE ALARM key to close the opened combo box.

Command buttons
Command buttons are used to invoke an action, e.g. to execute or cancel a command. "OK" and "Cancel" are examples of command buttons. They generally appear on the right-hand side of a dialog box.
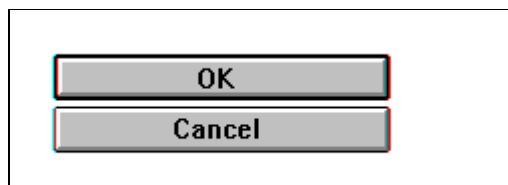
Fig. 3-13   Command buttons

Command buttons followed by omission dots (...) open another dialog box. A command button displayed with the >> character expands the active dialog box.

You select command buttons in one of the following ways:

- Press the EDIT key until the desired command button is selected.

- Press the INPUT key to execute the action.

As with the items in a menu bar, the SINUMERIK 840C displays the command buttons of a dialog box in the softkey bar. You can therefore select a command button by pressing the appropriate softkey.

Note

Buttons are only displayed as softkeys if the dialog box is **not system modal**. System modal dialog boxes are usually only displayed by the operating system. They must be filled in or acknowledged before the user can continue working in the application or switch to a different application.

System modal dialog boxes can only be operated on the MF2 keyboard on the keyboard input of the MMC module. They cannot be operated using the operator panel keyboard.

Closing dialog boxes

When you select a command button, the dialog box is closed and the command is executed.

You can also close a dialog box to cancel a command.

- Press the EDIT key until the "Cancel" button is selected and then press the INPUT key
  or

- Press the SOFTKEY assigned to the button.■

Machine

4

4.1 Selecting the Machine area

Press the DATA AREA key to activate **Area Switchover**.

Fig. 4-1   Area Switchover

When **Area Switchover** is selected:

- Press the MACHINE softkey to display the machine area

or press the MACHINE AREA key.

You can press the MACHINE AREA key at any time to activate the machine operation window (i.e. to bring it into the foreground).

4.2   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

| ... |
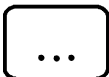
If you have, for example, six mode groups and one machine control panel, you must first assign the machine control panel to the desired mode group.

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.
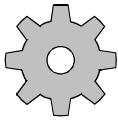
Machine manufacturer

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.

```
┌────────────────────────────────────────────┬──────────┐
│                                              │  12:48   │
│                                              ├──────────┤
│                                              │ ◈ ⤩ ◈  │
├──────────────────────┬───────────────────────┼──────────┤
│ AUTOMATIC            │                       │Mode grp.:1│
│         Program reset│                       │Channel  :1│
└──────────────────────┴───────────────────────┴──────────┘
```
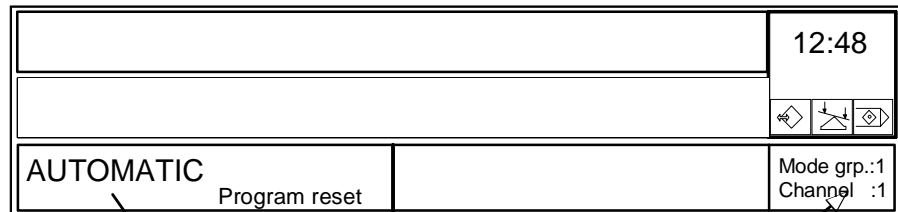
Display of the currently selected operating modes depends on the preset mode group
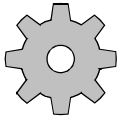
Machine manufacturer

Switchover of the machine control panel is configured by the machine manufacturer. Please read the machine manufacturer's specifications.

4.2.1 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

With certain error statuses, e.g. due to wrong parameterization of machine data, NC START is disabled in the respective mode group (= NC START disable).

Note

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

Example

For example, if channels 1 and 2 of mode group 1 are locked with respect to NC start and if channel 3 belongs to mode group 3, the NC start can still execute a program in channel 3.

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.

Table 4-1   Overview - Operating modes

From	To	AUTOMATIC			JOG		TEACH IN			MDA		
		Program reset	Running	Stop	Program reset	Stop	Program reset	Running	Stop	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					

4.2.2 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.

Table 4-2   Overview - Machine functions

Mode / Functions	AUTOMATIC			JOG		TEACH IN			MDA		
	Program reset	Running	Stop	Program reset	Stop	Program reset	Running	Stop	Program reset	Running	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			

SINUMERIK 840C, OEM Version for Windows (BA)

4.2.3   Status displays

4.2.3.1   Channel-independent status displays 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   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.
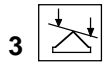
3   Axis moves as soon as a traversing command is active.
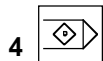
4   Program is being executed when in one channel the signal "Program running" is active.
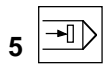
5   Block search, program interrupt if the signal "Program running" is not active in any channel and if a program interrupt is initiated by means of STOP, operating mode change, or M00/M01 active, or if block search has been triggered, or if the last block has already been executed in single block mode.

                                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

---
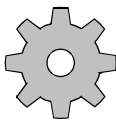
Machine manufacturer

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

4.2.3.2   Spindle utilization display

**Bar chart
Utilization**

This bar chart displays the percentage of spindle utilization in the current
operating mode. The display appears only if a digital drive is used.

Actual value	Override	Position	Setpoint value
F = 0.00	70%		F = 0.00 M
S1 = 0	85%	0.00	S1 = 0
Utilization (%)			

Note

Designation of the bar chart

- until SW 5.2:   Performance
- as from SW 5.4:   Utilization

4.2.4   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.

Fig. 4-2   Basic display in JOG mode

4.2.4.1   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

| +X | -X |

Press the direction keys to traverse the axes.

840C machine control-
panel for milling
machine

| X | Y |

| Z | … |
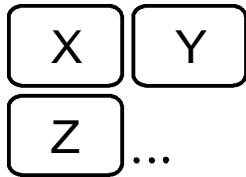
Select the axes.

| - | + |

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.
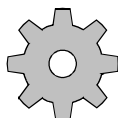
Machine manufacturer

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%.

4.2.4.2 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

Note

The machine manufacturer can change the reference point approach by means of the configuration. In this case, the following description does not apply.
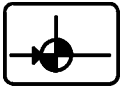
In JOG mode, "Approach reference point" may be selected by pressing the appropriate key.

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

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

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

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

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

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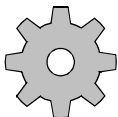
Machine manufacturer

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

4.2.4.3   User Agreement (Safety Integrated option)

The User Agreement is part of the Safety Integrated option. With the User
Agreement (via key-operated switch), an authorized person confirms that the
displayed current actual value of an axis corresponds to its actual position at
the machine. With the User Agreement active, the safely limited absolute
positions (SE) and / or safe cams (SN) become active and safe.
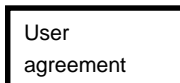
| User |
| agreement |

In the JOG mode, you can select the USER AGREEMENT screen via the
USER AGREEMENT softkey.

Preconditions

- Key-operated switch position 3

- The status "Axis referenced" has been reached

- The message 1340*/300950 "Axis is not safely referenced" is present.
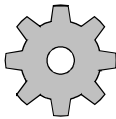
Machine manufacturer

With regard to referencing (incremental/absolute encoders), please refer to the
specifications of the machine tool manufacturer.

**Operating
sequence**

- Move the axis to a known position (e.g. visual mark).

- Compare this position with the NC actual value and the Safety Integrated
  actual value.

  – If the position corresponds to the actual values, the user can give his
    agreement with a toggle softkey "Axis in position" yes/no.

    The message 1340*/300950 "Axis is not safely referenced"
    disappears.

    The axis is now safely referenced and the safely limited absolute
    position (SE) and/or safe cams (SN) are active and safe.

  – If the position does not correspond to the actual values, the user
    cannot give his agreement.

    The axis settings must then be checked again.

4.2.4.4 Increment mode - selection

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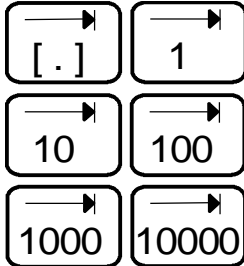
Before traversing the axes using the direction keys, select the appropriate incremental weighting on the associated machine control panel.
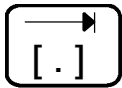
This key (INC variable) is used for traversing an increment previously set:
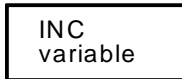
For setting the variable increment, activate this softkey and enter the value you require in the input field.

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.
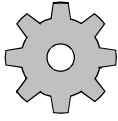
Machine manufacturer

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

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.
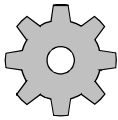
Machine manufacturer

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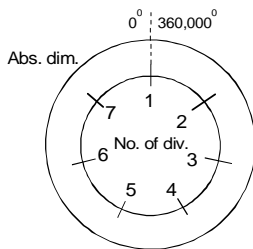
Example of "Indexed traversing" with a rotary axis:

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.

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.

4.2.4.5   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 or TEACH-IN 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.
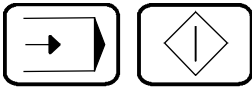
After the contour has been reached, select AUTOMATIC mode and press the NC START key.

Notes
- 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!

4.2.4.6   Scratching

If, for example, a tool breaks 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.
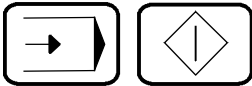
Select AUTOMATIC mode and press the NC START key.

The block end point is approached and program execution continued.
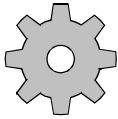
Machine manufacturer

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!

4.2.4.7 Finish thread (option)

> **>**
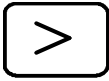
Press the ETC key to expand the softkey bar.

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.
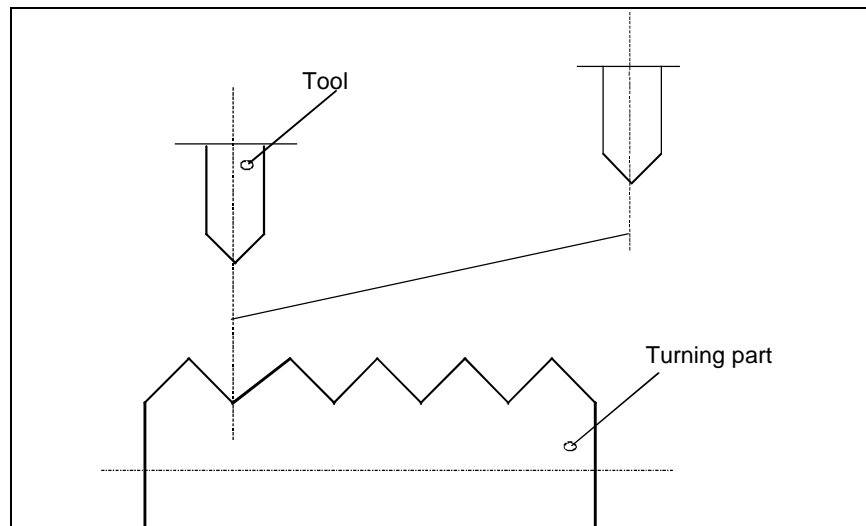
Fig. 4-3   Finish thread

Note

The offset angle is not retained after POWER ON.

Maximum value: 359.99999 degrees
Resolution: 10^{-5} degrees, without sign

4.2.5   TEACH IN mode

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

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

- Edit mode and
- Teach block by block.

Edit mode

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

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

Block-by-block teach-in

Block-by-block teach-in 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.

```
┌─────────────────────────────────────────────────────────────────────────────┐
│                                                                               │
│       TEACH  IN                                                               │
│                                                                               │
│       ┌──────────────────────────────┐                                        │
│       │  ● Automatic                  │                                        │
│       │    - NC start enable          │                                        │
│       │    - Set-up disable           │                                        │
│       │    - Program selection        │                                        │
│       └──────────────────────────────┘                                        │
│                                                                               │
│       ┌──────────────────────────────┐    ┌────────────────────────────────┐ │
│       │  ● Edit mode                  │    │  ● Teach block by block        │ │
│       │    - NC start disable         │    │    - NC start disable          │ │
│       │    - Set-up disable           │    │    - Set-up disable            │ │
│       │    - Teach-in possible        │    │    - Teach-in only possible    │ │
│       │      without restrictions     │    │      in the teach block        │ │
│       │    - Set block structure      │    │    - Teach-in not possible     │ │
│       │                               │    │      in the predecoded area    │ │
│       │                               │    │    - Set block structure       │ │
│       │                Accept blocks  │    └────────────────────────────────┘ │
│       │                               │                                       │
│       Test                                                                    │
│       ┌──────────────────────────────┐    ● Other functions                   │
│       │  ● Teach block by block       │      - Modify program                  │
│       │    - NC start disable         │      - Block search                    │
│       │    - Set-up disable           │      - Single block mode               │
│       └──────────────────────────────┘                                        │
│                                                                               │
└─────────────────────────────────────────────────────────────────────────────┘
```

Fig. 4-4   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.
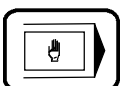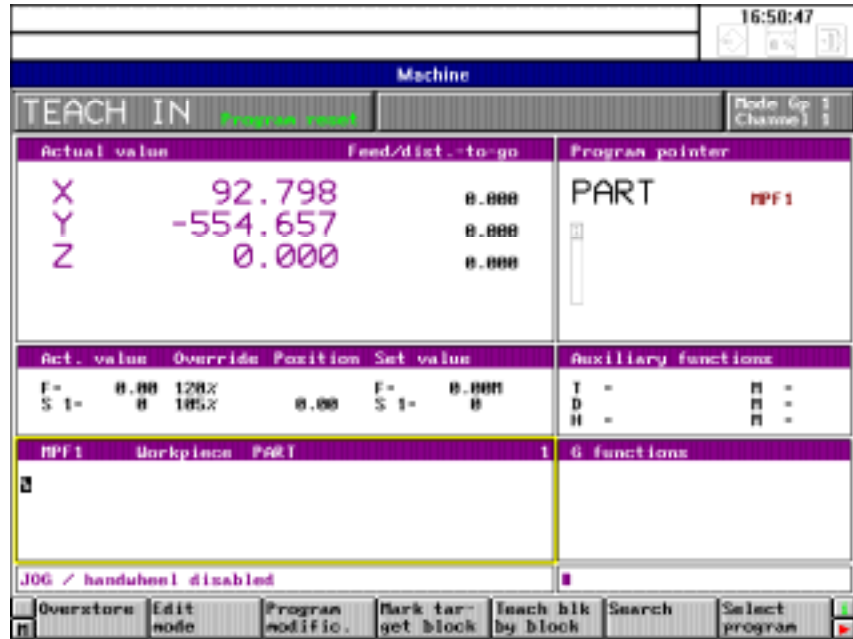
Fig. 4-5   TEACH IN

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.

Note

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.).

---

i

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

Continue

You scroll 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.

4.2.5.1   Set breakpoints

Breakpoint   The softkey function MARK TARGET BLOCK allows you to define a
                              breakpoint in an existing program.
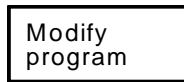
Modify   Select the "Modify program" menu.
program

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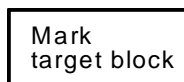
Mark   press the MARK TARGET BLOCK softkey. The target block is highlighted in
target block   colour.
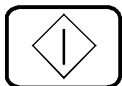
[◇]   When you press NC START, the block containing the breakpoint is displayed.
                              The started program executes up to and including the selected block with the
                              breakpoint.

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

Fig. 4-6   Block with breakpoint

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

**Program step
Decode single
block**

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

Note

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

Note

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).

4.2.5.2 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.

Note

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

Notes

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 window.

4.2.5.3   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:

Fig. 4-7   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.

SINUMERIK 840C, OEM Version for Windows (BA)
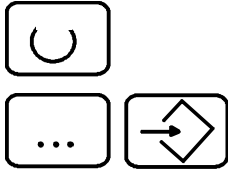
Use this key to select the toggle field options.

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	G 53
Workpiece dimensions	none
Axes	traversed

Fig. 4-8   Block structure/Settings

Block number	– Generated NC blocks are numbered
Next block number	– Generated TEACH block is No. N10
Block number difference	– Block spacing (N15, N20)
Settable zero offset	- G53 is selected
Workpiece dimensions	– No switchover in inch/metric measurement system
Axes	– 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).
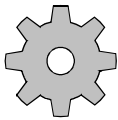
Machine manufacturer

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.

4.2.5.4 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
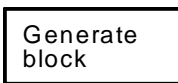
| Generate block |

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

Fig. 4-9   Axis selection screen

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

Fig. 4-10 Axis selection screen

The axis positions of axes, Y, Z and A are entered in the NC block with "Accept position", regardless of whether or not 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.

4.2.5.5 Accepting axis positions

Milling machine

| X | Y | .. | Select the appropriate axes and

| + | - | traverse these according to the desired position in positive or negative direction.

Turning machine
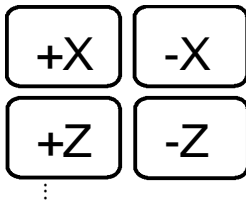
| +X | -X |
| +Z | -Z | Traverse the axis according to the desired position.
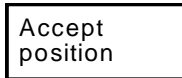
⋮

| 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").

4.2.5.6 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.
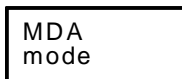
| MDA mode | Press the MDA MODE softkey. The MDA window is highlighted by a yellow frame.
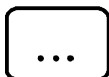
| ... | Use the keyboard to enter blocks in this window.

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
```

Fig. 4-11 Selecting an MDA block

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

Note

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

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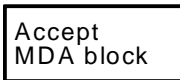
Accept
MDA block

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  L_F
N15  M3  S1025  M7  L_F
```

Fig. 4-12 Accept MDA block

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
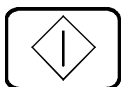
- The functionality of the MDA mode in edit mode/teach-in is identical to the MDA mode. The CLEAR action is ignored in the MDA mode/TEACH IN.
- If an MDA program has been executed, it can only be edited after the last existing MDA block.

- 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.

Note

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.

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.

4.2.5.7 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 "Edit error, no edit allowance" is displayed).

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)

Note

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.

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.
If 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.
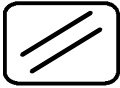
You can only switch between "Edit mode" and "Teach-in block by block" after pressing RESET.

4.2.5.8   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.

The block number and block number difference settings are also valid in the NCK editor.
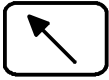
Switch to the "Block structure/settings" selection screen.

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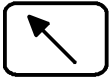
Switch to the "Axis selection" screen and enter the desired settings. You will find a detailed description of these selection screens in the previous sections.

Press RECALL to exit the selection screens.
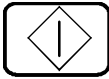
Now start the part program by pressing NC START. The program executes until the selected breakpoint.

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                                     │
│                                                    │
└──────────────────────────────────────────────────┘
```

Fig. 4-13 Breakpoint block

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
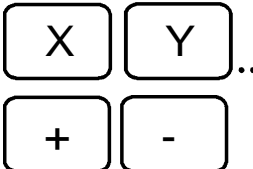
| X | Y | .. | Select the appropriate axes and |

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

Turning machines
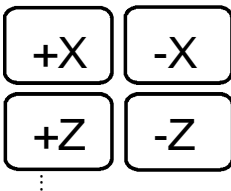
| +X | -X | Traverse each axis to the required position. |

| +Z | -Z |

| 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. |

       6FC5198-6AA60-0BP0
                           SINUMERIK 840C, OEM Version for Windows (BA)

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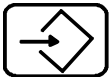
You can insert a block **before** the teach block.

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

```

Fig. 4-14 Insert block

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).
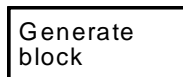
Generate block

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= -25600  Y= 68.300  L_F
  N20  G0  Y0  L_F
```

Fig. 4-15   Generate block

Notes

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
```

Fig. 4-16   Settings for block structure

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, with the described possibilities of program interruption.

Note

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

Notes

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 in JOG mode. 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 overstore and block search functions in the sections of the same name.

REPOS in teach-in mode

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. This is the prerequisite for the operating sequence in "teaching-in block-by-block". If DSB is not enabled, you can neither edit nor reposition with the Repos offsets.

4.2.5.9   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.
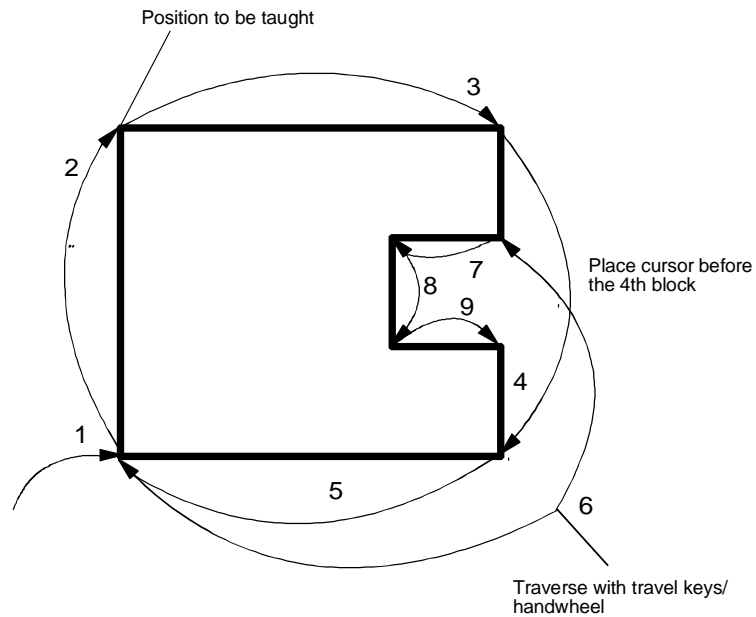
Fig. 4-17   Teach-in in edit mode

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

Example for teaching-in block by block

First teach-in 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.
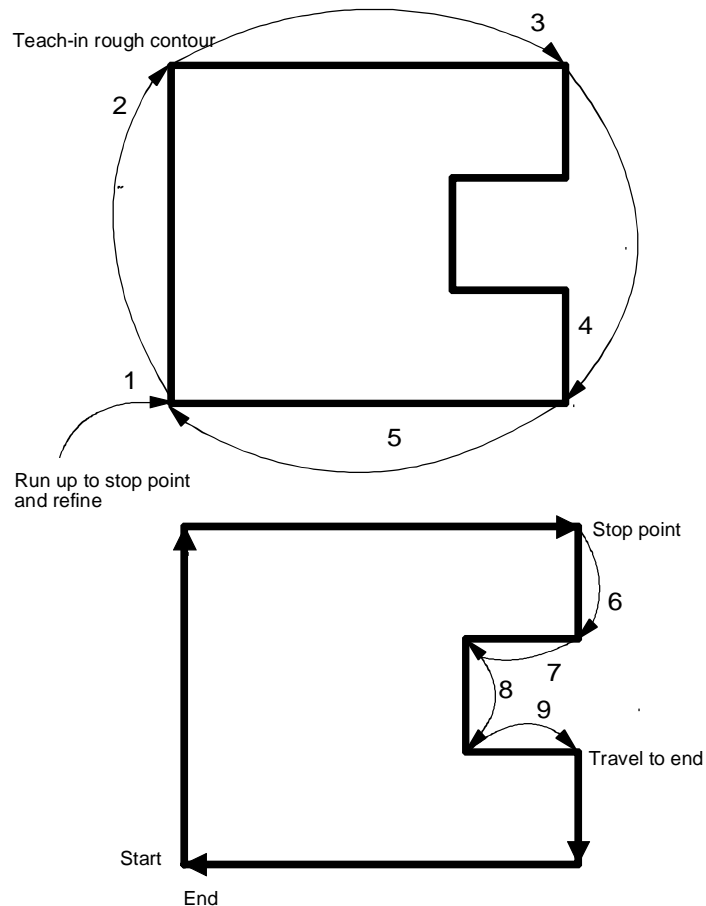
Fig. 4-18 Block by block teach-in 1

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.

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-in position C and D.
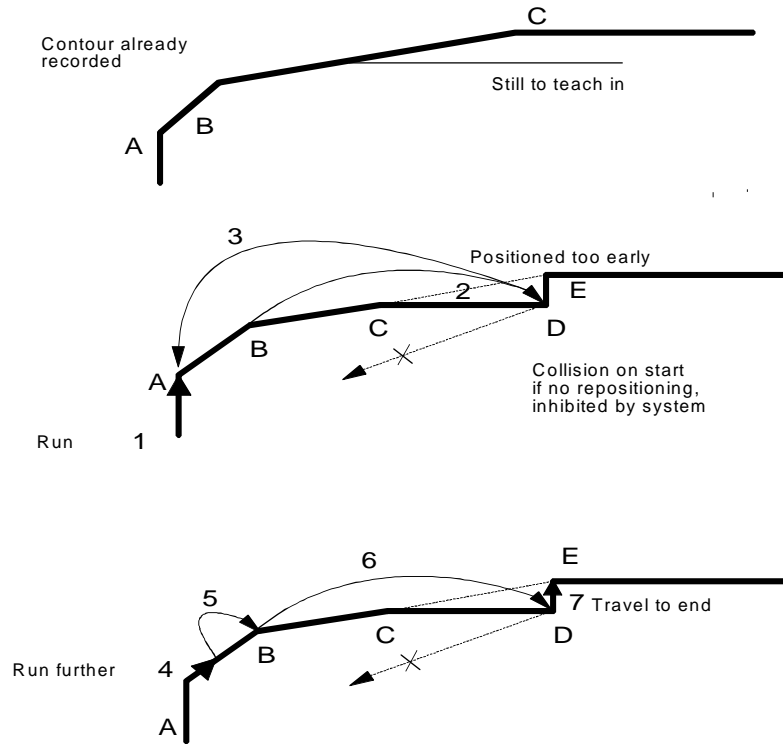
Fig. 4-19 Block by block teach-in 2

4.2.6 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.
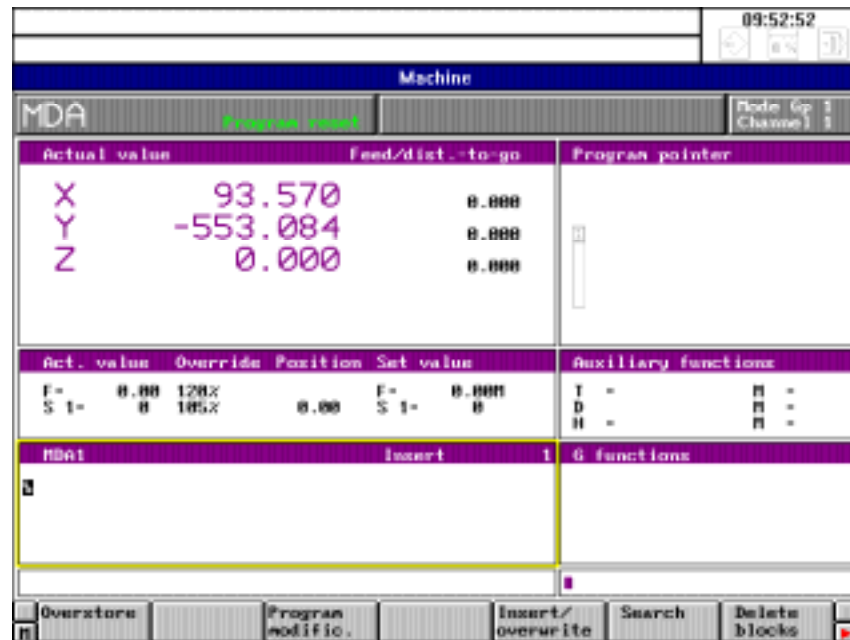
Fig. 4-20 Basic display in MDA mode

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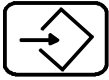
Press the INPUT key at the end of each block. An end of block character (L_F) is generated automatically and the cursor moves to the next line.

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

- Pressing FEED STOP

The program is restarted by pressing

- NC START

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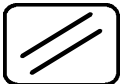
If you press the RESET key, execution is terminated. A RESET is also generated by changing the mode, i.e. execution is interrupted. Pressing NC START starts the program from the beginning. If clear (CLR) is active, RESET deletes the MDA blocks.

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  │
└─────────────┘

┌─────────────┐
│  Program    │
│  modific.   │
└─────────────┘
```

The machine functions

- Overstore

- Program modification

are described in the "Additional machine functions" section.

Note

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

The contents of the MDA program can be copied into a part program using the data selector (see next section).

4.2.6.1 Copying MDA programs

First select the area **Programming Edit NC**.
A list appears with the program types MPF, SPF and MDA.
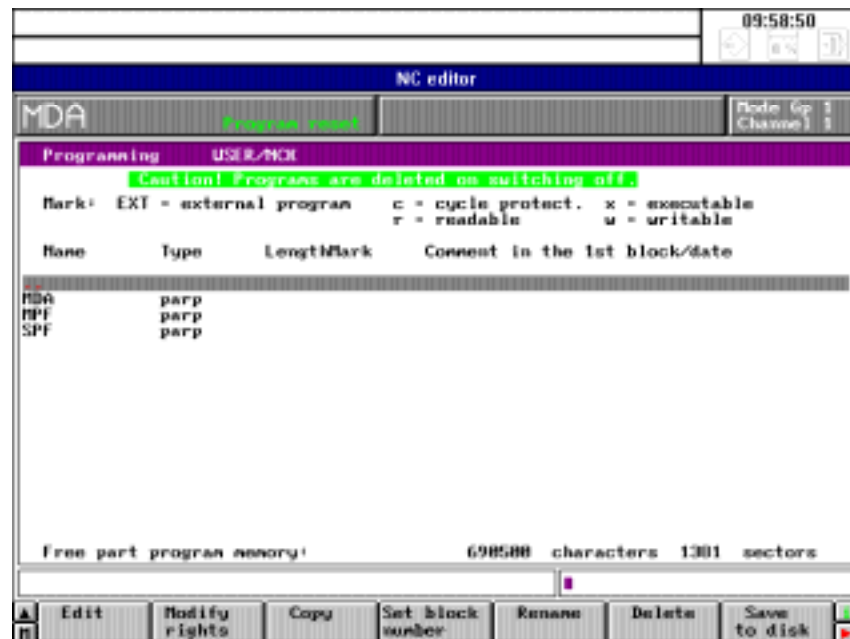
Fig. 4-21 List of program types in the NC editor
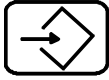
Position the selection cursor on MDA and press the input key.

A list of possible MDA programs is displayed.

Copy

Now select the MDA program that you wish to copy and press the softkey COPY.
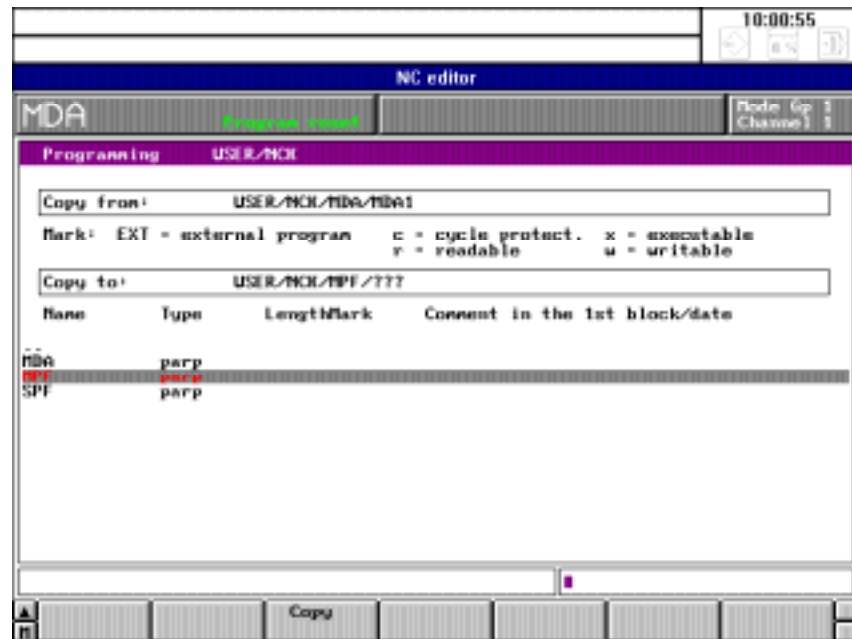
Fig. 4-22 Copying MDA programs

Now place the cursor on the directory "MPF" (for the purposes of this example). You can see how the selected directory is updated.

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

The NC blocks of MDA 1 now become main program MPF 1234.

Note: COPY is possible only within the directory NCK, e.g. from MDA to MPF.
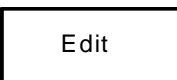
Edit

To save the data on disk, it is best to use (in the NC editor area) the softkeys EDIT/SAVE.
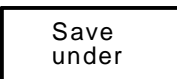
Save
under

Saving of the currently selected program
with the SK: PRGORAM CORRECTION/SAVE ON DISK/
or   PROGRAM CORRECTION/SAVE UNDER
is described in section "Saving programs".

4.2.7   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.
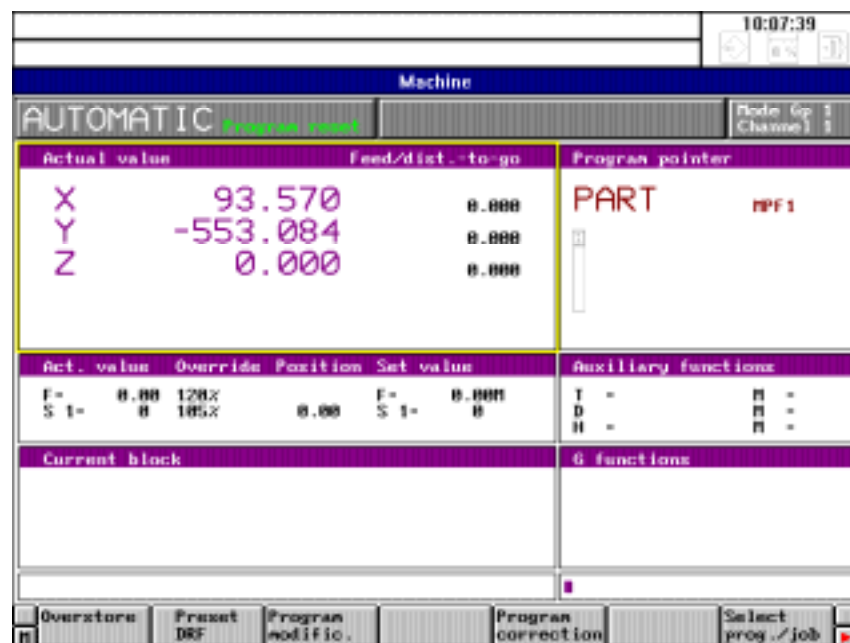
Fig. 4-23 Basic display in AUTOMATIC mode

6FC5198-6AA60-0BP0
SINUMERIK 840C, OEM Version for Windows (BA)

4.2.7.1 Workpiece and program selection with the data selector

The data selector can be used to access the workpiece data on the hard disk or data on another computer in the network from the NCK area **Machine**.

Operating sequence

| Select prog./job |

Press the softkey SELECT PROG./JOB in the automatic basic display. The "data selector" appears in the basic display for Automatic.

Note

When selected first, the data selector displays automatically the workpiece directory LOCAL.

If you want to deselect another workpiece directory, select the root directory first.

For this, position the selection cursor by means of the ARROW keys on the entry (**..**)
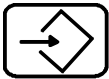
Press the INPUT key. The workpiece directories in the root of the data selector are displayed.
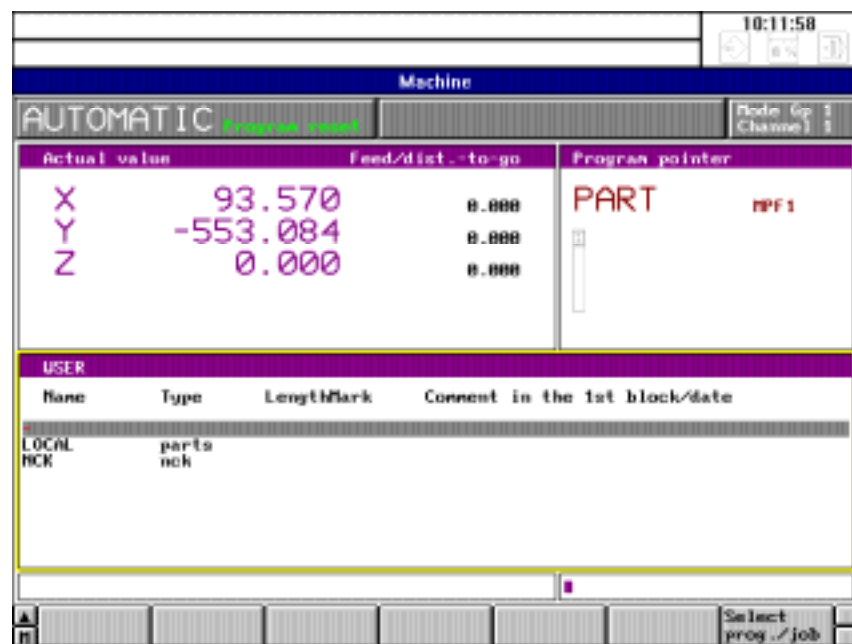
Fig. 4-24 Data selector

The following directories are displayed in the data selector:

• The root directory is USER. It always contains the subdirectories LOCAL and NCK.

- The directory LOCAL designates the Windows directory on the local hard disk that usually contains all the workpieces.

- Directory NCK is a virtual directory with which the user can return to the part program overview of the NCK.
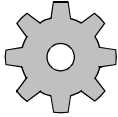
Machine manufacturer

The machine manufacturer can configure additional entries in the data selector. These additional entries might be directories on the local hard disk (also directories of the FlexOS file system), diskette drives and/or network drives.

Please consult the documentation supplied by the machine manufacturer for additional information.

Precondition

The workpieces are stored in the directory LOCAL. A job list (load list) exists for every workpiece.
Please refer to the Section "Programming" for more detailed information about the job list.

Operating sequence

Place the cursor on the directory LOCAL.
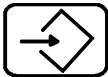
Now press the INPUT key. The workpieces in the directory LOCAL are listed.
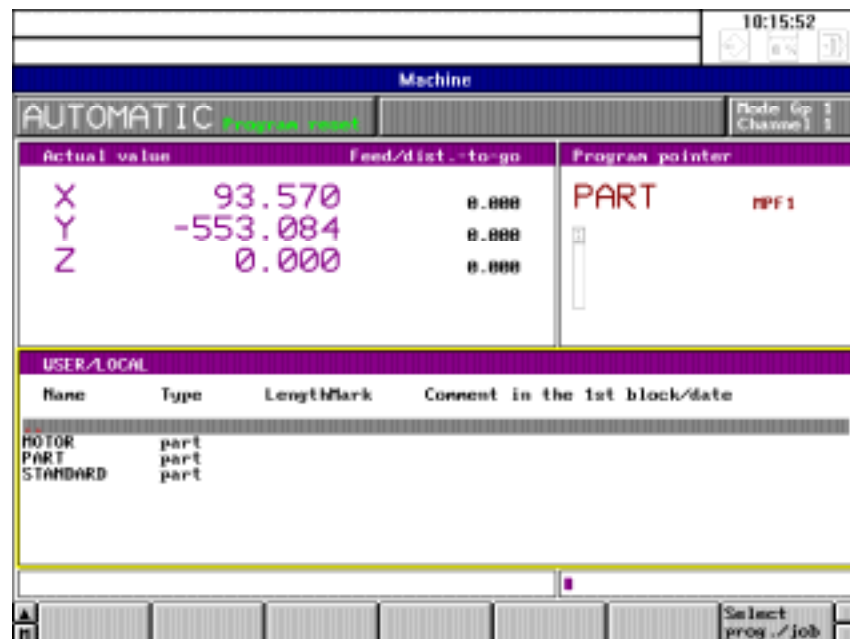
Fig. 4-25 Workpiece selection 1

Now position the cursor on the required workpiece.

Select prog./job

Now press the softkey SELECT PROG./JOB. The following dialog text is displayed: "Load workpiece?"

If you press the softkey SELECT PROG./JOB again, all the workpiece data as listed in the job list are loaded from the hard disk into the NCK memory.

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

Fig. 4-26 Workpiece selection 2

If a workpiece has been selected and a job list exists, the data manager and current processing is displayed during data transfer.

Stop work-piece sel.

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

Important

If workpiece data transfer is aborted, incomplete program parts might be left in the NCK memory.

Notes

- If a workpiece does not have a job list or the standard job list and if several or no main programs exist, no program is selected for processing and no workpiece name is displayed in the "program pointer".

- Several workpieces, even those from other directories (not NCK!), can be loaded into the NCK memory using the function SELECT WORKPIECE.

- Any files already existing in the NCK memory are overwritten without warning.

- By selecting a workpiece and pressing the INPUT key you can branch into the directory of the workpiece. Then, all the NCK files (including the job

list) are displayed in the data selector. However, individual files cannot be transferred to the NCK memory.

Dialog messages and alarms

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

Table 4-3   Dialog messages during loading

Error	Dialog message
No directory has been called.	Illegal directory
A directory but no workpiece has been selected.	Please enter correct name
The softkey has been 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

Table 4-4   Alarm messages during loading

Error	Alarm message
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:

The instructions in the job list are processed one after the other during workpiece transfer. If one of the instructions is incorrect (e.g. syntax error), processing of the job list is aborted and a message box is displayed.
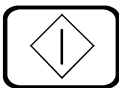
Workpiece processing is triggered with the NC START key and the selected program is started.
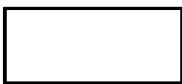
The machine functions can be selected by pressing the relevant softkeys.

The individual machine functions are described in the subsection "Additional machine functions".

Selecting part programs

The normal procedure for selecting a workpiece is to load several main programs and subroutines from the hard disk into the NCK memory and selecting max. one part program per channel.

With the data selector, however, it is also possible to select a specific program for processing from the NCK memory and put it in the "program pointer".

Operating sequence

| Select prog./job |

Press the softkey SELECT PROG./JOB. Use the cursor keys to select the NCK entry in the open data selector.
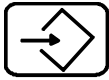
When you press the INPUT key the program types available in the NCK memory are displayed. The following are available:

- ..
- MDA
- MPF
- SPF
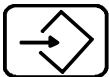
Now position the selection cursor on the program type you wish to select (e.g. MPF) and press the INPUT key. This action lists all the main programs in the NCK memory.
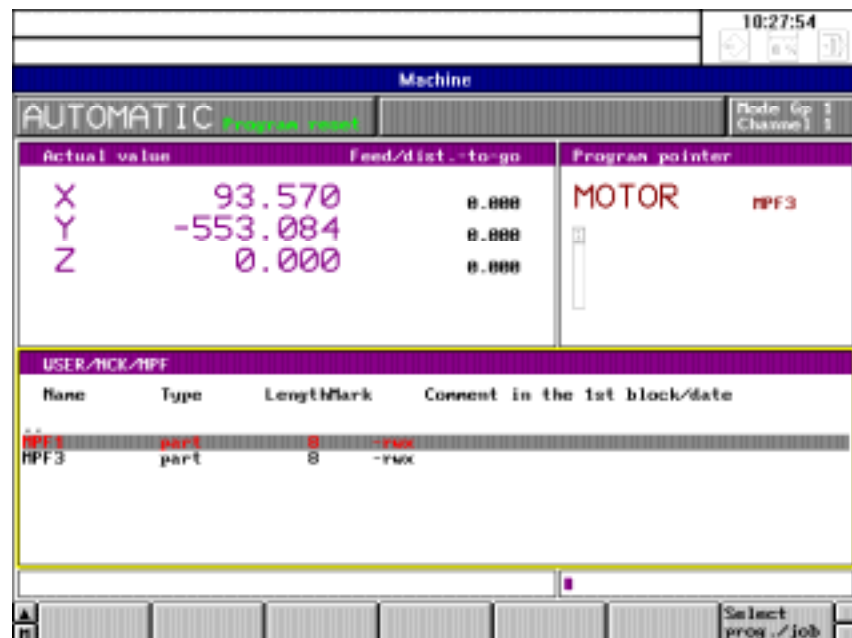
Fig. 4-27   Select program

| Select prog./job |

Now select the program you require with the CURSOR keys and press the softkey SELECT PROG./JOB.

The program is put in the "program pointer" and can be processed by pressing NC START.

4.2.7.2   SELECT PROGRAM function

In addition to program selection with the DATA SELECTOR, it is also possible to select a program (MPF, SPF) in the conventional way by entering %... and L ... as is the case 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
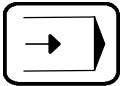
Press this key to select AUTOMATIC mode.

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

Procedure

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

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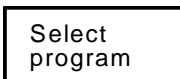
Select program

Press the SELECT PROGRAM softkey. The selected program is accepted by the program pointer.

Caution

If a program is entered as % ... or L ..., it appears in the program pointer as MPF ... or SPF ....

It is also possible to select programs that are not in the part program memory of the NCK. The error message "Program not in memory" then appears if NC START is pressed.

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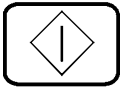
To start the part program, press the NC START key.

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 HOLD ("FST" message) you can stop the axis movements. The NC program is not interrupted. With FEED START you enable the axis movements again.
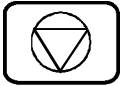
NC STOP

NC START

FEED START

FEED HOLD

Notes

- While the part program is being executed, a "Program running" display appears in the mode field.

- If, during program execution, FEED HOLD 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 the RESET 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

A program can be edited at the breakpoints. The section "Program correction" describes which type of interruption allows editing without RESET.

Note

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

Note

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.

Remedy: Use RESET

4.3   Additional machine functions

4.3.1   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.
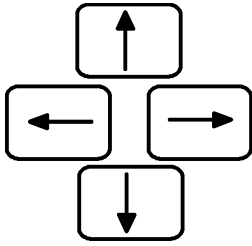
Now, using the cursor keys, set the reverse video marking on a desired input field.

Overstore

S ■ =   M   =
T   =
D
H   =
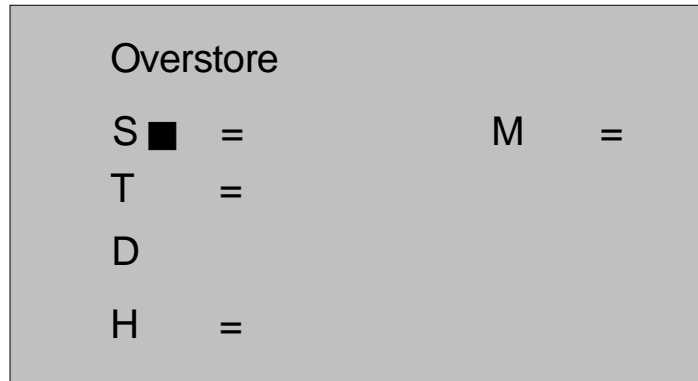
Fig. 4-28 "Overstore" input window

Enter the correction values via the keyboard and conclude each value by pressing the input key.
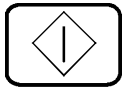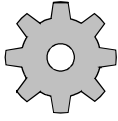
Press the NC START key when all corrections have been carried out.

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).

6FC5198-6AA60-0BP0
SINUMERIK 840C, OEM Version for Windows (BA)

4.3.2   Extended overstore

Machine manufacturer

The softkey function "Extended overstore" must be configured by the machine 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.
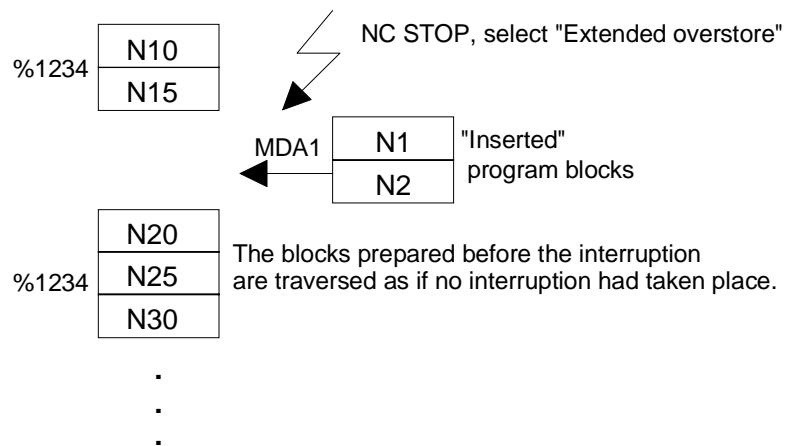
Preconditions

- 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.
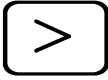
**Operating
sequence**

> `>`

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

```

Fig. 4-29 "MDA1" window

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

Note

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.

`/\`

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).

- The following table shows when certain actions are possible:

Table 4-5   Overview - Extended overstore

Mode	Channel state	Message
AUTOMATIC, JOG, TEACH IN	Running	No extended overstore possible
	RESET	Extended overstore
	NC STOP	Extended overstore
	Program stop	Extended 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 extended overstore possible
MDA mode	Of no significance	No extended 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.

4.3.3 Altering F and S values on-line

The feedrate value F (axis-specific 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**
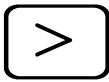
> Extend the softkey bar with the ETC key.

F/S
on-line
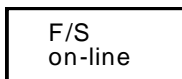
Press the softkey F/S on-line to call up the following display:
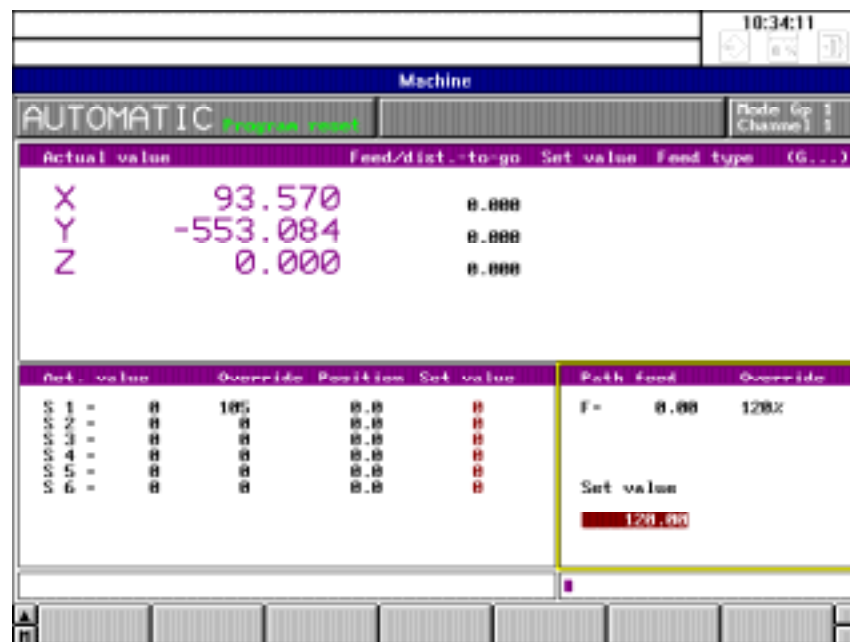
Fig. 4-30 Changing F/S values

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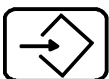
Transfer the entered value into the selected screen form with the INPUT key.

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 feedrrate for block-related simultaneous axes is active only in 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).

4.3.4 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 Press the PROGRAM MODIFICATION softkey. The "Program modification" screenform appears on the CRT.

Program modific.
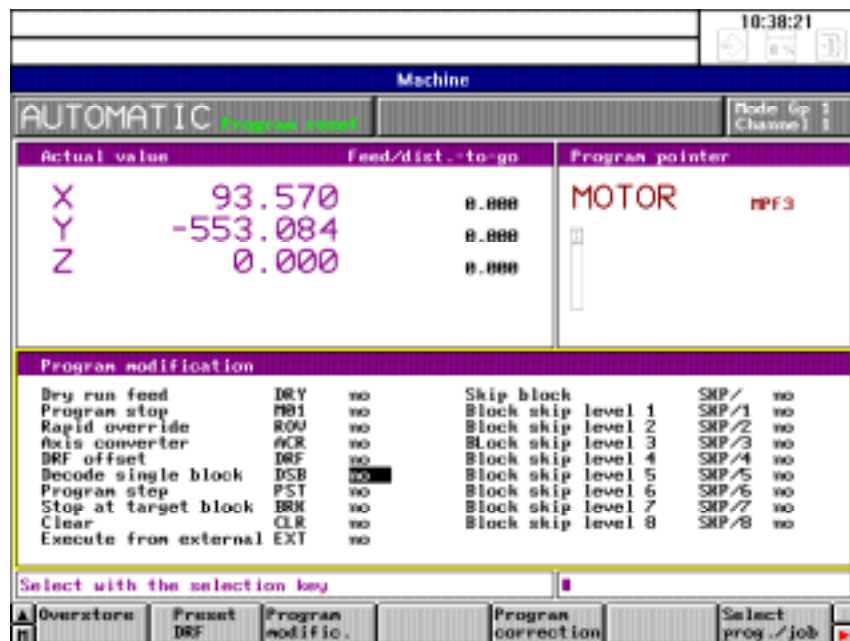
Fig. 4-31 Program modification
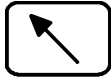
Use the window switchover key to switch to the "Program modification" screenform. The active window is marked (by a yellow frame).
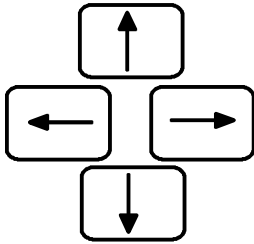
With the cursor keys you select the program modification function. A cursor marks the current position.

Each function can be selected or deselected via an associated toggle field by means of yes/no.

All active functions are displayed in the status field with the corresponding abbreviation (SKP, DRY, M01, ROV, DSB, DRF, ACR, PST, BRK, CLR or EXT). 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.

4.3.4.1 Description of individual 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 (GO1, 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.

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 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 (Procedure Step)
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

- SET STOP POINT 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 (Clear)
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.
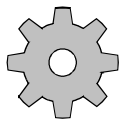
Machine manufacturer

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.

EXT

Execution from external; Display: EXT (Option)
This function makes it possible to execute an NC program from the hard disk or via computer link (see also the job list description).

NCV

Start of program/End cycles OFF: Display of NCY (no cycle)
Automatic processing of program start and end cycles can thus be interrupted.
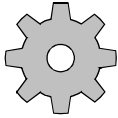
> **Machine manufacturer**
>
> The machine manufacturer determines whether program start and end cycles are used and (if this is the case) which subprograms are used.

SAV

Automatic saving; Display of SAV in the NCK editor
If this function is active, saving in the appropriate workpiece is started automatically after edtiting of part programs in the NCK (e.g. with SK: PROGRAM/NC editor or machine/program correction).

If no workpiece is known (new program), the modified program is stored in the directory LOCAL/STANDARD.
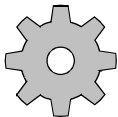
> **Machine manufacturer**
>
> Via technology MD, the machine manufacturer can provide the workpiece NCKTMP instead of STANDARD for this case.

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

> **Note**
>
> With the setting date 5001.2=1 you can activate permanently the function SAV (even after power on).

Restrictions

Automatic saving is not active in the TEACH IN operating mode. Saving is performed only after program modification and leaving the efitor or area switchover.

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.

They can also be configured by the machine manufacturer via the PLC program. See also the machine manufacturer's documentation.

Predec. blocks   The currently necessary number of block buffers is displayed. A maximum of
                        23 block buffers is provided as standard for predecoding.

maximum   The maxium number of block buffers can be further restricted.
                        Application: Fast decoding in single block mode.

4.3.4.2 Single block/decoding single block

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

**Operating
sequence**
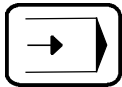
                        The SINGLE BLOCK function is selected when you press the appropriate key
                        on the machine control panel.

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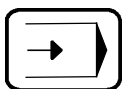
                        Pressing the SINGLE BLOCK key on the machine control panel a second time,
                        deselects the function.

                        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.

Table 4-6   How SINGLE BLOCK and DEC SINGLE BLOCK work

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 circuit / CRC .3)	X[1])	X[1])
Blocks generated within the control / CRC... 3)	[2])	X[1])
Threading blocks without dry run feedrate	X[1])	X[1])

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.

- 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".

Note

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.

4.3.5 Block search

Preconditions

- The machine area has been selected.

- AUTOMATIC mode has been selected.

- The control is in the reset state.

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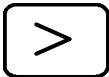
> On pressing the extension key (ETC) in the AUTOMATIC or TEACH IN basic displays, the BLOCK SEARCH softkey is displayed.

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

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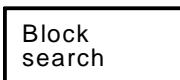
Block search

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 block search are not displayed 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			

Fig. 4-32 Block search window

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 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.

**Search with
calculation**

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.

Note

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**

| 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**

| Search from
main block |

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

| ◇ |

After NC START the program is started and executed.

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.

This main block (or subblock) is only found at the main program level.

Everything else is the same as for SEARCH WITH CALCULATION.
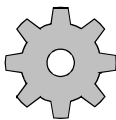
Machine manufacturer

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.

Note

With block search run together with TRANSMIT, please ensure that before
starting the search run and before NC START both the tool tip and the
toolholder are located behind the turning center (positive x direction);
otherwise, TRANSMIT processing may start in front of the turning center and
will be interrupted with the alarm "Speed set value alarm limit".

This problem arises only with angle head cutter.

4.3.6   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.
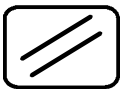
It is only possible to edit at the breakpoint in these cases if the program
execution has been aborted by pressing the RESET key. This resets the entire
decoding. When you press NC START, execution begins from the start of the
program.

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

Correct program

Press the CORRECT PROGRAM softkey.

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$_F$

Fig. 4-33 Program correction display

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 detected only during processing (e.g. error in tool radius compensation), the defective block is not marked; it is possible to detect this via the block number in the alarm.

Problem, remedy

Neither is the block marked, nor is the block number displayed in the alarm. If necessary, complete the relevant program section with block numbers and start the program again.

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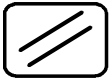
After the RESET key is pressed, you can edit anywhere in the entire part program. The selection of the correction block is cancelled.

Note

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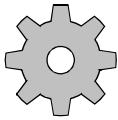
Machine manufacturer

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

Notes   The CORRECT PROGRAM 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).

4.3.7   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

Correct Program

Press the CORRECT PROGRAM 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.
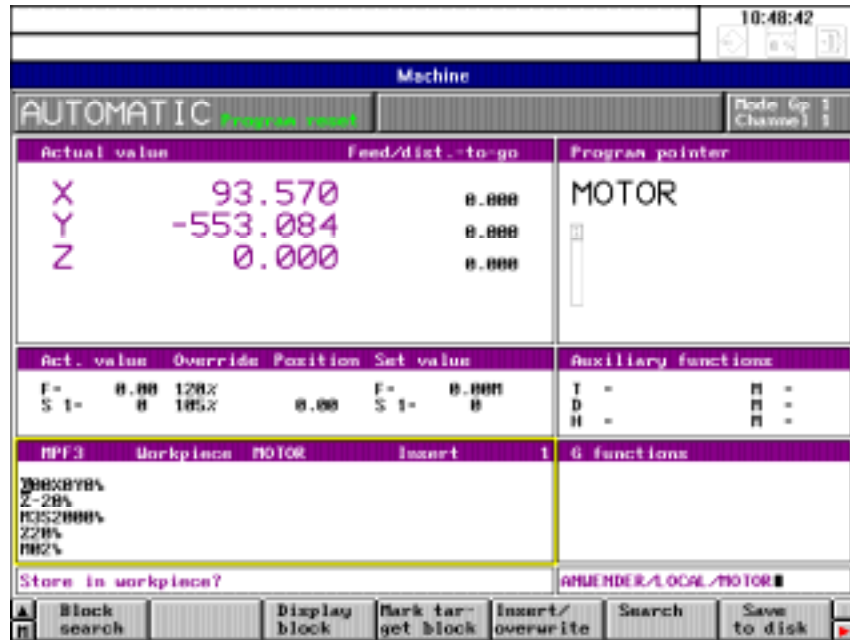
Fig. 4-34 Save 1

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.

Note

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

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.

SINUMERIK 840C, OEM Version for Windows (BA)
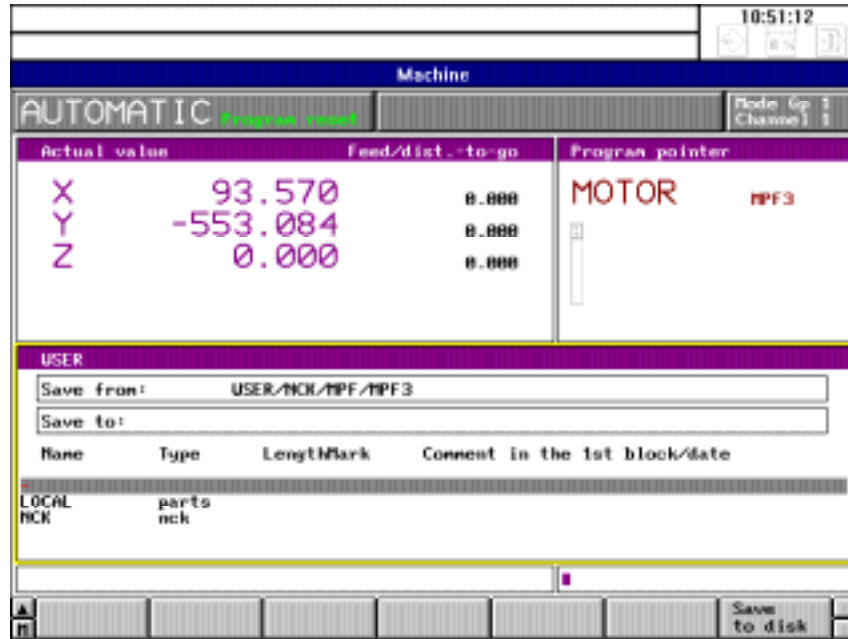
Fig. 4-35   Save 2

The workpiece can now be selected with the cursor in the 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.

Note

Programs can be saved in all displayed workpieces (including network drives and/or diskette drives).

4.3.8   PRESET (Set actual value) / DRF

4.3.8.1   PRESET - Offset

In the PRESET machine function, you can offset the control zero to any point in the machine coordinate system.
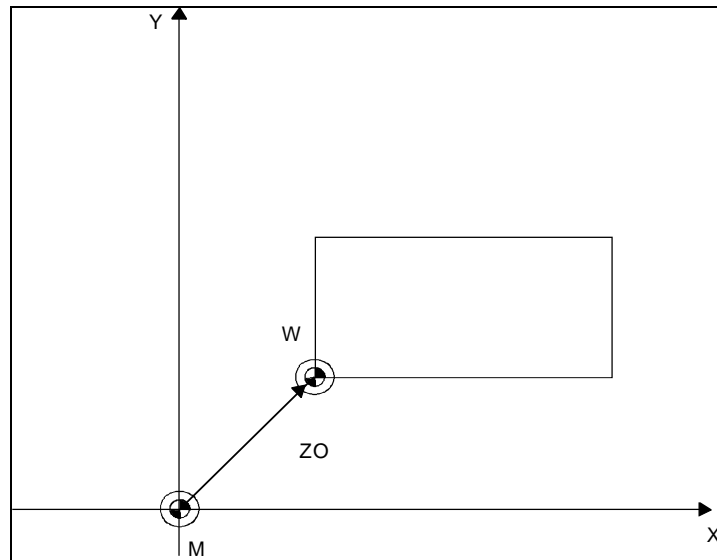
Fig. 4-36 Machine coordinate system **before** PRESET/DRF offset
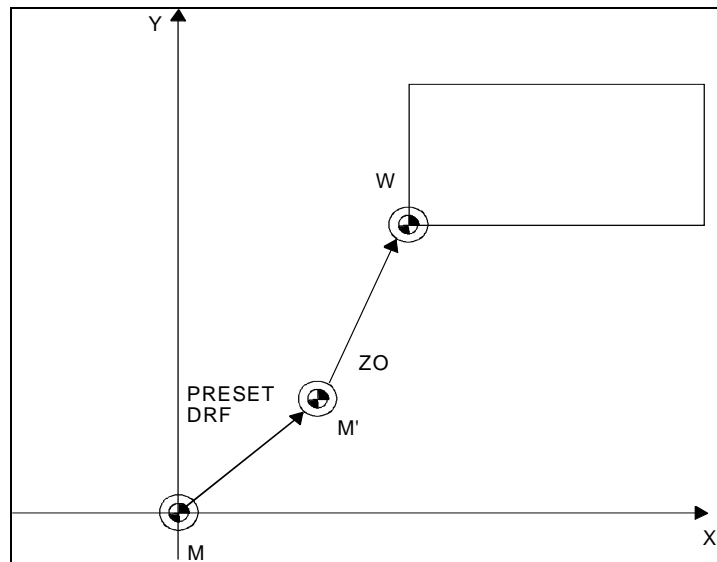
Fig. 4-37 Machine coordinate system **after** PRESET/DRF offset

M ...   Machine Zero
W ...   Workpiece Zero
ZO ...   Zero Offset
M'...   Control Zero

Note

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.

Note

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

Before the PRESET offset, enter the
- identifying number, the
- correction number, and the
- correction direction.

Switch to the upper window, if necessary.

	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.)			

Fig. 4-38   Input window PRESET/DRF
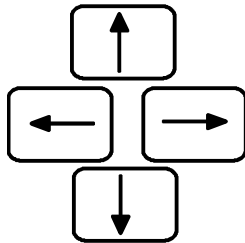
Position the cursor to the input field for the ID number.

Enter the appropriate value via the keyboard.
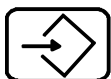
Confirm your input by pressing the INPUT key
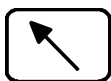
Switch to the lower input window.
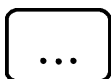
Enter the value for the correction number D... via the keyboard.
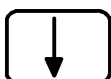
Position the cursor on the correction direction input field.

Enter the appropriate value (0 = positive, 1 = negative) via the keyboard.
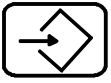
Confirm your input by pressing the INPUT key.

**Without including
the tool offset**
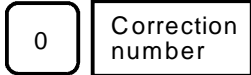
Enter the value 0 and press the CORRECTION
NUMBER softkey.

After having selected this data, enter the actual PRESET offset.
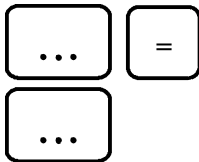
Enter the axis name and the new actual value for the axis.

Conclude the input by pressing the STORE ACTUAL VALUE softkey.
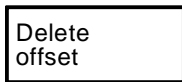
The DELETE OFFSET key is used to clear all PRESET offsets of the selected
mode group.
The offset displayed includes the input value and the included tool correction.

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

The PRESET offset remains stored:

- After "End of program"

- After RESET.
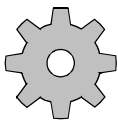
Machine manufacturer

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).

6FC5198-6AA60-0BP0
SINUMERIK 840C, OEM Version for Windows (BA) 4-81

4.3.8.2   DRF offset

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

Note

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

An active DRF offset cannot be deleted via the program (@434).

4.3.9   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.

Order number option

The endlessly turning rotary axis is an option.

The G functions

- G27   Handwheel override
- G105   Endless rotation OFF
- G103   Endless rotation ON clockwise
- G104   Endless rotation ON counter-clockwise
- G119   Oriented STOP
- G94   Simultaneous feedrate F in mm/inch/degrees per min.
- G195   Simultaneous feedrate F in mm/inch/degrees per revolution with reference to the speed/feedrate setpoint of the defined spindle or rotary axis.
- G295   Simultaneous feedrate F in mm/inch/degrees per revolution with reference to the speed/feedrate actual value of the defined spindle or rotary axis.
- G98   Simultaneous feedrate F in revolutions per min.
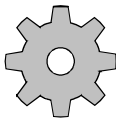
Machine manufacturer

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

4.3.10   Extended stop and retract

Order number option

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.
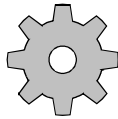
Machine manufacturer

The function must be configured by the machine-tool manufacturer. It is, for example, dependent on the drives used.
Please consult the machine 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.

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
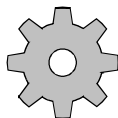
Machine manufacturer

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

4.4   Multichannel display

If the corresponding option is active, 2 channels are displayed simultaneously on the screen.
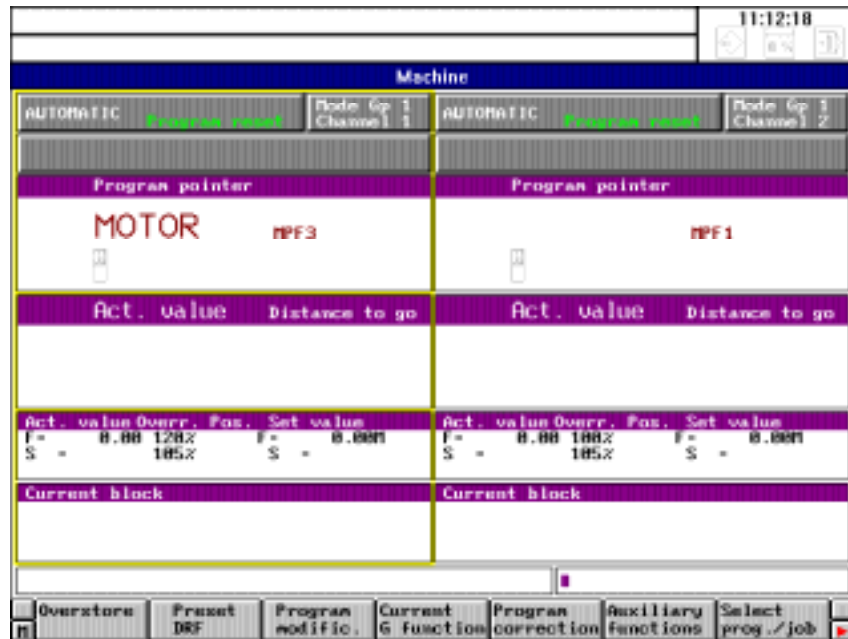
Fig. 4-39 Multichannel display in AUTOMATIC mode

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

There are additional softkeys in the AUTOMATIC display in conjunction with the multichannel display:
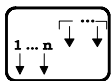
With the channel change key, you advance the channel in the active window.
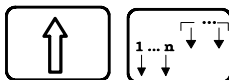
With the mode group change key, you change the displayed mode group on the screen.
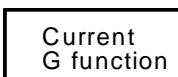
Instead of the display "Current block", the current G functions are displayed.
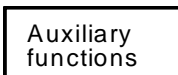
Instead of the display "Current block", the auxiliary functions are displayed.

The RECALL key is used to return to the display "Current block".
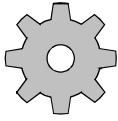
Machine manufacturer

The MD determines which axes and spindles are displayed in the respective channel window (configured by machine manufacturer).

Display screens for more than two channels of a mode group can also be configured by the manufacturer (WS 800A).
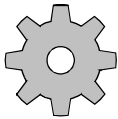
Machine manufacturer

The machine manufacturer defines the effect of keys and rotary switches of the machine control panel on individual channels, axes and spindles. For example the operating mode keys may have an effect on the display mode group (with one machine control panel and several mode groups).

If you have any questions, please refer to the machine manufacturer's documentation.

■

Parameters

5

5.1   Selecting the parameter area

First press the DATA AREA key to select **Area Switchover**.
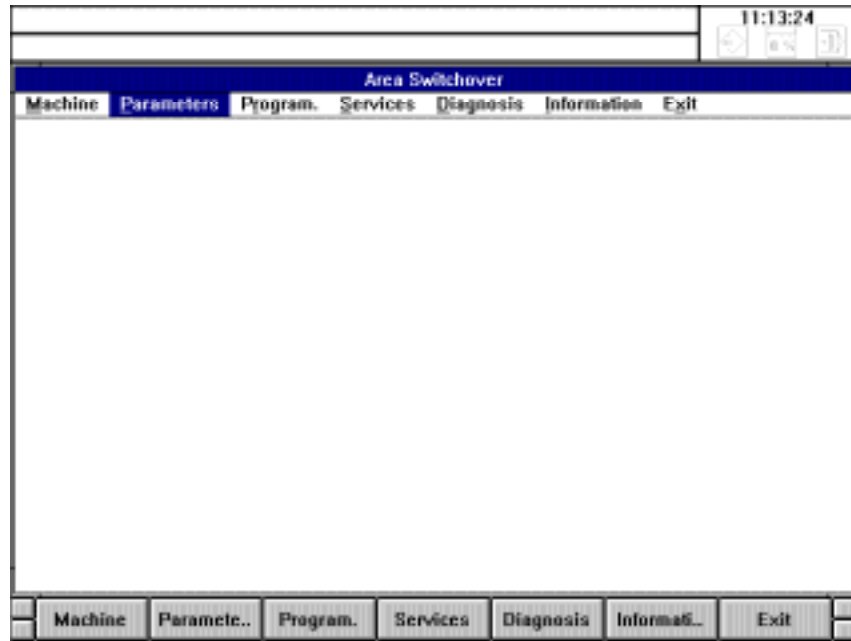
Fig. 5-1   Area Switchover

When **Area Switchover** has been selected:

- Press the PARAMETER softkey. This opens the relevant area immediately.

The **Program Parameters** NCK areas are used to enter program parameters and setting data. The entered values are stored directly in the NCK memory. This memory area is battery-backed, i.e. the data are retained even when the control is switched off.
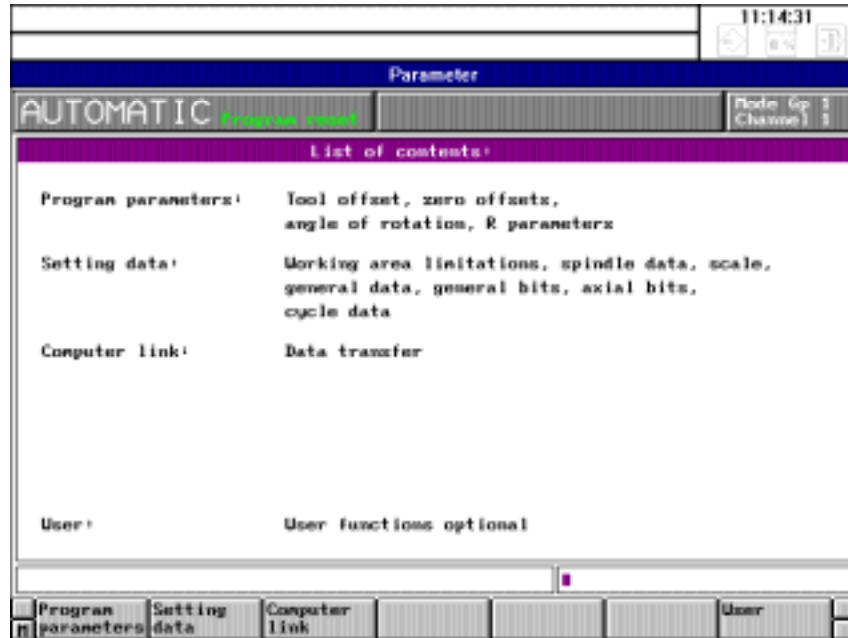
Fig. 5-2   Basic display area: **Program Parameters**

The program parameters and setting data are referenced by the control during execution of a part program and throughout operation of the machine. In contrast, all of the workpiece files in the **Programming**/ **Data Management** areas are stored on the hard disk. Their values therefore have no immediate effect on machine operation.

The program parameters and setting data assigned to a workpiece are loaded from the hard disk to the NCK memory and are active only when the workpiece has been loaded.

The computer link in the NCK area is optional and is therefore described in separate publications.

5.2   Editing data in the PARAMETER area

You can edit the following data in the PARAMETER area:

Preconditions
- 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:

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 manufacturer's documentation.

5.2.1   Selecting data

The procedure for selecting and editing is the same for all data types in this
NCK area. The following paragraphs describe the operator actions for editing
data using the operator panel keyboard.

**Operating
sequence**
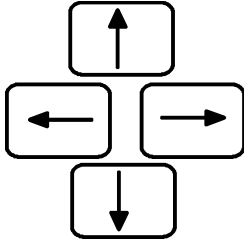
Position in any input field with the cursor keys.
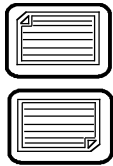
With these keys you page the screen display "down" and "up".
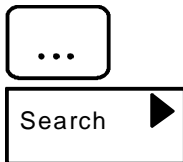
If you are searching for a specific item of data, enter the desired number via the
alphanumeric keyboard and then operate the SEARCH key (do not complete
your entry by pressing the INPUT key). If the value you are looking for is
within the selected data range, the cursor is positioned on this value.

5.2.2 Entering and correcting data

If the cursor has been positioned on the desired item of data, enter the desired value via the alphanumeric keyboard.
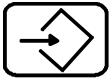
If the input value is correct, you conclude the input with this key. The value is transferred to the memory.

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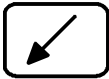
You can activate the cursor in the input window and move it with the cursor keys to insert and delete characters. If you press this key again, you deactivate the cursor.

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
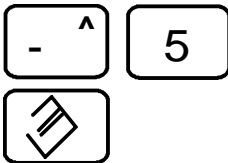
Current value: 50
New value: 45

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.

5.2.3 Entering PLC data in ASCII format

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

If this function is configured accordingly, you can input texts - as for example tool designations - in plaintext into 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-justified with the contents of the input line.

- The text line is deleted with the CANCEL key.

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

```
┌─────────────────────────────────────────────┬────────────┐
│                                               │            │
│                                               │  14:47     │
│                                               │            │
├───────────────────────────────────┬───────────┴────────────┤
│                                                             │
├───────────────────────────────┬────────────┬────────────────┤
│ Automatic   Program reset      │            │ Mode grp:1     │
│                                │            │ Channel :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:                                  │
│                                                               │
│                                                               │
│                                                               │
│                                                               │
│                                                               │
├───────────────────────────────────────────┬─────────────────┤
│                                             │ MILL            │
├──┬──────┬──────┬──────┬──────┬──────┬──────┴┬────────────────┤
│▲ │      │      │      │      │      │       │                │
└──┴──────┴──────┴──────┴──────┴──────┴───────┴────────────────┘
```

Fig. 5-3   Input in ASCII format

# 5.3	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 ▶

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

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

## 5.3.1	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:

Fig. 5-4	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

Name 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 suppressed.

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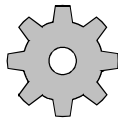
Machine manufacturer

The machine-tool manufacturer determines whether the control is equipped with flexible tool management (see the machine-tool manufacturer's documentation).

Tool type	1. Lathe tool Example 　Turning tool 　Facing tool	2. Lathe tool Example 　Grooving tool 　(left-hand 　edge)	3. Tools with length compensation only Example 　Twist drill	4. Tools with radius compensation and one length compensation Example 　End mill	5. Tool with radius compensation and two length compensations Example 　Angle head cutter
Type	1 ... 9 (Machining in front of/behind the turning centre)		10	20	30

Fig. 5-5　Tool types

　　　　6FC5198-6AA60-0BP0

SINUMERIK 840C, OEM Version for Windows (BA)
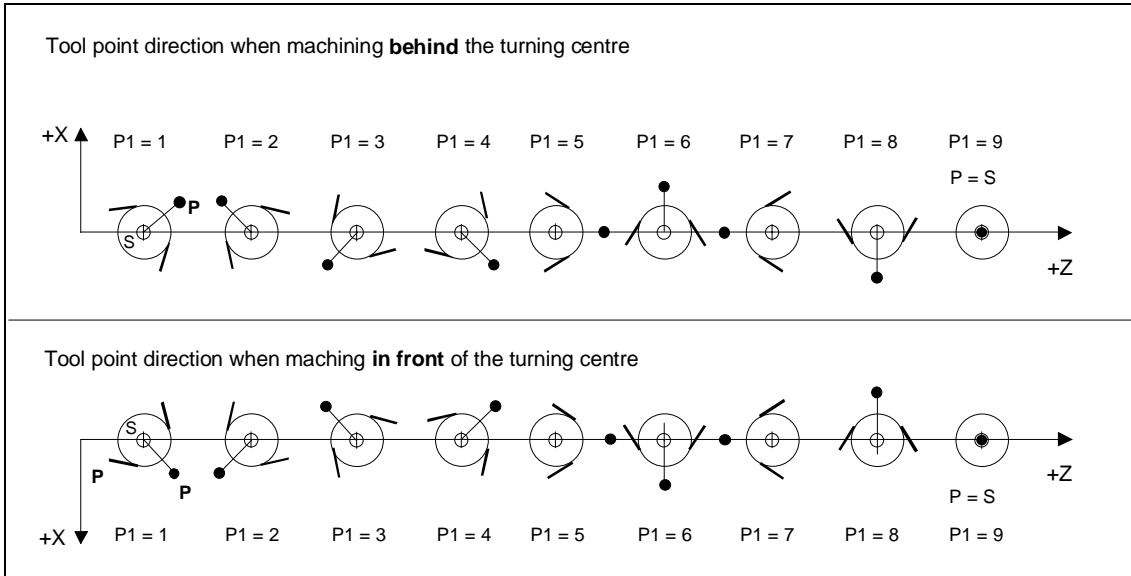
Fig. 5-6   Tool point direction

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 TO parameters "8" and "9" are provided for special applications. The "Base dimension" makes a further tool length compensation possible.
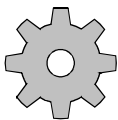
Machine manufacturer

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!
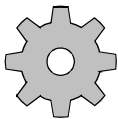
Machine manufacturer

The maximum standard wear value in parameters P5 to P7 is 9.999 mm.

This limit can be modified by the machine manufacturer.
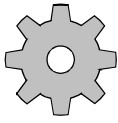
Machine manufacturer

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.
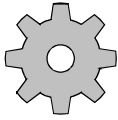
Machine manufacturer

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).

5.3.2   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.
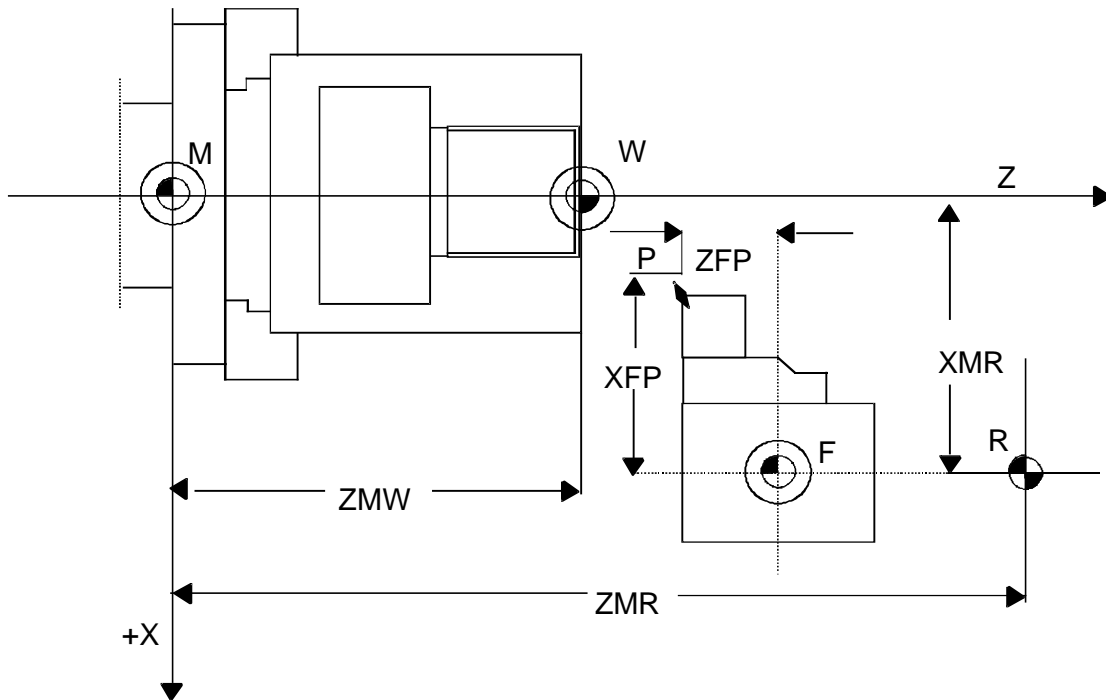
Fig. 5-7   Example: Lathe

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

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 screen.

Note

From SW version 5, also zero offsets from a subprogram Lx under Ny can be activated via G58 Lx Ky or G59 Lx Ky.

The number of the zero offsets which can be stored is limited only by the size of the part program memory.
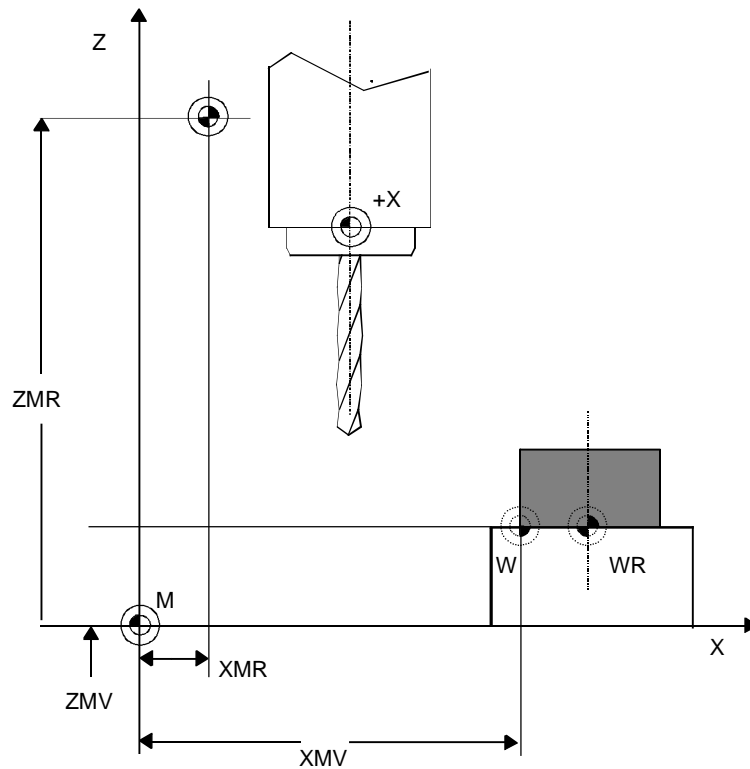
Fig. 5-8   Example: Drilling and milling machine

5.3.3   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).

5.3.4   R parameters

R or calculation parameters in a program represent the numerical value of an address. You can allocate them values within the program and can thus adapt a program to several similar applications (e.g. different feed, spindle feed, spindle speed for different materials, working cycles). Parameters consist of the address R and a maximum four-digit number (except ASCII-R parameters).
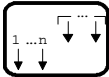
Select the desired channel for channel-specific parameters by pressing the CHANNEL SWITCHOVER key.

| R param. central |

Press the R PARAMETER CENTRAL key.
The parameters R700 ... R1299 are displayed.

Note

This standard number of 600 R parameters may be modified, see also the planning note below!

| 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.

| > |
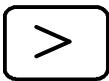
With the ETC key you can extend the menu bar and

| 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 manufacturer's documentation.

Editing is described in the section "Editing data in the parameter area".

Note

Only certain R parameters are freely available to the user. The parameter assignments are shown in the following overview.
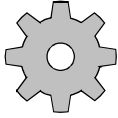
Machine manufacturer

The SINUMERIK 840C has a variable memory configuration. R parameter areas can be reconfigured.

Please consult the machine manufacturer's documentation.

Structure and use of R parameters

Channel 1 to channel 4

	Ch 1	Ch 4	Description
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	**R50 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 subroutine 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 assignment
	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 parameters Supplementary	R600 ⋮ R699	R600 ⋮ R699	**R600 to R699** Reserved for users
Central parameters	R700 ⋮ R999		**R700 to R999** 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** Reserved for users
Chan.spec. ASCII R parameters	R10000 ⋮ R10019		**R10000 to R10019** Reserved for user for the "Select part program for editing" function.

Fig. 5-9   Division of R parameters
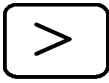
SINUMERIK 840C, OEM Version for Windows (BA)

5.3.5   Plane

> | |
|---|
| **>** |

Press the ETC key to extend the softkey bar.

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.
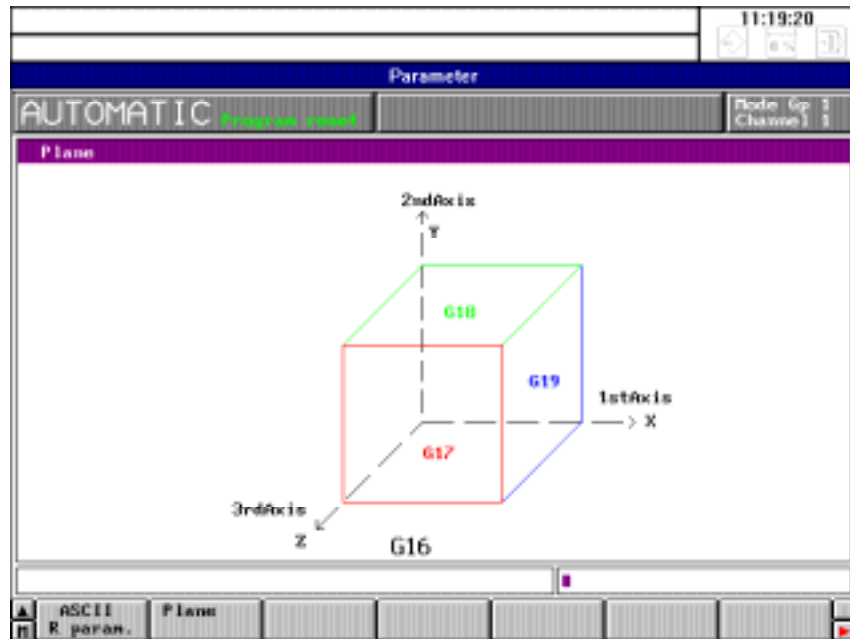
Fig. 5-10   Menu for plane selection in the parameter area

The possible planes are displayed on the screen.

5.3.6 Setting data

| Setting data |

With this softkey you select the input menus for the setting data.

The following menu appears in the softkey bar:

▲Axis/spind converter	General data	Spindle data	Scale	General bits	Axial bits	Position ▶ meas. sign.

| > |

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

▲ Axis/spind converters	Cycle data	Gearbox interpol.	Travel to fixed stop			Search ▶

5.3.6.1 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.

5.3.6.2 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
- Variable increment weighting
- Smoothing constant for thread

Dry run feedrate (SD0)   The dry run feedrate (units/min) 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 modal.
Smoothing constant for thread (SD1)	This function reduces the wear on drives when thread cutting and allows the speed to be kept more constant. 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.

5.3.6.3 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) (SD 401*)	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.

- 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.

5.3.6.4 Scale
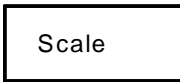
Scale

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 screen.

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.

5.3.6.5 General setting data bits
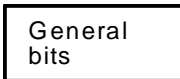
General bits

You select GENERAL SETTING DATA BITS.

The display with the setting data bits of setting data 5000 to 5599 appears on the screen.

The setting data from byte No. 5000 (up to 5799) are made up of 8 bits each: 0 to 7.

Explanation of the setting data structure: (e.g. for setting data (SD) 5001)

You can set the desired bit pattern for any byte in the range from 5000 to 5799 on the screen 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 cycle 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
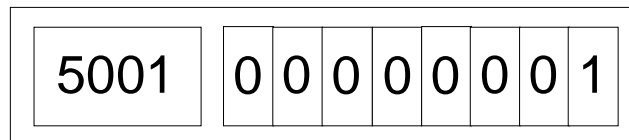
```
┌──────────────────────────────────────────────┐
│  ┌──────┐  ┌──┬──┬──┬──┬──┬──┬──┬──┐          │
│  │ 5001 │  │0 │0 │0 │0 │0 │0 │0 │1 │          │
│  └──────┘  └──┴──┴──┴──┴──┴──┴──┴──┘          │
└──────────────────────────────────────────────┘
       ↑            7 6 5 4 3 2 1 0

  Setting data        Bit pattern for setting
  No.                 data 5001
                      Possible bit states:
                      "0" = Bit is not set
                      "1" = Bit is set
```

5.3.6.6   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 has no effect on the axis feed.
	Bit 1 = 1 :	Override switch has no effect on the rapid traverse of the axis.
	Bit 2 = 0 :	Scale factor G51 is effective during machining.

5.3.6.7   Position measuring signals

Position
meas. signals

You select the softkey and the following display appears:
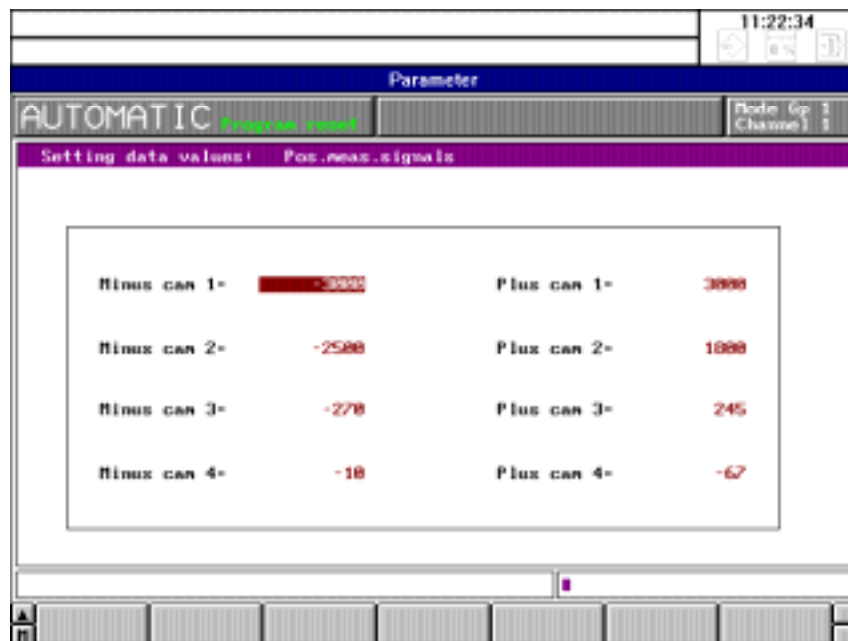
Fig. 5-11 Position measuring signals

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
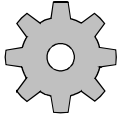
Machine manufacturer

The software cams function must be configured in the PLC user program by the machine manufacturer. Please refer to the documentation for the machine concerned.

Notes

- The software cam function is an option.

- 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.

- The positive and negative cams combined in a cam pair always refer to one NC axis.

- After cam positions have been changed, you must first activate a signal "Transfer of cam values" with an operating sequence (see manufacturer's documentation) before the changes become effective.

5.3.6.8 Cycle setting data

>

Press the ETC key to extend the SETTING DATA softkey bar.

Cycle data
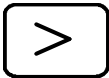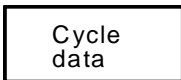
You select CYCLE SETTING DATA.

The first display for cycle setting data (system data) appears on the screen.

Cycle setting data are separate for each channel. They are grouped into four system and user areas:

Table 5-1   Overview of 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.

5.3.6.9   Axis and spindle converter
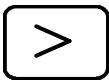
> Press the ETC key to extend the softkey menu bar. The AXIS/SPINDLE CONVERTER softkey is displayed.

Axis/spind
converter

When you press this softkey, the input screenform for the axis/spindle converter function is displayed.
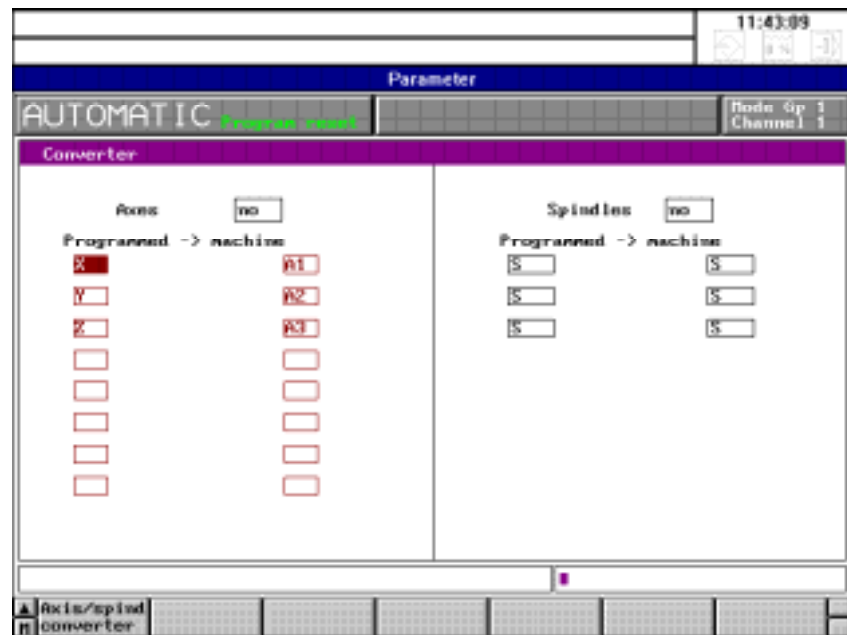
Fig. 5-12   Axis/spindle converter

6FC5198-6AA60-0BP0

SINUMERIK 840C, OEM Version for Windows (BA)

With this function you can convert axis and spindle addresses into other axis and spindle addresses.

Place the cursor on the toggle field 'Axes no'. If the contents can be switched to 'yes', the function can be used in this channel.
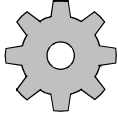
Machine manufacturer

This function must be released and activated by the machine manufacturer for the channel affected (technology MD).

In addition, a release is required in the channel-specific SD 540*.0.
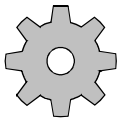
Machine manufacturer

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 be entered via the PLC program or via the part program (@ function).

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.

5.3.6.10 Gearbox interpolation
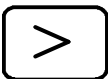
Press the ETC key to extend the SETTING DATA softkey bar.
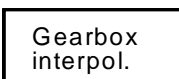
Gearbox interpol.

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 programming with G commands, i.e. each softkey represents a GI command. The G commands are described along with the softkeys.

Fig. 5-13 Gearbox interpolation
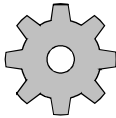
Machine manufacturer

The function gearbox interpolation must be configured by the machine manufacturer (machine data).

Please consult the machine 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 for the GI type linear 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.

Fig. 5-14 IKA relations

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 data record 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	

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$_1$POS I J...LA$_5$ LA$_5$POS 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, the link is switched on, over or 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 data record is loaded from the hard disk into the NCK, the GIA data are also activated without a warm restart.

Operating sequence

Basically, the sequence of operations for activating a gearbox interpolation group is as follow:

1. Define following axis

2. Define leading axis and link type (the IKA data must be loaded for link via IKA tables)

3. SK: Accept configuration (2x)

4. Enter link factors

5. SK: KOP_On/KOP_Switch (2x)

6. If necessary, enter start position

7. Possibly SK: Start position KOP_on/switch (2x)

8. SK: KOP_on all (2x)

Any modification of the configuration is possible only completely new after SK: KOP_off all and SK: Delete configur.
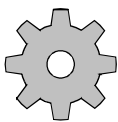
Machine manufacturer

The selected axis must be released as GI following axis via machine data (MD); the control parameters of the compensatory controller must have appropriate values.

The actions "Configure new", "Switchover, link factor" and "Change start position" can be disabled independently of one another and according to the respective axis.

Gearbox interpolation is an option: Please see also the machine manufacturer's documentation.

5.3.6.11 Travel to fixed stop

> Press the ETC key to extend the SETTING DATA softkey bar.
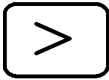
Travel to fixed stop
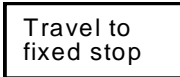
Press the softkey and the input screen is displayed with the configured axes. You can use the function "Travel to fixed stop" to set-up defined forces for clamping workpieces, e.g. tailstocks, sleeves, grippers, etc.
The clamping force for this function is entered in the input fields assigned to the axes.

Input range: 0 to 999999
Tolerance: 0.1% of max. power setpoint (motor torque)

Note

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

- The function can be used for all axes.

- Spindles that are to travel to the fixed stop must first be switched to C axis mode.
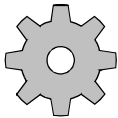
Machine manufacturer

The function "Travel to fixed stop" must be configured in the PLC user program. Please refer to the documentation for the machine concerned.

■

Programming 6

6.1   Selecting the Programming area

Press the DATA AREA key to select Data Management.
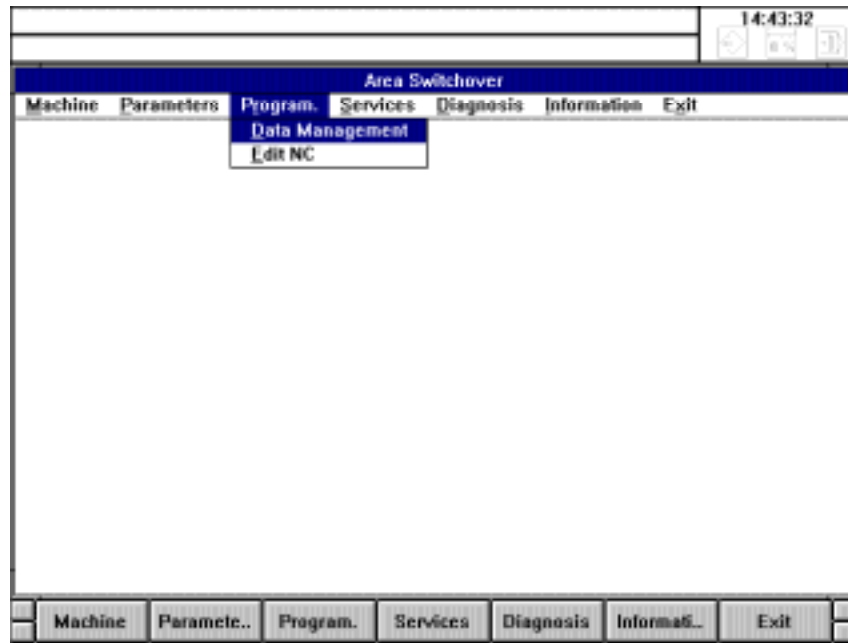
Fig. 6-1   Area Switchover

When **Area Switchover** has been selected:

- First press the PROGRAM. softkey to open the menu

- You can select the areas in the menu by entering the following letters:

  - "D" Data Management

  - "E" Edit NC

Select **Data Management** to manage files on the hard disk or transfer files between the NCK and MMC.

The **Edit NC** area is used exclusively to manage and create part programs in the NCK memory.

6.2   Data management

A special data manager has been developed for managing the NCK/MMC data on the SINUMERIK 840C.
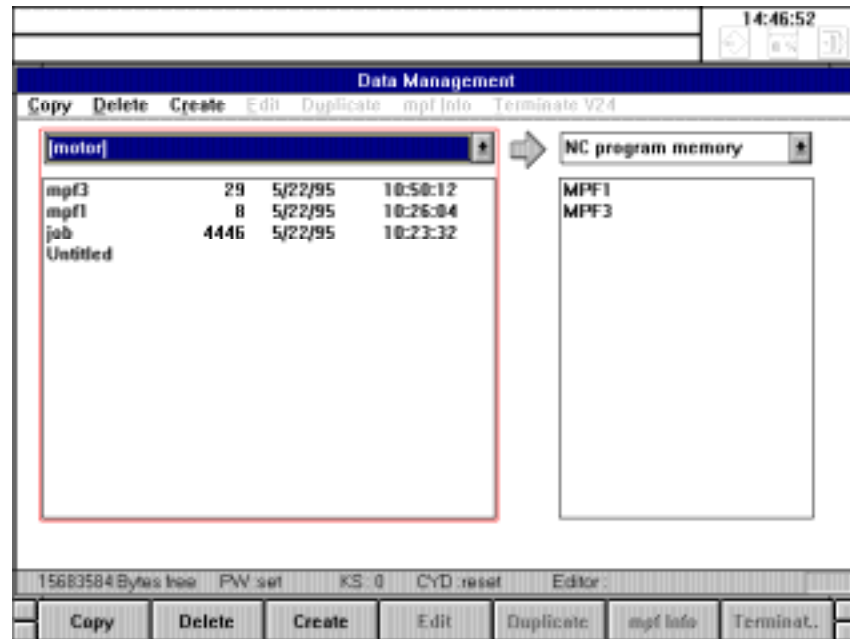
Fig. 6-2   Data Management basic display

In addition to the usual functions such as copy, delete, create, edit and duplicate, the data manager provides the following extra advantages:

- NC data management is workpiece-oriented, i.e. NCK data such as main programs, subroutines, setting data, tool offsets, zero offsets and R parameters can be assigned to a workpiece. A workpiece corresponds to a directory. This directory can either be on the local hard disk of the MMC, on a disk drive or on a remote computer.

- All the data required for a workpiece can be transferred from the hard disk to the NCK or peripheral device with a single operator action.

- When working with the data manager, you do not need to worry about the operating system names of PC drives and directories.

- In contrast to the complex MS-Windows File Manager, you cannot accidentally delete or alter system files when using the **Data Manager**.

6.2.1 Structure of data management

The central elements of the data manager are presented in two boxes containing NC files.

Left-hand list box   The left-hand list box contains the files belonging to a workpiece or another directory on the hard disk. Only those files whose names comply with NCK conventions are displayed in the left-hand list box. You select the workpiece or directory in the combo box (single-line list box) displayed directly above the left-hand list box. You can select the following directories in the combo box:

- Workpieces: Workpieces are directories where all the data required for the production of a physical workpiece can be stored. When the workpiece is created, the name of the workpiece (up to 8 characters are permitted) can be freely assigned by the user. The names of workpieces stored on the hard disk in directory C:\LOCAL are enclosed in square brackets [...].

  Workpieces that are located in a directory other than C:\LOCAL display an identifier after the square brackets that can be freely configured by the machine-tool manufacturer. The identifier refers to a higher-level directory (container for workpieces) that is either located on the local hard disk, the diskette box or a remote computer.

  Examples: [...]-*network*, [...]-*flexos* etc.

  For workpieces that lie in a FlexOS directory, the workpiece names are displayed together with their extension *.064.

  The same higher-level directories (workpiece container) are displayed in the DATA SELECTOR of the NCK as are displayed in the data management display.

- MD user: This directory contains the manufacturer's machine data. The data can be loaded into the NCK system memory during start-up.

- MD standard M/ MD standard T: These directories contain the standard Siemens machine data for M and T controls.

- Cycles-user: This directory contains the cycles of the machine manufacturer.

- Cycles-standard: This directory contains the turning and milling cycles supplied by Siemens as standard with the control.

- Clipboard: The Clipboard is used to store files when data is input via the serial interfaces. The files can be transferred from the Clipboard to the desired directory. Section 6.2.6 "Data communication between the MMC and peripheral devices" describes which of the individual data are stored.
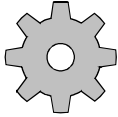
Machine manufacturer

Depending on the type of file (e.g. cycles, MD data, IKA data etc.), functions such as delete, edit, copy, duplicate etc. can be disabled or released individually on the control MMC.

Please refer to the machine manufacturer's documentation.

The following table provides an overview of the data types that exist and the directories in which they can be stored. Only those files whose names comply with NCK conventions can be transferred to the NCK memory.

Table 6-1   Data types for SINUMERIK 840 C

Data type	File name	Parameter	Directories on hard disk
Job list	job	---	WORKPIECE
Main program	mpfxxxx	xxxx: 4-digit program number	WORKPIECE
Subroutine	spfxxxx	xxxx: 4-digit program number	WORKPIECE
Cycle	spfxxxxzpfxxxx	xxxx: 4-digit program number	CYCLES-USER CYCLES-STANDARD WORKPIECE
Zero offset	zoa0 zoak	k: 1 ...6 channel number	WORKPIECE
Tool offset	toak	K: 1 ... 6 channel number	WORKPIECE
R parameter	rpa0 rpak	k: 1 ...6 channel number	WORKPIECE
NC setting data	sea	---	WORKPIECE
Cycle setting data	sea4	---	WORKPIECE
Cycle machine data	tea40	---	MD-user
Cycle machine data	tea4k	k: 1 ... 6 channel number	MD-user
NC machine data	tea1	---	MD-user
PLC machine data	tea2	---	MD-user
IKA data	ikan	n: 1, 2, 3	MD-user workpiece
GIA data	gia	---	MD-user

Notes

On the control MMC, the new data type zpfxxxx is used for the type "cycle". With this new data type, separate disables or releases of functions (e.g. delete, copy etc.) can be implemented for cycles (zpfxxxx) and subprograms.

When being transferred to the NCK part program memory, the "zpf files" are reconverted into "spf files" with cycle ID. Prerequisite for this is that the number areas for subprograms and cycles are strictly separated.

The new individual, that means data-type-dependent disables enable for example the machine manufacturer to store the cycles also in workpieces without that those can be edited or read out.

Right-hand list box The contents of the right-hand list box depend on the "device" selected in the combo box displayed directly above the right-hand list box.

The following "devices" and directories are available:

- All workpieces and directories of the MMC (these are also displayed in the left-hand combo box). The names of workpieces stored on the hard disk in directory C:\LOCAL are enclosed in square brackets [...].

  Workpieces that are located in a directory other than C:\LOCAL display an additional identifier after the square brackets.

❗ **Important**

With the help of the **WEdit** editor, files whose names do **not** comply with the NCK convention (e.g. %100 instead of mpf100) can be created in workpieces or in other directories. These files are displayed in the right-hand list box **only**.

These files are **not** displayed in the left-hand list box and **cannot** therefore be transferred to the NCK or via the serial interface or to the diskette drive.

- The NCK program memory and the system data memory of the NCK
- The FD-E2 diskette drive (if installed)
- The settings for the serial interfaces (devices) of the MMC.

You can use the EDIT key to switch between the individual list boxes in the data manager.

Arrow The direction of the arrow between the list boxes visualizes the direction in which files are copied and depends on the list box selected.

Menu bar Individual files or complete workpieces can be selected within the list boxes using the ARROW keys and edited using the commands on the menu bar.

The commands on the menu bar are described in the following paragraphs.

Status line The status line in the data manager displays additional information.

"1234567 bytes free" The far left-hand box on the status line of the data manager shows you the amount of free memory still available on the hard disk.

PW

The password status is displayed next to this. The following statuses are possible:

- PW on (set)

- PW off (deleted)

KS

KS displays the current setting of the keyswitch. Depending on its position (1-4), certain functions are disabled or can be executed (see section 2.2.4).

CYD

The CYD entry shows you the current status of the cycle disable:

- CYD on (set)

- CYD off (deleted)

Cycles in the NCK memory can only be loaded in the editor when the cycle disable is off. The behaviour of cycles on the MMC side can be planned separately by the machine manufacturer.

Status

The read/write rights in the status line of the data management are displayed according to the position of the focus and disables or releases planned by the machine manufacturer.

- The focus marks the combo box:

  - Write right if you may copy in the directory selected.

  - Read right if the edit function is allowed for individual files of the directory selected.

  - In all other cases, no rights are dissplayed.

- The focus marks the list field:

  - Write right if the selected file may be deleted.

  - Read right if the selected file may be edited.

  - In all other cases, no rights are displayed.
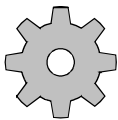
Machine manufacturer

The files are displayed together with the date in the left-hand list box in the data manager.

The date display can be configured by the machine manufacturer.

6.2.2 Workpiece management on hard disk

The SINUMERIK 840C data manager provides convenient functions for simple and efficient management of workpieces and their files. This section describes how to perform the following tasks:

- Create workpieces and their related files and store them on the hard disk.

- Perform data management tasks, such as copying, deletion and duplication of data on the hard disk.

- Edit part programs, job lists, R parameters, setting data, cycle setting data, zero offsets and tool offsets with the aid of the **WEdit** editor.
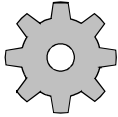
Machine manufacturer

The machine manufacturer can plan the workpiece directories to be incorporated in the data management.

As a standard, only the directory C:\LOCAL is existing.

6.2.2.1 Creating workpieces

New workpieces (directories) can be created on the hard disk or on other drives with the data manager.

- When created, these workpieces automatically appear in the workpiece directory selected (MMC hard disk, network drive etc.).

- Workpieces cannot be created in a FlexOS directory.

When the data manager is active:
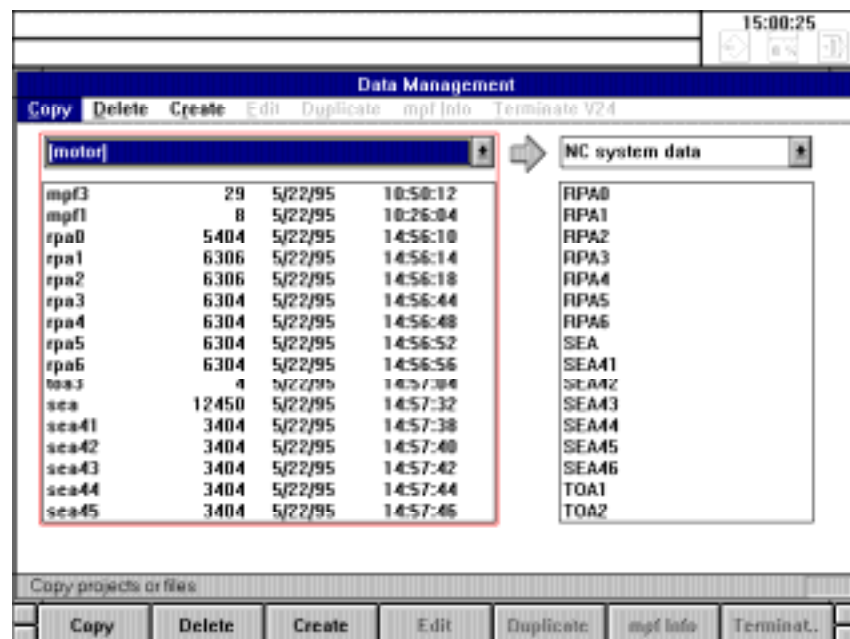
Fig. 6-3   Data manager

- Press the EDIT key to select the combo box on the left-hand side of the data manager.

- Press the CREATE softkey.

6FC5198-6AA60-0BP0   © Siemens AG 1997 All Rights Reserved
SINUMERIK 840C, OEM Version for Windows (BA)

The following input boxes and list fields are added to the existing list boxes.
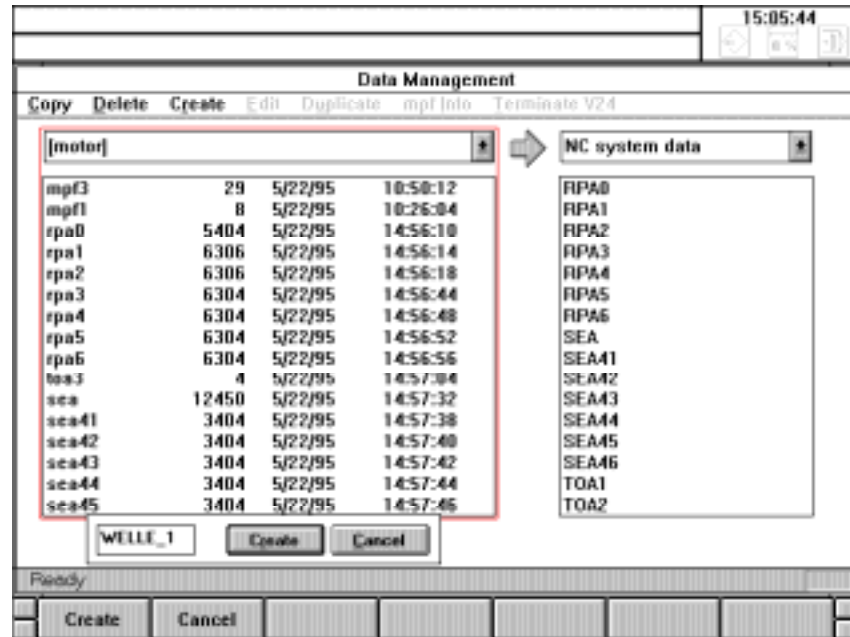
Fig. 6-4   Creating workpieces

- Select the desired workpiece directory in the list field. When starting a dialog, an additional list field with the possinble workpieces is displayed.

  The workpiece which has been activated in the active combo box before executing the function is selected.

- Select the desired workpiece in the list field.

- In the left-hand text box, enter the name of the workpiece to be created (WELLE_1 in the above example).

- Press the CREATE softkey.

A workpiece with the desired name is created in the workpiece directory selected and displayed in the single-line list box.
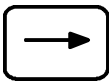
You can use the RIGHT ARROW key to open single-line list boxes.

Note

By entering the workpiece name (e.g. WELLE_1), you have simply created a subdirectory for the workpiece type in the workpiece directory selected. A standard job list is also created automatically.

Up to 350 files can be stored in a workpiece.
You can also store the data described in section 6.2.1 in a workpiece.

6.2.2.2 Creating NCK files

Any program files in an existing workpiece on C:\LOCAL or another directory (if access is authorized) can be created or edited with the aid of the integrated editor **WEdit**.

NCK files in a FlexOS directory can only be read by the **WEdit** editor. They cannot be created or edited.

- First press the EDIT key to select the combo box on the left-hand side of the data manager.

- Open the single-line list box with the RIGHT ARROW key.

- Select the desired workpiece with the vertical ARROW keys.

- Press the INPUT key. The selected workpiece is displayed in the single-line list box.
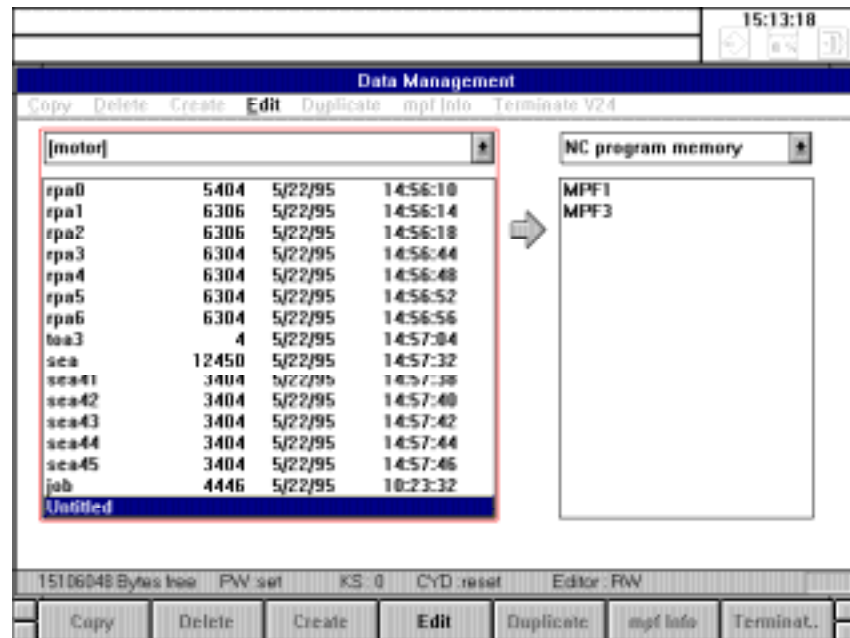
Fig. 6-5   Creating NCK files

- Press the EDIT key to switch to the lower list box.
- Use the ARROW keys to select the item in the list called "Untitled".
- Press the EDIT softkey.

The **WEdit** editor is started.

When you have finished editing, you can save the created file under any DOS name (**8** characters.**3** characters).

- To do this, select **Save As...** from the **File** menu.

The following dialog box appears:

```
┌──────────────────────────────────────────────┐
│ ▬        WEdit - Save as...                    │
├──────────────────────────────────────────────┤
│ Save file as                                   │
│ ┌──────────────────┐   ┌──────────────┐        │
│ │ MPF100           │   │      OK       │        │
│ └──────────────────┘   └──────────────┘        │
│                        ┌──────────────┐        │
│                        │    Cancel     │        │
│                        └──────────────┘        │
│                                                │
└──────────────────────────────────────────────┘
```
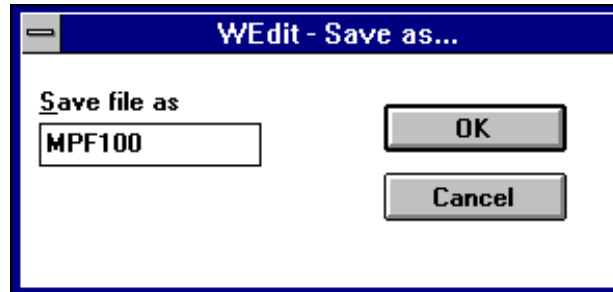
Fig. 6-6   "Save As ..." dialog box

- Enter the file name in the text box.
- Press the **OK** softkey.

The file is stored in the selected directory.

Note

The file name is only displayed in the left-hand list box of the corresponding directory if the naming complies with NCK conventions.

If an illegal name has been assigned to the file (e.g. %100 instead of mpf100), the file is only displayed in the list box on the right.

6.2.2.3   IKA data

IKA stands for interpolation and compensation with tables. This function can be used on the SINUMERIK 840C in relation to machines and/or workpieces.

1. IKA data **defining a machine**
   The IKA function is used as a compensation for machine-dependent error curves caused by mechanical considerations and temperature, etc. These data are entered by the machine manufacturer, stored in the MD user directory and protected by password.

2. IKA data **defining a workpiece**
   The function is used here to describe the contours of a workpiece in terms of IKA data. Up to 16,000 interpolation points can be defined in a table. Up to 32 IKA relationships (IKA data records) are supported.
   This function can be used to generate complex contours using a burst of interpolation points.
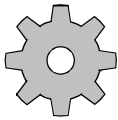
Machine manufacturer

The IKA function must be configured by the machine manufacturer. Please refer to the machine manufacturer's documentation for further information.

Caution

The IKA is **not** included in the actual value display.

6.2.3   Creating and editing job lists

A job list can be created for each workpiece to be machined. The job list contains instructions to help the user control the loading of workpieces onto the NCK.

A job list contains not only the main programs and subroutines required for the production of the workpiece, but all data related to the machining process. This data includes tool offsets (TOA), zero offsets (ZOA), R parameters (RPA) and setting data (SEA, SEA4).

The job list can also be used to select part programs on the individual channels of the NCK.

The job list is executed whenever a workpiece is selected in the Machine area (AUTOMATIC display) or copied (loaded) into the part program memory using the data manager.

If no job list is stored in a workpiece, all the files of the workpiece are loaded when the workpiece is selected in the Machine area or copied from the data manager to the NCK memory.

When you create a new workpiece, a standard job list is automatically created in this workpiece. This standard job list contains a description of the job list syntax (entered as comments), as well as a LOAD * command that loads all of the workpiece files into the NCK memory.

The job list of a workpiece **must** have the file name "job".

Please refer to the next section "Commands and syntax of job lists" for further information on the syntax of the job list.

6.2.3.1 Creating job lists

If no job list is in the selected workpiece, you can create a standard job list by creating a new workpiece and copying the standard job list from here to the desired directory:

- Activate the Program Management window.

- Create a new workpiece.

A standard job list is automatically generated when the workpiece is created. It then serves as a template for the job list in your selected workpiece.

- In the left-hand single-line box, select the workpiece for which you want to create a job list (example: [welle_1]).

- In the right-hand single-line box, select the workpiece that you have already created (e.g. [welle_2]). Check that the selection bar is highlighting the file "job" in the right-hand list box.
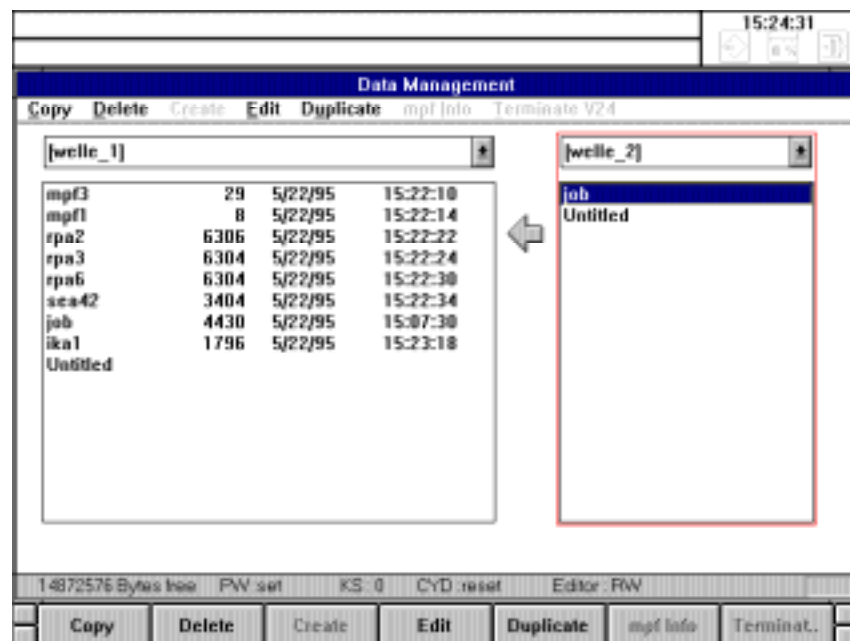
Fig. 6-7   Creating job lists

- Select the COPY command from the softkey bar of the data manager.

The job list is copied from workpiece [welle_2] to the selected workpiece and can be edited with the **WEdit** editor.

6.2.3.2 Editing job lists

You can edit job lists and thereby adapt them to your requirements with the aid of the integrated editor **WEdit**.

- Activate the **Data Management** window.

- Select the workpiece whose job list you want to edit in the left-hand single-line box

- Position the selection bar on the file named "job" in the left-hand list box.

- Press the EDIT softkey in the menu bar of the data manager.

Fig. 6-8   Editing the standard job list

An operating guide is displayed in the job list with a complete list of the instruction set. A job list can contain the following instructions in any order:

1. Clear files for NCK memory (CLEAR ...)

2. Load instructions (LOAD or LOADCYC ...)

3. Start preparation (SELECT ...)

4. Start of MMC applications (CALL...)

5. Comments

Each line contains exactly one instruction. The instructions are processed in the order in which they appear in the job list.

The individual sections of the job list (CLEAR, LOAD etc.) begin with the 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.

6.2.3.3   Syntax description for the job lists

Four types of instruction are used in the job lists:

- Comments
- Clear instruction   CLEAR
- Load instruction   LOAD or LOADCYC
- Select instruction   SELECT.

This sequence is mandatory.

Comment

All terms in round brackets are comments and are not taken into account when processing the job list. Comments must stand on their own in a line.

CLEAR instruction

With the CLEAR instruction, part programs are deleted from the NCK part program memory.

Command syntax

CLEAR ⌴ *name* ⌴ [attr.]L$_F$ (⌴ = 'blank')

The following identifiers can be used for "*name*":

MPFn (one main program)
SPFn (one subroutine)

MPF[n,m] (part programs MPFn...MPFm)
SPF[n,m]   (subroutines SPFn...SPFm)

MPF* (all main programs)
SPF* (all subroutines)

* refers to all part programs in the part program memory (in the NCK).

The following attribute can be used for the CLEAR command:

-u (unconditional deletion)

If cycle disable is not set, cycles can be deleted in the NCK memory independently of their status.

Examples

Deletion of the MPF7 part program:

CLEAR ⌴ MPF7 L$_F$

Deletion of all subroutines or cycles SPF 1 to 999:

CLEAR ⌴ SPF[1, 999] ⌴ -u L$_F$

Deletion of all MPF, SPF:

CLEAR ⌴ * L$_F$

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

Load instructions   The LOAD instruction can be used to load one or more files from the current workpiece into the part program memory of the NCK.

Entry of the workpiece from which the data are to be loaded is optional. If the workpiece is not included in the LOAD instruction, the file to be loaded is searched for in the current workpiece.

Command syntax   LOAD ⌴ *name* ⌴ [channel/TO-area] L$_F$

The identifier "*name*" consists of the optional workpiece name and the file name "WORKPIECE ⌴ FILE":

LOCAL/WORKPIECE ⌴ MPFn or
WORKPIECE ⌴ MPFn (a main program in directory
C:\LOCAL\WORKPIECE on the local hard disk)

SPFm (one subroutine in the current workpiece)

ZPFm (one cycle file in the current workpiece)

MPF[n,m] (part programs MPFn...MPFm

SPF[n,m] (subroutines SPFn...SPFm)

ZPF[n,m] (cycle files ZPFn...ZPFm)

* selects all part programs of a workpiece, i.e. all files of a workpiece are loaded into the NCK memory.

Note   The cycle files ZPFn are renamed automatically into SPFn during loading from MMC into the NCK memory.

❗ Important

Drive identifiers such as C: must not be included in the "name" identifier. If a drive identifier is used, the job list is aborted with an error message.

The *PATH* file can be used to access data from any drives.

Only NCK-capable files can be loaded via the job list: MPF, SPF, TOA[1], ZOA[3], RPA[3], SEA, SEA4[2], IKA1, IKA2, IKA3. The above file names must be strictly observed, as the SINUMERIK 840C derives the file type from its name..

   1:     TO area must be specified

   2:     Channel number must be specified

   3:     Channel number must be specified, 0 means data for all channels

Where files stored in the NCK have specific channel or area assignments, the last character of the file name must identify the channel or area number.

"PATH" file   If an NCK file with a LOAD instruction is to be loaded, the file is initially searched for in the specified workpiece or, if no workpiece name has been specified, in the current workpiece. If the file is not found, a search is made in the workpiece indicated in the optional file *PATH* under the section [Path]:

name=path

Name designates, for example, the part program name (e.g. MPF100) of the load instruction; path the entire disk drive path in which the file "*name*"is stored. With this redirection function it is possible to load NCK files from any computers in the network into the NCK memory transparently.

Example 1:   Loading part program MPF4711 from workpiece WELLE1:

LOAD ⌴ WELLE1 ⌴ MPF4711 L$_F$ or
LOAD ⌴ LOCAL/WELLE ⌴ MPF4711 L$_F$

The file *PATH* with the following entries exists in workpiece WELLE1:

[path]
mpf4711=h:\network\parts\welle2
mpf3933=m:\partprg\motor

Part program MPF4711 is loaded in the specified network path.

Note

The PATH file is **not** searched if wild card characters are used.

Example 2:   LOAD ⌴ WELLE1 ⌴ * L$_F$ (load all NCK files)
LOAD ⌴ . ⌴ * L$_F$

In Example 2 the file *PATH* is not searched.

Example 3   LOAD ⌴ WELLE3 ⌴ MPF[10,99] L$_F$
LOAD ⌴ LOCAL/WELLE3 ⌴ MPF[10,99] L$_F$

In Example 3 the file *PATH* is also searched.

Load instructions for cycles   The LOADCYC instruction can be used to load one or more cycles from the CYCLE-USER and/or CYCLE-STANDARD directories into the part program memory of the NCK.

These files can be given additional attributes in the NCK memory to prevent overwriting. This is especially useful for preventing OEM applications from accidentally overwriting or deleting important programs (cycles) in the NCK memory.

Command syntax   LOADCYC ⌴ STANDARD|USER ⌴ name ⌴ [attr.] L$_F$

The following identifiers can be used for "name":

SPFn (a cycle)

SPF[n,m] (cycles SPFn...SPFm)

The character * symbolizes all cycle In the selected directory (STANDARD or USER).

The following individual attributes are permitted:

-r :   **not** readable

-w :   **not** writeable

-x :   **not** executable

-u :   unconditional writing

Note

- If the LOADCYC instruction is used without an attribute, all attributes (-r, -w, -x, -u) are set.

- The parameter '-u' signifies unconditional writing, i.e. a write-protected cycle that already exists is overwritten.

Examples

LOADCYC $\sqcup$ USER $\sqcup$ * L$_F$
Loads all user cycles into the NCK memory

LOADCYC $\sqcup$ STANDARD $\sqcup$ SPF[n,m] L$_F$
Loads all (standard) Siemens cycles from SPFn to SPFm

LOADCYC $\sqcup$ USER $\sqcup$ SPF4711 $\sqcup$ -x $\sqcup$ -u $\sqcup$ -w L$_F$ or
LOADCYC $\sqcup$ USER $\sqcup$ SPF4711 $\sqcup$ -xwu L$_F$

Cycle SPF4711 is read from the USER directory and loaded into the NCK memory as an undeletable and non-executable file.

Note

When the control is started up the cycles are transferred to the NCK memory by the LOADCYC entry in the job list independently of "cycle disable".

Selection instruction

With the SELECT instruction a part program is selected for execution. Both the part program name and the execution channel must be specified. The transferred workpiece is also transferred to the program pointer of the NCK.

Execution from external

The optional parameter DISK can be used to implement **Execution from external**, i.e. from the local hard disk drive (redirection via *PATH* file).

!

Important

Execution from external is not possible due to the necessary name conversion ZPFn -> SPFn for cycles (zpf files).

With the execution from external, the name of the workpiece in which the file to be executed is located must be unambiguous within the different workpiece directories. Different workpieces must not be stored under the same name in different workpiece directories (e.g. C:\LOCAL and H:\NETWORK\PARTS).

Command syntax   SELECT $\sqcup$ *name* $\sqcup$ CH=nr $\sqcup$ [DISK] L$_F$

name:	Part program name (MPF... or SPF...)
CH=nr:	Channel number (1 ... 6)
DISK:	"Execution from external" is selected "

If "Execution from external" has not been selected, the selected part program must first have been transferred to the part program memory of the NCK using a load instruction.

Examples   SELECT $\sqcup$ MPF7 $\sqcup$ CH = 1 L$_F$

Select part program MPF7 in channel 1:

SELECT $\sqcup$ MPF7 $\sqcup$ CH=1 $\sqcup$ DISK L$_F$

Select part program MPF7 in channel 1 for "Execution from external". After NC START, part program MPF 7 is loaded section by section from the remote mass storage device from the workpiece specified in the program pointer.

!

Important

Execution of the job list of a FlexOS workpiece is only possible if the job list contains no SELECT instruction.

Notes   The SELECT instruction of the job list is only processed if the NCK requests a workpiece. It is **not** processed if a workpiece is copied using data management.

Start instructions   The existing job list syntax has been extended with SINUMERIK 840C OEM Version for Windows so that MMC applications can also be started from the job list.

Applications which perform e.g. a format conversion can thus be started.

Command syntax   CALL $\sqcup$ [*Drive*][*Path*][*Program*];[*Status*]

Drive:	Drive on which the application is located
Path:	Application path
Program:	Program name of the application
Status:	WAIT

SINUMERIK 840C, OEM Version for Windows (BA)

You can define with status whether the system awaits an application feedback when processing the job list or if processing of the job list is continued after successful start of the application with the next instruction.

Example   CALL ⊔ C:\OEM\PROG\CONVERT.EXE;WAIT

Note   If an application for which the status = WAIT has been defined in the job list does not give any feedback within a planned period of time, the following alarm is displayed:

100203 Overrun of the time provided with application in the job list
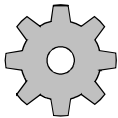
Machine manufacturer

For the MMC applications to be used on your machine tool, please refer to the machine manufacturer's documentation.

! **Important**

If a syntax error occurs in the job list, loading is interrupted with an error message. If this is made by the data management, a dialog box is displayed with the text of the defective line. If this is made via PLC, no dialog box is displayed but the error message is sent directly to the PLC

6.2.4 Copying, deleting and duplicating files

The data manager on the SINUMERIK 840C provides functions for copying, deleting and duplication for managing the files and directories of the MMC and NCK areas in the familiar way.

Copying   The copy function is used to copy individual NC files from one workpiece to another:

- Select the left-hand combo box with the EDIT key.

- Open the combo box with the RIGHT ARROW key.

- Use the ARROW keys to position the selection bar on the workpiece containing the desired file ([shaft1] in the example).

- Press the INPUT key. The workpiece is displayed in the combo box.

- Select the right-hand combo box with the EDIT key and open it.

- Use the ARROW keys to position the selection bar on the workpiece to which the file is to be copied ([motor] in the example) and press the INPUT key.

- Select the left-hand list box with the EDIT key and position the selection bar on the file to be copied (mpf123 in the example).
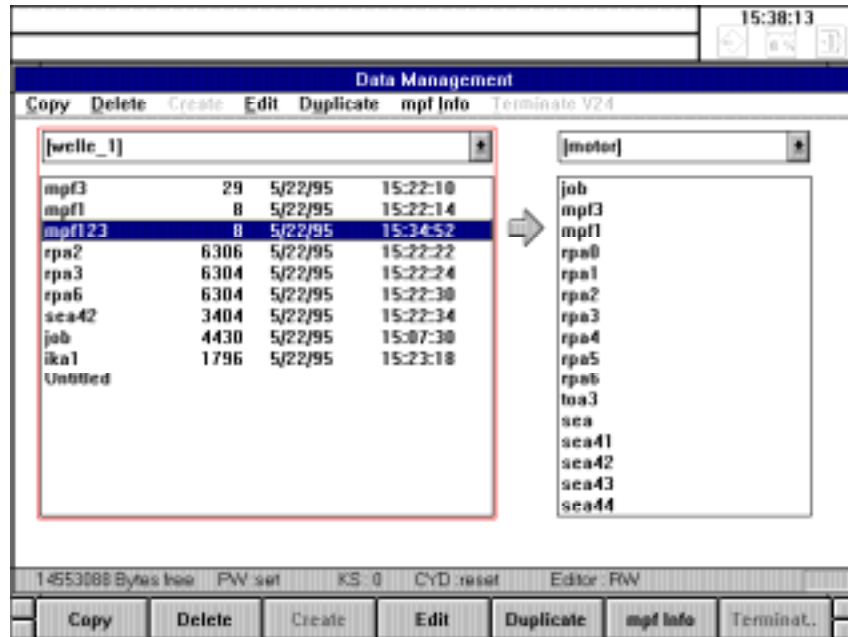
Fig. 6-9   Copying data

- Press the COPY softkey.

The file is copied under the same name to the selected workpiece and displayed in the appropriate list box.

If a file with the same name already exists in the destination workpiece, a dialog box prompts you to confirm whether the existing file is to be overwritten. Press the YES softkey to overwrite the existing file, otherwise select NO.

Note

This function can be used to transfer a complete workpiece with all files from the hard disk to an external "device" such as a diskette drive or serial interface.

If you change to another application when a message box is displayed by the data management, the data management can be selected again only via the task manager (see section 3.3).

6FC5198-6AA60-0BP0   © Siemens AG 1997 All Rights Reserved
SINUMERIK 840C, OEM Version for Windows (BA)

Delete

This command is used to delete individual files and empty workpieces. It can also be used to delete part programs in the NC program memory.

Workpieces in a FlexOS directory or workpieces on a network disk drive cannot be deleted.

Deleting files

Proceed as follows to delete individual files in workpieces or in the NCK part program memory:

- Select the right-hand combo box with the EDIT key and open it with the RIGHT ARROW key.

- Position the selection bar on the workpiece or on "NC Program Memory".

- Press the INPUT key. The selected item is entered in the combo box.

- Select the list box immediately below the combo box using the EDIT key.

- Position the selection bar on the file to be deleted (mpf123 in the example).

- Press the DELETE softkey. The following dialog box appears.

Fig. 6-10   Dialog box: Query before deleting

If you do not have sufficient access rights, DELETE is dimmed and the function cannot be executed. If necessary, use the keyswitch to increase your access rights before attempting to delete.

- Confirm deletion by pressing the YES softkey.

The selected file is deleted.

Deleting workpieces

The same procedure is used to delete workpieces on the local hard disk (not FlexOS workpiece):

- Select the left-hand combo box with the EDIT key.

- Position the selection bar on the workpiece to be deleted.

- Use the EDIT key to select the combo box above with the desired workpiece.

- Press the DELETE softkey. The "Delete Workpiece" dialog box appears.
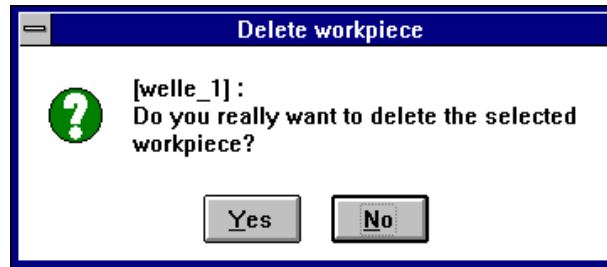
Fig. 6-11   Delete Workpiece dialog box

• Confirm deletion with the YES softkey.

Duplicating

The **Duplicate** function can be used to duplicate individual files of the currently selected workpiece or the NC program memory.

• Select one of the two combo boxes with the EDIT key.

• Open the combo box with the RIGHT ARROW key.

• Use the vertical ARROW keys to position the selection bar on the workpiece or on "NC Program Memory". You can select "NC Program Memory" in the right-hand combo box only.

• Press the INPUT key. The selected item is entered in the combo box.

• Select the list box below the combo box using the EDIT key.

• Use the vertical arrow keys to position the selection bar on the file to be duplicated.

• Press the DUPLICATE softkey. The following extra input boxes appear.
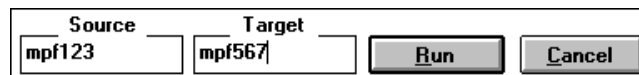
Fig. 6-12   Duplicating

• Enter a new name for the duplicated file and confirm your entry with the RUN softkey.

The selected file is saved again under the new name in the same workpiece or in the NC program memory.

Note

It is not possible to duplicate workpieces using this procedure. To duplicate a workpiece you must first create a new destination workpiece and then copy all of the source files into the destination workpiece.

6.2.5   Data communication between NCK and MMC

The data manager can also be used to exchange NCK-compatible data between the MMC and the NCK memory.

6.2.5.1   Loading data

Loading workpieces

Proceed as follows to load a complete workpiece:

- Select the right-hand combo box with the EDIT key.

- Open the combo box with the RIGHT ARROW key and use the vertical ARROW keys to position the selection bar on "NC Program Memory".

- Press the INPUT key to enter the selected item in the combo box.

- Select the left-hand combo box with the EDIT key.

- Open the combo box with the RIGHT ARROW key and use the vertical ARROW keys to position the selection bar on the desired workpiece.

Fig. 6-13 Loading workpieces

- Press the INPUT key to enter the workpiece in the combo box.

- Press the COPY softkey.

The selected workpiece is loaded into the NCK program memory, i.e. the files of the workpiece are transferred according to the commands in the job list. The SELECT instructions in the job list are **not** processed.

Loading individual files   Proceed as follows to load an individual file of a workpiece:

- Select the right-hand combo box with the EDIT key.

- Open the combo box with the RIGHT ARROW key and use the vertical ARROW keys to position the selection bar on "NC Program Memory" or "NC System Data".

- Press the INPUT key to enter the selected item in the right-hand combo box.

- Select the left-hand combo box with the EDIT key.

- Open the combo box with the RIGHT ARROW key and use the vertical ARROW keys to position the selection bar on the workpiece which contains the desired file.

- Press the INPUT key to enter the selected item in the combo box.
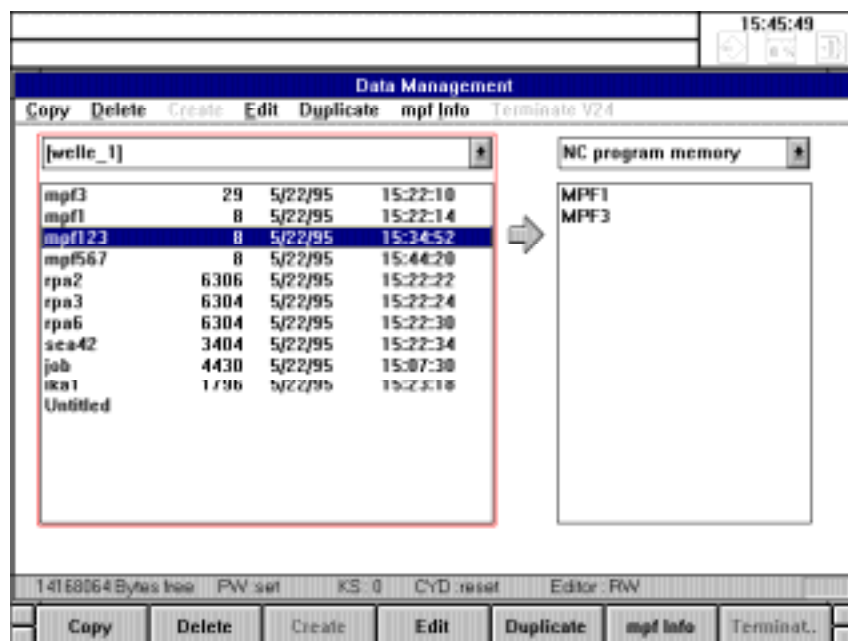
Fig. 6-14 Loading individual files

- Select the list box immediately underneath using the EDIT key.

- Position the selection bar on the desired file. Note the direction of the copy arrow.

- Press the COPY softkey.

The selected file is loaded into the NCK memory. If a file with the same name already exists in the NCK memory, a safety prompt appears.

6.2.5.2 Saving files

NCK files can be copied from the NCK memory onto the hard disk as follows:

- Select the left-hand combo box with the EDIT key.

- Open the combo box with the RIGHT ARROW key and use the vertical ARROW keys to position the selection bar on the workpiece in which the file is to be saved.

- Press the INPUT key to enter the selected item in the left-hand combo box.

- Select the right-hand combo box with the EDIT key.

- Open the combo box with the RIGHT ARROW key and use the vertical ARROW keys to position the selection bar on "NC Program Memory" or "NC System Data".
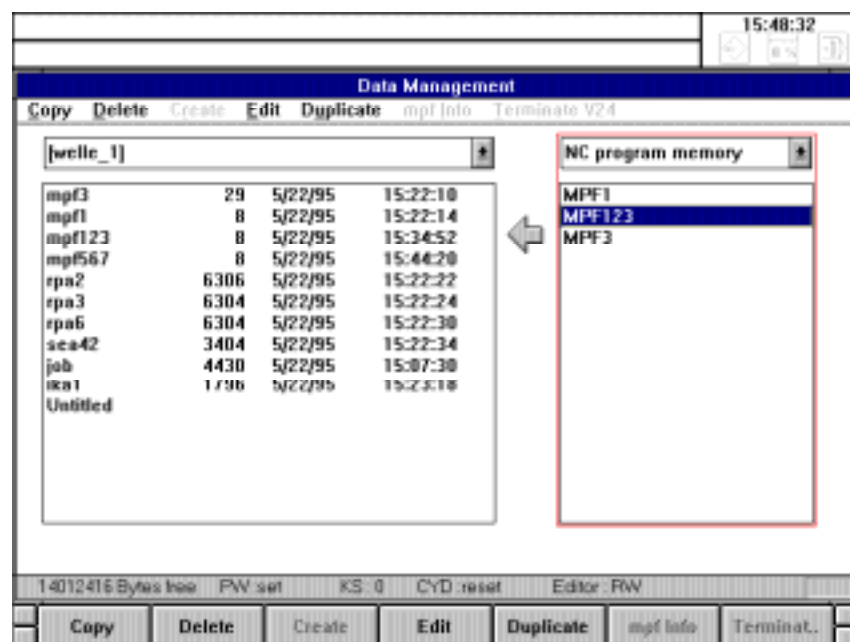
Fig. 6-15   Saving files on the hard disk

- Press the INPUT key to enter the item in the combo box.

- Select the list box immediately underneath using the EDIT key.

- Position the selection bar on the desired file. Note the direction of the copy arrow.

- Press the COPY softkey.

The selected file is copied from the NCK memory to the specified workpiece. If a file with the same name already exists, a dialog box appears.

6.2.6   Data communication between MMC and peripheral devices

The data manager can also be used to transfer data to or from a peripheral device. The device names defined as active in "V24 Configuration" are displayed in the right-hand combo box of the data manager and can be selected for the data transfer.

> **!**
>
> **Warning**
>
> The user guarantees that the data read in are free of viruses!
>
> The control is provided with a virus protection activated in the BIOS. It monitors the boot sector and the DOS File Allocation Table (FAT).

6.2.6.1   Output of workpieces

To transfer a complete workpiece to a peripheral device, proceed as follows:

- Select the right-hand combo box with the EDIT key.

- Open the combo box with the RIGHT ARROW key.

- Use the ARROW keys to position the selection bar on an active device name declared in "V24 Configuration".

- Press the INPUT key to enter the selected item in the right-hand combo box.

- Select the left-hand combo box with the EDIT key.

- Open the combo box with the RIGHT ARROW key and use the vertical ARROW keys to position the selection bar on the desired workpiece.

- Press the INPUT key to enter the selected workpiece in the combo box.
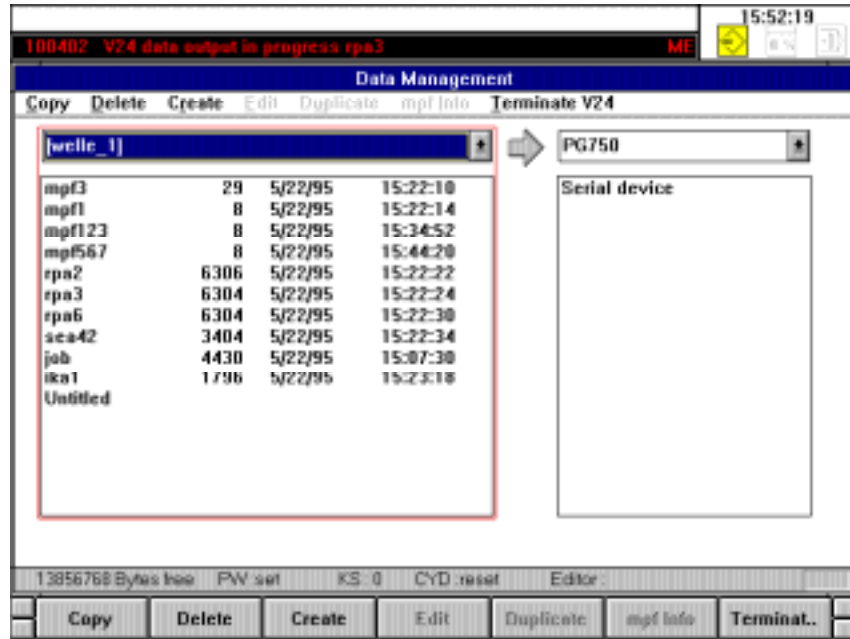
Fig. 6-16 Output of workpieces to the serial interface

- Press the COPY softkey.

The interface is initialized with the parameters specified in "V24 Configuration" and the output of the selected workpiece is started. All files of the workpiece are output to the RS232C (V.24) interface with no intermediate gap.

The V24 Data I/O icon blinks while the data output is taking place and the name of the file currently being output is displayed in the status line..

The **Cancel V24** menu item is also enabled. The data transmission can be cancelled at any time by pressing the appropriate softkey.

Note

Direct reading in/out of workpieces in the FlexOS file tree is not possible.

Remedy:

You can use the data manager to create any workpiece and then copy the workpiece data of the FlexOS workpiece into this workpiece. The data of the workpiece can then be output in the usual way via the V24.

Important

When transmitting data via the serial interface, the receiver must first be activated. Only then can the sending be started.

6.2.6.2   Output of individual files

To transfer an individual file of a workpiece to a peripheral device via the RS232C (V24) interface, proceed as follows:

- Select the right-hand combo box with the EDIT key.

- Open the combo box with the RIGHT ARROW key.

- Use the ARROW keys to position the selection bar on an active device name declared in "V24 Configuration".

- Press the INPUT key to enter the selected item in the right-hand combo box.

- Select the left-hand combo box with the EDIT key.

- Open the combo box with the RIGHT ARROW key and use the vertical ARROW keys to position the selection bar on the desired workpiece.
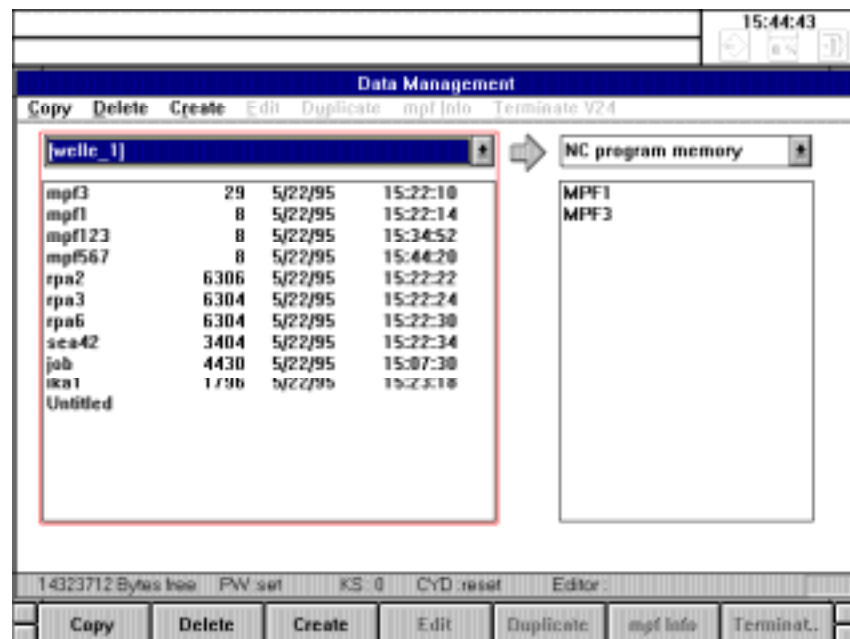
- Press the INPUT key to enter the selected workpiece in the combo box.

- Select the list box immediately underneath using the EDIT key.

- Position the selection bar on the desired file.

- Press the COPY softkey.

The data output is started, i.e. the V.24 interface is initialized and the selected file is activated.

The Cancel V24 menu item is also enabled. The data transmission can be cancelled at any time by pressing the appropriate softkey.

6.2.6.3   Input of files

The data manager can be used to transfer NCK files to the hard disk via the V24 interface. The files can be saved in a selected workpiece or in the Clipboard:

- Select the left-hand combo box with the EDIT key.

- Open the combo box and position the selection bar on the desired workpiece or on the "Clipboard" item.

- Press the INPUT key to enter the selected workpiece or the "Clipboard" item in the left-hand combo box.

- Select the right-hand combo box with the EDIT key.

- Open the combo box and position the selection bar on an active device name declared in "V24 Configuration".

- Press the INPUT key to enter the device name in the combo box. Note the direction of the copy arrow.
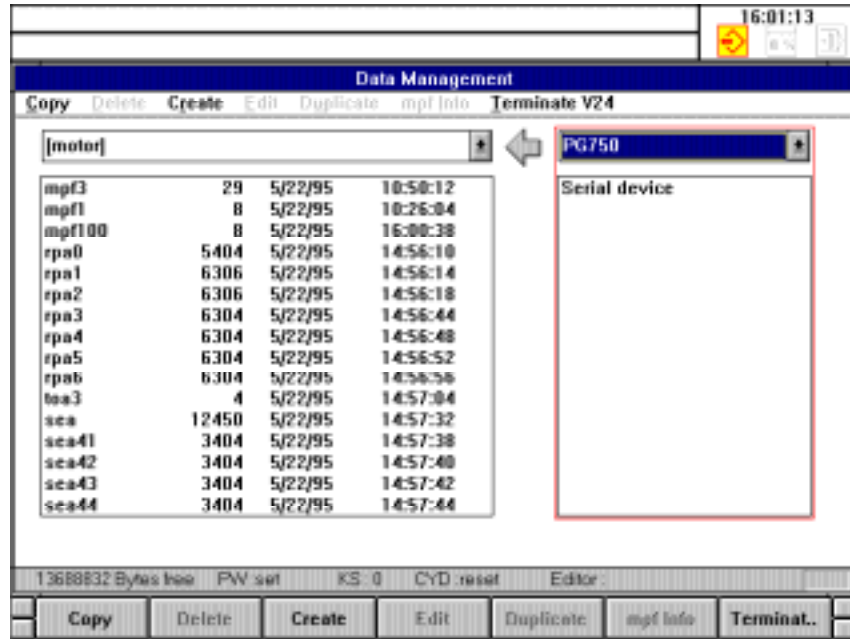
Fig. 6-17 Input of files

• Press the COPY softkey.

The V24 interface is initialized and the data input is started. The files being read in are stored in the selected workpiece or in the Clipboard. The name of the file being read in is displayed in the message line.

When data are read in in punchtape format, the punchtape identifiers, such as %MPFxxxx are deleted and converted into the file name.

The **Terminate V24** menu item is also enabled. The data transmission can be cancelled at any time by pressing the appropriate softkey.

Notes

The following points should be noted with reference to data input:

• Password-protected data, such as IKA data, GIA data and machine data, are not copied into the specified workpiece, but are always stored in the Clipboard.

• Files of the type UMS cannot be read in using the data manager; the data transmission is cancelled with the error message "Illegal object type". UMS files can be read in using the PCIN data transfer program.

• The machine manufacturer can choose between three different modes:

  − Overwrite mode: existing files on the destination directory are always overwritten without prompting the user for confirmation.

  − Append mode: files which are read in are always appended onto existing files.
    If the program being read in is of the type MPF or SPF, it is not possible to append the files. Instead, the "V24 data input" dialog box appears and the "overwrite" and "skip" modes can be selected.

  – User dialog: the "V24 data input" dialog box prompts you to choose
    between the append, overwrite or skip modes.
    If the program being read in is of the type MPF or SPF, you can only
    select the "overwrite" or "skip" buttons. It is not possible to "append"
    part programs.
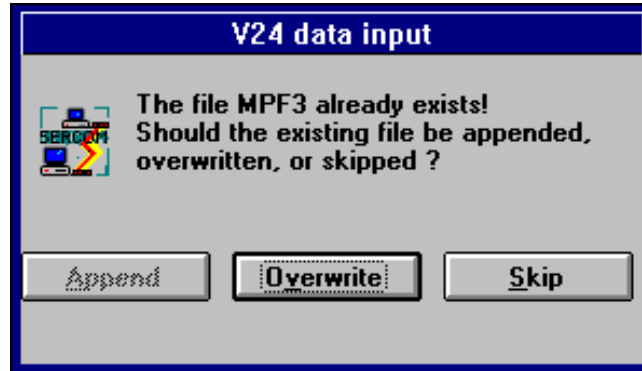
Fig. 6-18   Append/overwrite

Please refer to the machine manufacturer's documentation for further
information.

6.2.6.4 Transferring data to the FD-E2 diskette drive

The procedure for transferring data between the hard disk and the connected FD-E2 diskette drive is identical to data transfer between the MMC and a peripheral device.

The user must first select the device type "diskette drive" in the combo box.
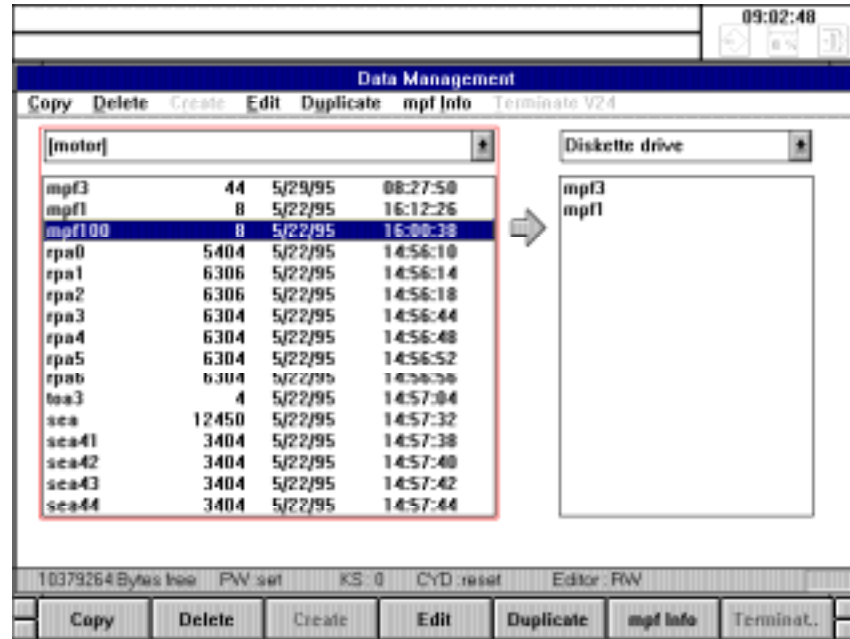
Fig. 6-19 Transferring data to the FD-E2 diskette drive

Note

Workpiece-oriented data management is not supported on a diskette drive selected via the right-hand combo box. You can only transfer individual files between the hard disk and the diskette drive selected in this way.

Up to 112 files can be stored on a 3 1/2" DD diskette and up to 224 files can be stored on a 3 1/2" HD diskette.

Only formatted diskettes can be used to transfer data with the aid of the data manager. Unformatted disks must be first formatted using the MS-Windows File Manager.
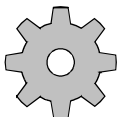
Machine manufacturer

A diskette drive can also be integrated into the data management as a directory for workpieces. In this case, the root directory of the diskette is used as a container for further workpieces.

6.2.7 Description of the WEdit editor

Overview
WEdit is a convenient text editor which can be used to create and edit MPFxxxx, SPFxxxx and TOA files, etc..

General functions
- In **WEdit**, commands are selected from menus and information and settings entered in dialog boxes.

- You can select and reposition, copy or delete blocks of text. You can use the Windows Clipboard to exchange information with other applications.

- Sections of text can be searched and replaced with others.

- You can print files or selected areas of files.

- The status line of WEdit displays the mode (overwrite or insert mode) and the number of the line in which the cursor is located.

NC-specific functions
- Block numbers can be automatically generated in the part program.

- Part programs in the NCK memory can be edited as well as files on the hard disk or other network drives of the MMC.

- The modification of files can be disabled individually using the keyswitch. Files can only be edited if the write access is enabled.

- Cycles in the NC program memory can only be edited when the cycle disable is deactivated.

Other features
- The maximum length of a line in a file is 255 characters (= the maximum length of an NC block). Lines exceeding this length are cut off when the file is read in.

- **WEdit** is capable of editing files up to a length of 32,000 lines.

- **WEdit** can be loaded twice for the editing of two files in parallel.

6.2.7.1 Starting the WEdit editor

The **WEdit** editor can be used to edit NCK files on hard disk and part programs in the NCK memory:

- Use the DATA AREA key to select Area Switchover.

- Select the **Data Management** command from the **Programming** menu.
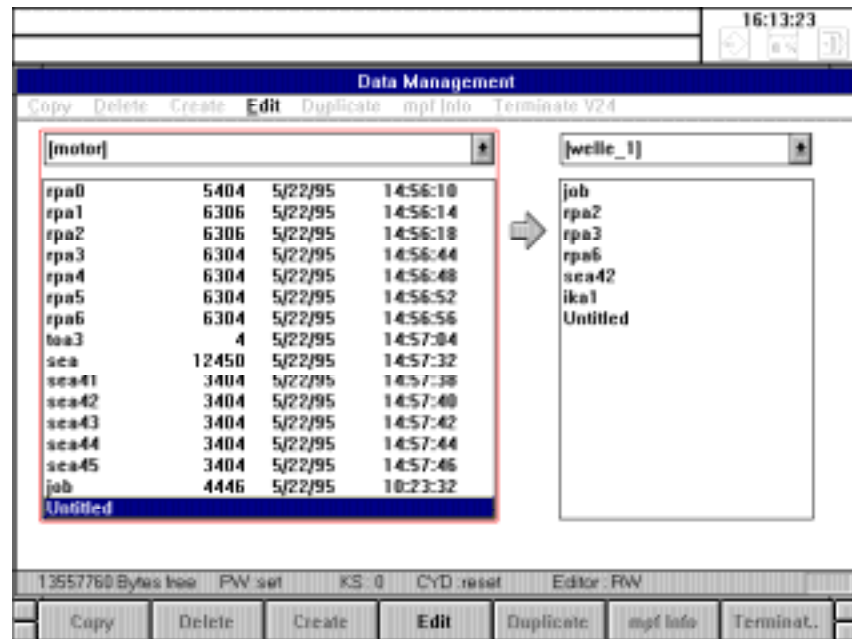
Fig. 6-20 **WEdit** NC editor

- Use the EDIT and ARROW keys to select the file to be edited. To create a new file, select the file name "untitled".

- Selecting the EDIT softkey starts **WEdit** and opens the selected file.
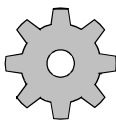
Machine manufacturer

Cycles on the hard disk and in the NCK memory can only be edited if the cycle disable is not active.
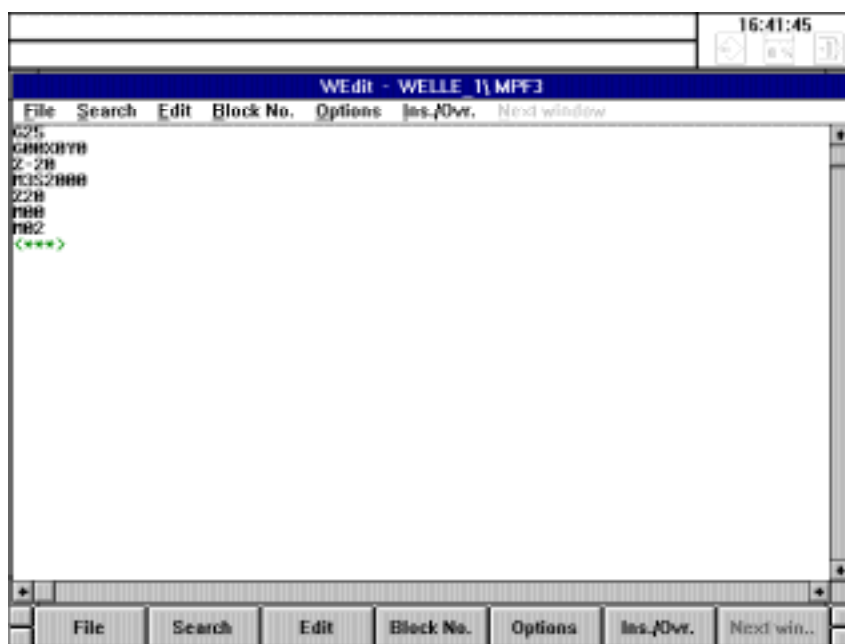
Fig. 6-21 **WE**dit in the Programming area

- The following information is displayed in the title bar:

  - Application name (always **WEdit**)

  - NC: the file is located in the NC memory. When a file is edited on the hard disk, no entry appears at this point.

  - File name (without path) or <untitled> when a new file is created.

  - <<RO>: the file can only be read (read only). No entry appears at this point when the file is read/write.

- The input cursor | or ■ identifies the position where entered text appears. When **WEdit** is started or an existing file is opened, the input cursor appears in the top left-hand corner of the window.

  In addition, the status line of **WEdit** displays in which line the input cursor (line=...) is located.

- The style of the input cursor identifies the current mode: in insert mode, the input cursor appears as a blinking line | ; in overwrite mode it appears as a blinking square ■. You can switch between insert and overwrite mode by selecting the command INSERT/OVERWRITE from the **WEdit** menu bar.

  In addition, the insert/overwrite mode is displayed in the status line of **WEdit**.

- The end mark <***> identifies the end of the file. When a new file is created, the end mark appears directly underneath the insertion point.

6.2.7.2   Key functions

Moving the cursor
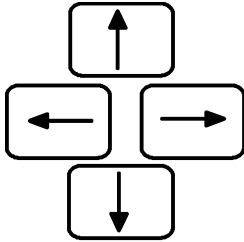
You move the input cursor through one character or one line by pressing one of the ARROW keys.
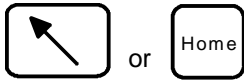
Puts the input cursor at the beginning of a line.
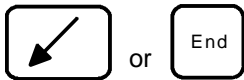
Puts the input cursor at the end of the line.

PAGE UP/DOWN: Pressing these keys scrolls the screen one page up/down.
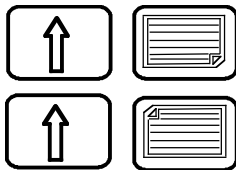
SHIFT + PAGE UP/DOWN: Puts the input cursor at the beginning/end of a file.

Deleting a character

The CANCEL key deletes the character underneath the input cursor. Identical to the <DELETE> key on the MF2 keyboard.
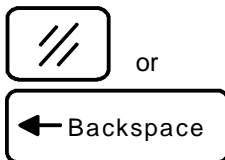
The CLEAR key deletes the character to the immediate left of the input cursor. Identical to the <BACKSPACE> key on the MF2 keyboard.

Terminating a line

or

Enter
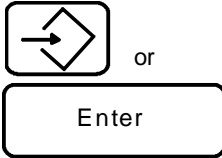
INPUT key. Identical to the <ENTER> key on the MF2 keyboard. In the editor, the INPUT key is used to terminate a line.

Moving within dialog boxes

or

Tabulator
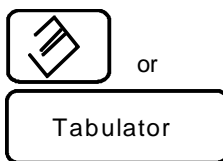
In dialog boxes, the EDIT key can be used to switch from one input box to the next. Identical to the <TAB> key on the MF2 keyboard.

6.2.7.3 Editing text

Selecting text

You select a block of text by setting a start and end mark as follows:

- Position the input cursor on the first character to be selected.

- Choose **Select** from the **Edit** menu.

- Position the cursor on the last character to be selected.

- Choose **Select** from the **Edit** menu.

The selected text block is displayed inversely and can be edited using the commands in the **Edit** menu.

When a block is selected, the menu items **Delete Block** and **Copy Block**, which are normally "gray" (deactivated), are available for selection.

The start mark may be set before the end mark in the text.

Copying text

You can use the commands **Copy** and **Paste** to copy a section of text:

- Select the section of text to be copied.

- Choose **Copy** from the **Edit** menu. This command copies the text block into the Windows Clipboard. The start and end marks of the block are cleared.

- Position the input cursor at the point where you want to insert the text.

- Choose the **Paste** command from the **Edit** menu to insert the text at this point.

Note

The Clipboard only contains the last text to be copied or deleted. If you want to paste a text block several times, you should not delete or copy any other text into the Clipboard between the paste operations.

Moving text

To move a section of text, the procedure is similar to copying, except that the text is deleted from the original position:

- Select the section of text to be moved..

- Select **Delete** from the Edit menu. The selected text block is transferred to the Clipboard. The start and end mark of the block are cleared.

- Position the cursor at the point where you want to insert the text.

- Use the **Paste** command in the **Edit** menu to insert the section of text from the Clipboard to the chosen position.

Deleting text

You can delete sections of text as follows:

- Select the section of text to be deleted.

- Select **Delete** from the **Edit** menu. The selected text block is transferred to the Clipboard.

Finding text

The automatic find is an invaluable aid when working with longer files. You can search for individual characters, words, sentences or complete sections of text:

- Select **Find** from the **Find** menu. The "Find" dialog box appears.
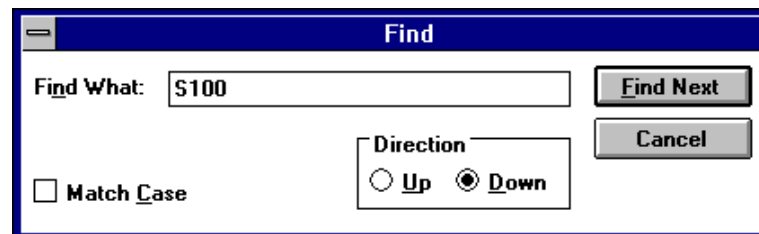
Fig. 6-22 "Find" dialog box

- Enter the search key in the edit box.

- Select the "Find Next" button to start the search. **WEdit** selects the first term matching the text entered in the Find dialog box.

- Select the "Find Next" button to continue the search for the next match.

You can switch between the individual input boxes in the dialog box using the EDIT key. Alternatively, you can press the underlined letter (HOTKEY), when the cursor is outside the text boxes, to jump to the desired input box. You can select individual check boxes using the SPACE key.

Finding and replacing text

With the Find and Replace function you can not only find text, but you can replace the found text with new text:

- Position the input cursor at the starting point for the replacement of text.

- Select **Replace** from the **Find** menu.
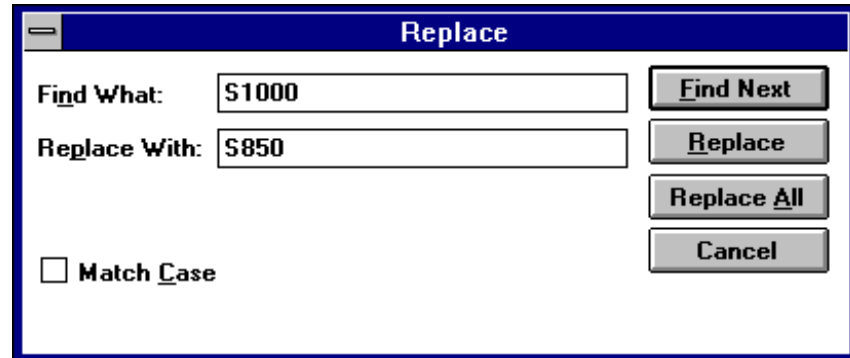
Fig. 6-23 "Replace" dialog box

- Enter the text to be found in the "Find What" text box. Enter the text which is to replace the found text in the "Replace With" box.

- If you select the "Replace All" button, all text matches found in the entire text are replaced without prompting.

- If you start the find/replace process by selecting the "Find Next" button, the first text found to match the original text is selected. The text can be replaced by selecting the "Replace" button.

- Select the "Find Next" button to continue the search for the next match.

- If no match is found, a dialog box appears.

Go to

You can jump to a specific line using the "Go to" dialog box.
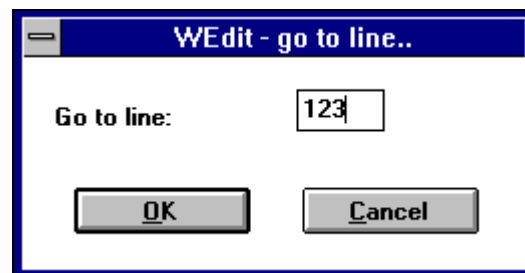
Fig. 6-24 Dialog box: "Go to"

- Enter the number of the line you are looking for in the upper input field and press the softkey OK. The cursor is positioned at the beginning of the line you are looking for.

6.2.7.4   File management

Opening and creating
files
(see: Starting the **WEdit** editor)

Saving files

When you have created a new file or made changes to an existing file, you can save the data using the commands **Save** and **Save As...** from the **File** menu. It is advisable to save files regularly during editing, in order to minimize the loss of data in the event of a power failure or other fault.

To save a new file (name: "untitled") or to save an edited file under another name:

- Select **Save As...** from the **File** menu if you want to store a new file (Name: "unknown") or an existing file under a different name. The "WEdit - Save As..." dialog box appears.
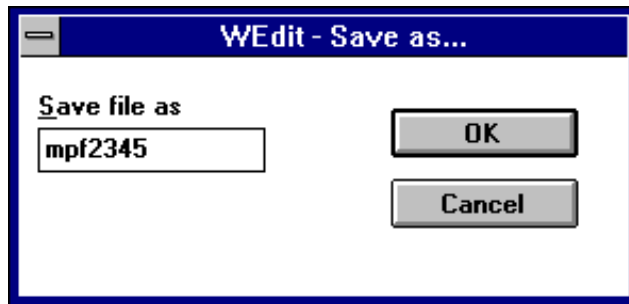
Fig 6-25   "Save As..." dialog box...

- Enter a valid DOS name for a part program (mpf... or spf...) in the file name box (**8** characters**.3** characters). The file is only displayed in the left-hand list field of the data manager if a valid NCK name has been entered. Otherwise the file is merely displayed in the right-hand list field and can therefore not be transferred to the NCK memory.

- Select the "OK" button.

To save an existing file:

- Select **Save** from the **File** menu.

Save and load

The function **Save and load** can be applied to a part program file (MPF or SPF file) being executed which is not located in the NC program memory and the contents of which have been modified.

The modified file is both stored in the mass storage and transferred to the NC part program memory.

- Select **Save and load** from the **File** menu.

Printing files

You can choose the **Print** command from the **File** menu to print all or part of an open file. This menu item is only activated when a printer is set up:

- Select the block of text to be printed if you only want to print part of the file.

- Choose **Print** from the **File** menu. The "Print File" dialog box appears.
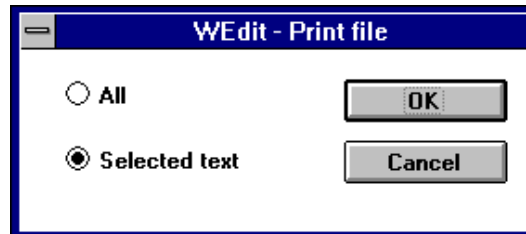
Fig. 6-26   **WEdit - Print file"** dialog box

- Select the option "Selected Area" if you only want to print the selected text, select "All" to print the complete file.

- Select the "OK" button.

Note

The above dialog box only appears if a block of text has been selected. If no text is selected, the loaded file is simply output to the printer interface when the Print command is chosen.

Backup copies

In the **WEdit - General Options** dialog box you can define whether **WEdit** automatically creates a backup copy when saving a file. When the option is selected, a backup copy is created every time an edited file is saved.

This backup copy can be loaded in the editor in place of the actual file:

- Select the **Fetch Backup Copy** command from the **File** menu.

You can also delete the backup copy:

- Select the **Delete Backup Copy** command from the **File** menu..
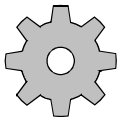
Machine manufacturer

The machine manufacturer can configure whether a dialog box appears or not when a backup copy already exists.

6.2.7.5 Other functions

Working with block
numbers

You can insert NC block numbers at the current cursor position in the text:

• Select the **Block Number** menu item by pressing the appropriate softkey.

The parameters of the block numbers can be defined in the **Block Number Options** dialog box (see below).

Options

Certain settings can be made for **WEdit** in the **Options** menu.

General options

Here, settings are defined for backup copies and for starting in insert mode:

• Select the **General** command from the **Options** menu. The "WEdit - General Options" dialog box appears.
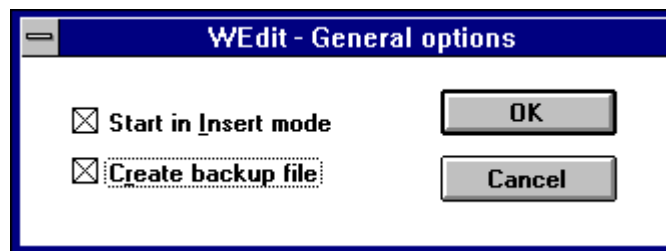
Fig. 6-27 **"WEdit - General Options"** dialog box

• You can use the ARROW keys to switch the input focus between the two boxes. You can select each of the check boxes using the SPACE key.

• Select the "OK" button to save the option settings.

Start in Insert mode

When this option is selected, **WEdit** automatically starts in insert mode. The input cursor appears as a blinking line | .

If the option is not selected, **WEdit** starts in overwrite mode. The input cursor now takes the form of a blinking square ■.

Create backup file

(see backup copies in the section entitled "File management".)

Block number, options	This dialog box is used to define the settings for the generation of block numbers:

- Select the **Block Number** command from the **Options** menu. The "WEdit - Block number, options" dialog box appears.
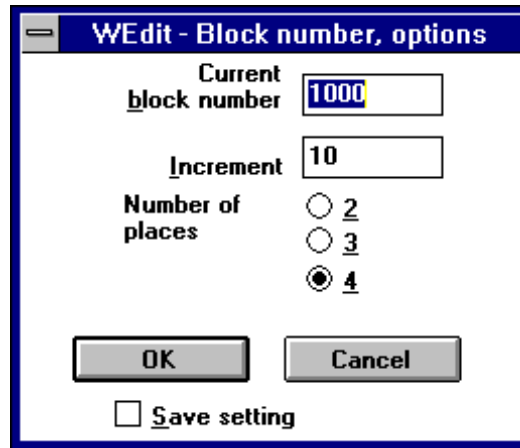
Fig. 6-28 "WEdit - Block number, options" dialog box

- You can use the EDIT key to switch the focus between the input boxes.
- Select the "OK" button to save the option settings.

Current block number	The number in this input box is entered at the current cursor position when the **Block Number** menu item is activated.
Increment	Specifies the value by which the block number is automatically incremented.
Number of places	Specifies the number of digits used for the block number.
Save setting	When this option is selected, the settings are saved. **WEdit** subsequently automatically starts with these settings.
Ins./Ovr.	This command can be used to switch between insert and overwrite mode. The currently selected mode is apparent from the cursor style (see "Starting in insert mode").

Next window   The **WEdit** editor can be started twice from the data manager. You can
               therefore edit two files simultaneously and exchange text via the MS-Windows
               Clipboard. The **Next Window** command in the menu bar can be used to switch
               the input focus between the two files.
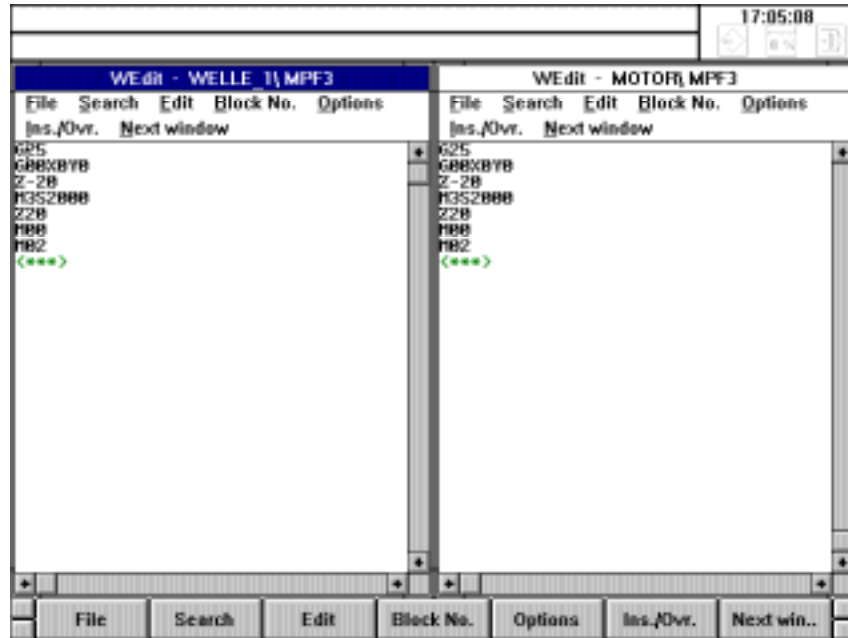
Fig. 6-29 **WEdit** loaded twice

Note

If you try to edit a third file in parallel, the two open edit windows of **WEdit**
are automatically put to the foreground and the corresponding message
"Already 2 files opened ...." is displayed.

One of the two editors must be closed before another file can be edited.

6FC5198-6AA60-0BP0

6.3 Programming in the NCK memory area
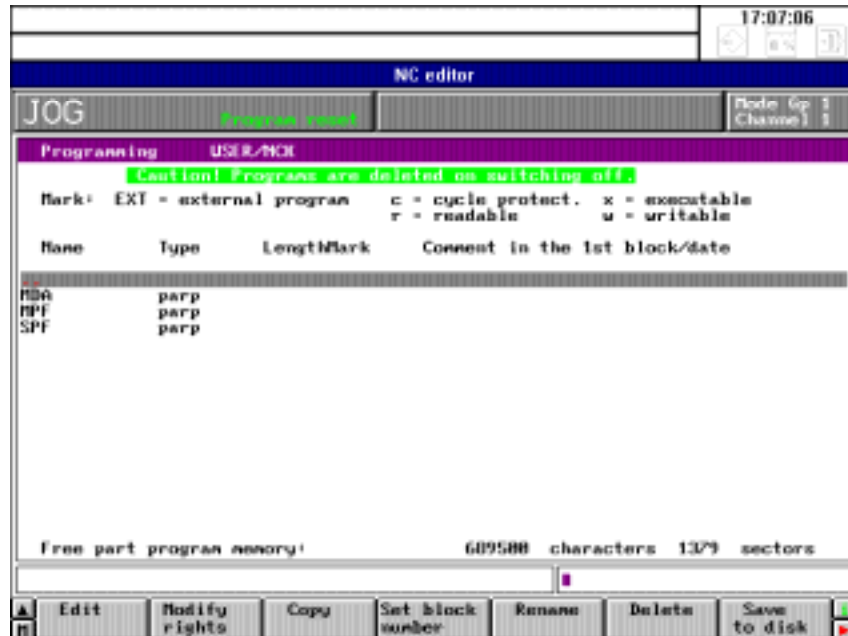
Fig. 6-30 Basic display: Edit NC

In the **Programming/Edit NC** area you can

- Edit part programs in DIN format directly in the processing memory, i.e. the part programs edited here must be saved to the hard disk if they are to be retained in the control after switching it off.

- Display overviews of main programs, subroutines and MDA programs in the part program memory of the NCK.

If the letter "i" is displayed in the 1st softkey window (left) notes relating the selected area appear after pressing the HELP key.

Important

The part programs edited here must be backed up on the hard disk as they are lost when the control is switched off.

6.3.1   Select program

Use the data selector to select the directory for **subareas** main programs, subroutines and MDA programs. The following display appears, for example, when you select area: Main programs:
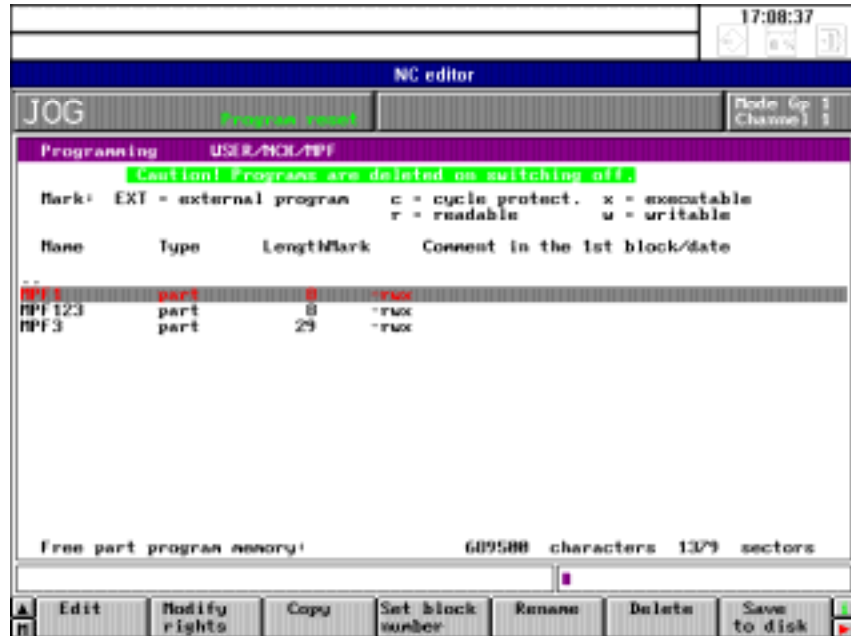
Fig. 6-31   Overview Main programs

Edit

Place the cursor on the program that you wish to edit and press the softkey EDIT. The part program is called up and can be edited.

If you wish to enter a new program, select MPF or SPF and then change to the keyboard and enter the desired program number (as %, L, MPF, SPF).

Edit

Then press the softkey EDIT.
You can now edit the program (e.g. MPF100).

6.3.2   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").
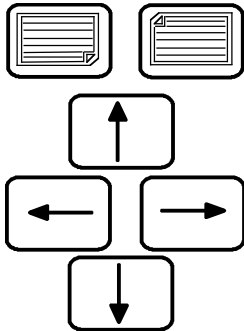
Using the page up/down and the cursor keys, you can position the cursor at any position of a part program already stored. Pressing the key combination SHIFT + ARROW or SHIFT + PAGE moves the cursor to the start or end of the line or program file.
If you press RIGHT ARROW again at the end of a block, 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 CLEAR key.

Press CANCEL to delete the character directly under the cursor. The result of a text search is deleted completely.

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.
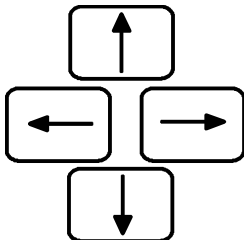
Use the arrow keys to select the desired text. You can press the "Page Up" or "Page Down" keys to select larger areas of text.
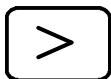
Press the ETC key to extend the softkey bar.
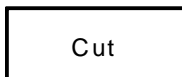
Press the CUT softkey to delete the selected text.

SINUMERIK 840C, OEM Version for Windows (BA)

Note

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.
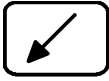
You can activate the cursor in the input line. The cursor is then inactive in the program, i.e. you can only enter characters in the input line (e.g. for the SEARCH function).

Pressing the key again deselects the activation.
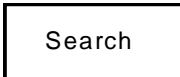
Search

Press the SEARCH softkey to search for a block or a character string in the selected program. The message

"Please enter search text".

appears.

...
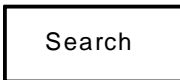
Search

Use the alphanumeric keyboard to enter the search term and then press the SEARCH softkey.
The cursor is positioned before the search term. The search is performed from the cursor position towards the end of the program.

Note

The search function is used to find character strings. This means that if you enter "N1" as the search text, the cursor finds the character strings **"N10"**, **"N11"**, **"N100"** etc. Spaces are also treated as characters. For example, if you enter the search term " N1 ⌴ "(⌴ = space), the cursor will only find the block numbers "N1 ⌴ ".

When you press the SEARCH softkey again, the old search text appears in the input line. You can then perform the search again.
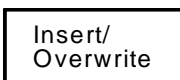
Insert/
Overwrite

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 a different color to the rest of the text.

Overwrite: The character is displayed on a black background, the character itself is displayed in inverse format.

The active mode is displayed in the header line.

You can enter characters directly at the cursor position. The new characters are inserted where the existing characters are overwritten according to the selected mode.

| Generate block number |

This softkey enters the block number from the cursor position. When the SELECT program function is used to choose the name of the program, the block number is taken from the default setting in the "Editor settings" input screen. The next time you press the GENERATE BLOCK NUMBER softkey, the block number is generated from the entry in the "Block number difference" field.

| Save to disk |

The softkey SAVE TO DISK and (in the softkey bar extension with the ETC key)

| Save under |

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

This function is described in the Section Machine (SAVING PROGRAMS).

Note

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, duplicate the part program, i.e. you insert it on the hard disk under another name (duplicate function).

Instead of this, you can change the name of the part program in the NC part program memory before saving in the overview displays main progr./subprogr. by means of the RENAME function.

| Tool path simulation |

When you press this softkey, the dialog message "Option not available" appears.

6.3.3   Editing a new NC program

A presetting can be made before editing the program.

Set
block number

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

The following display appears when you press this softkey:
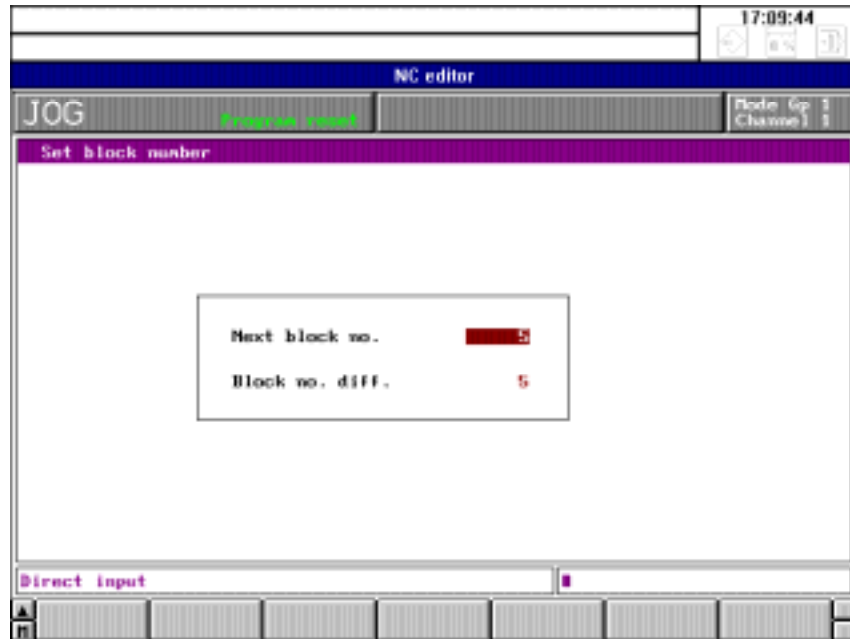
Fig. 6-32 Block number setting

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 ....   L$_F$
N10 ....   L$_F$
N15 ....   L$_F$

A maximum of 4 numbers can be input.

Note

The block settings made here are also effective in TEACH IN mode.

Edit

This calls the program.

Generate
block number

Press this softkey to generate a block number. The block number is inserted at
the cursor position according to the presetting.

...

Enter the program blocks using the alphanumerical keypad.

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 again, the system adds the block number difference to the block number and inserts the result at the cursor position.

Save to disk

The softkey SAVE TO DISK and

Save under

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

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)

Important

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.

Note

The workpiece STANDARD is set up 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 part programs or NCK data.
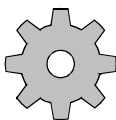
Machine manufacturer

The machine manufacturer can specify 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.

6.3.4   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.
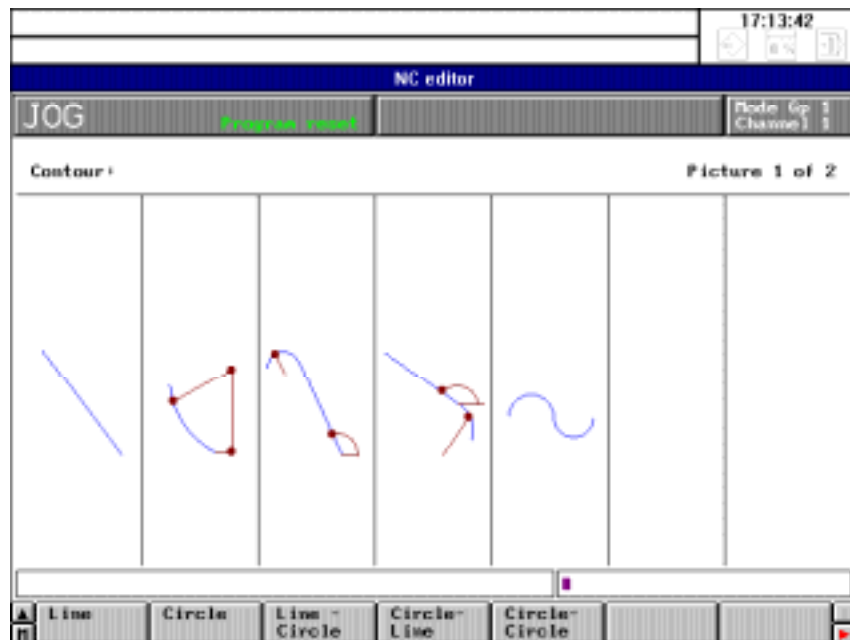
Fig. 6-33 Contour definition programming in the PROGRAMMING area

Selecting the contour
definition
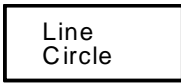
Line Circle

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.
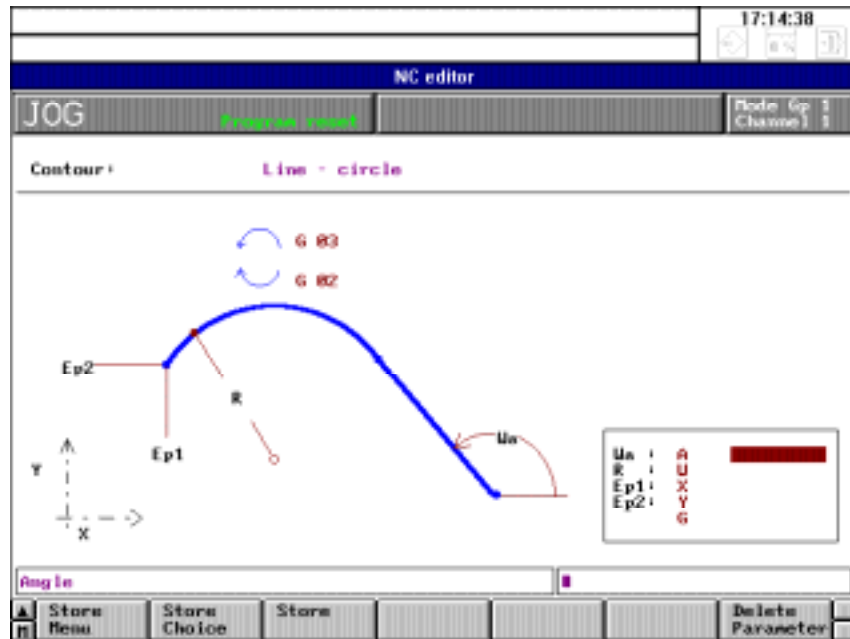
Fig. 6-34 Line - circle geometric contour programming

| ↑ |
| ← → |
| ↓ |
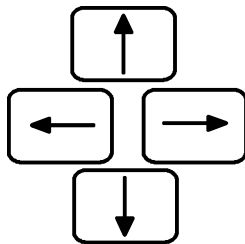
Using the cursor keys, you can choose any input field, for example if you wish
to subsequently modify geometric values for the selected contour.

Note

$$\begin{Bmatrix} X \\ Z \end{Bmatrix}$$

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.

6.3.5   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

6.3.6   Plane

Plane

Press the PLANE softkey.
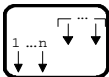
Switch over to the desired channel.

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.

6.3.7   Program management

6.3.7.1   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)
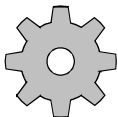
Machine manufacturer

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 manufacturer's 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 with the cursor.

| + ' | 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.

Note

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.
With cycles, the rights can be indicated in the LOADCYC instruction.

6.3.7.2 Copy program

**Operating
sequence**

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

Copy

The following screen form appears:

Fig. 6-35   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 MPF100 is to be copied to SPF1111.

The cursor is placed on directory MPF. This selects the directory and the new
program number can be entered.
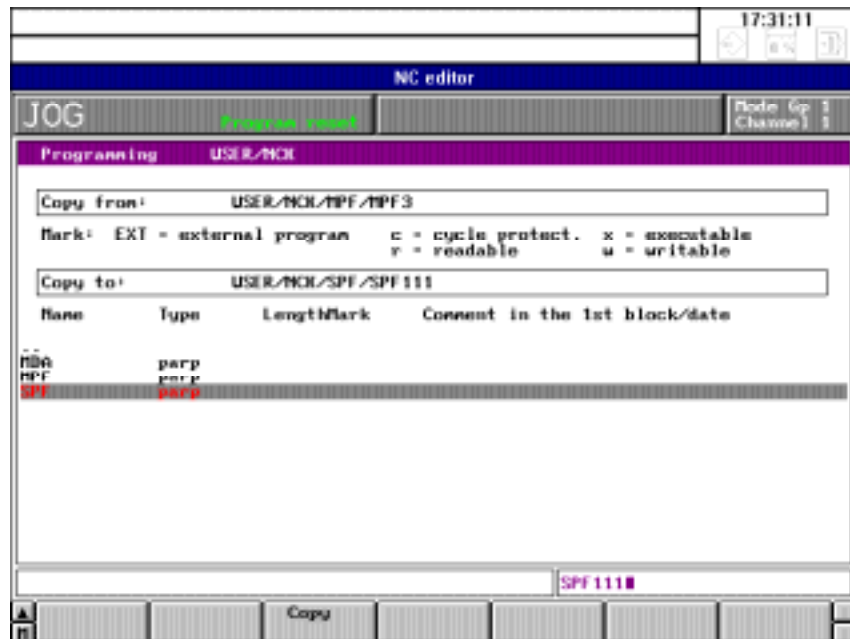
Fig. 6-36 Copy MPF/SPF 02

Copy

Now press the softkey COPY - this copies program MPF100 to SPF1111.

The basic display for editing automatically reappears. SPF1111 is now listed in the program overview.

Note

A main program can also be copied to a subroutine (and vice versa). The different program end (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).

Note

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.

6.3.7.3 Rename program

**Operating
sequence**

Select a part program (MPF/SPF) with the cursor.

Rename

Now enter the new program number (e.g. MPF222) and press the softkey
RENAME. The program is displayed under a new name.

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.

6.3.7.4 Delete program

Deleting with the data selector:

**Operating
sequence**

Select the program that you wish to delete with the cursor.
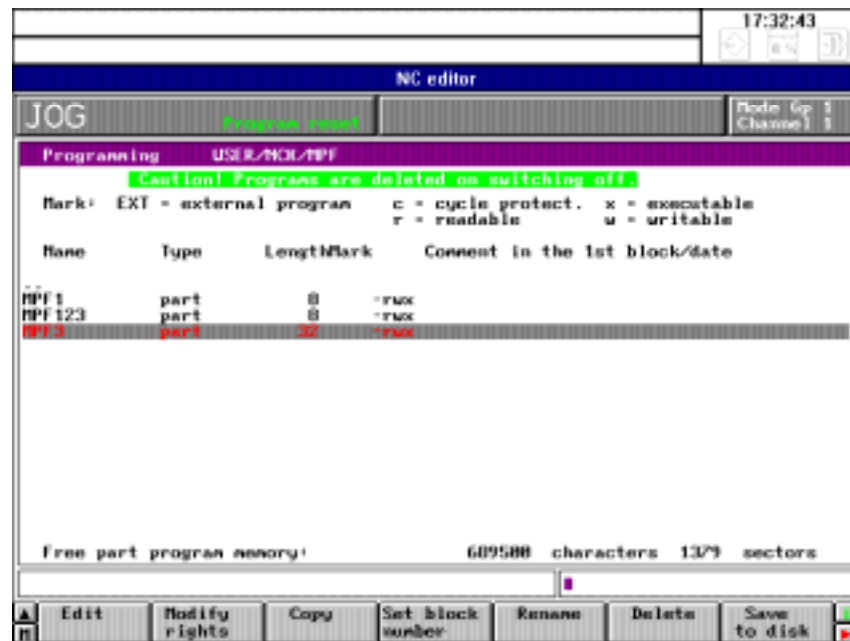
Fig. 6-37 Deleting NCK programs

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 | ... |

| Delete |

The program with the number entered is deleted by the control.

or

| %LF | ... |

| * |

| Delete |

all of the programs with identifiers before the "*" are deleted
e.g. MPF1*
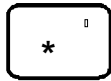
If you enter MPF* or SPF* all of the main programs or subroutines are deleted.

or

| %LF | ... |

| ⇑ | 6 ? |

| Delete |

you can use the question mark to represent an individual character in the
program name.
e.g. MPF?

All of the main programs with a single-digit program number are deleted, i.e.
MPF0 to MPF9
MPF2??

The programs MPF 200 to 299 are deleted.

You can also delete areas of a program by entering the start and end of the area
and pressing the delete softkey. For example, if you enter

%700 ⌞⌟ %710 or
%700=%710

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

 Note

 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.

6.3.8   Move cycles

Machine manufacturer

The cycles can be protected individually on the MMC. On the NCK side, they are normally dissolved for non-authorized editing from the PLC.

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 to the NCK memory.(see the Section "Changing the access rights").

You can now rename, copy and edit the cycle in the NCK memory just like a normal subroutine. You can assign the cycle as a subroutine to a workpiece. (according to the machine manufacturer's planning).

The cycle disable must be configured by the machine manufacturer, i.e. only cycles created by the machine manufacturer can be protected by a cycle disable.■

Services

7

7.1 Selecting Parameter Assignment V24

First press the DATA AREA key to activate the **Area Switchover**.
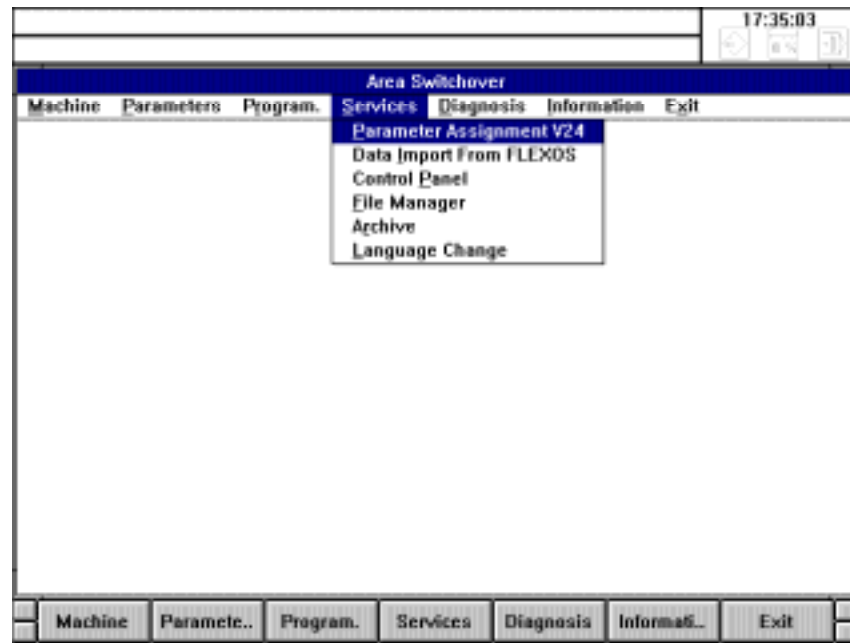
Fig. 7-1 Area Switchover

When "Area Switchover" is selected:

- First press the SERVICES softkey to open the services menu.

- In the open menu select the following HOTKEY to select the user area:

  - "P" Parameter Assignment V24

Note

The other areas are described in the documentation

SINUMERIK 840 C
Start-Up Difference Description
OEM Version for Windows

7.2 Description of parameters

You can output files to an external data storage device via the RS232C (V24) interface. The RS232C (V24) interface and the data storage device must be matched.

The SINUMERIK 840C provides input screen forms in which you can define the specific parameters for the serial interfaces. You can store various configurations on the control as "devices" and activate these devices as the need arises.

When you select the "Parameter Assignment V24" menu item, the following dialog box appears.
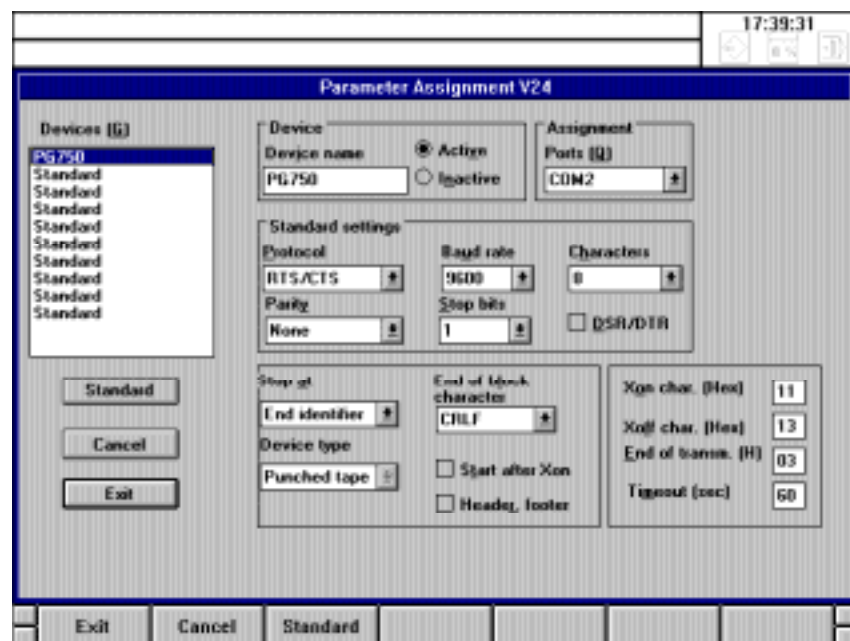
Fig. 7-2   "Parameter Assignment V24" dialog box

All of the interface settings are shown in the "Devices" list box. When the dialog window is opened, the data record of the first device is displayed.

Up to 10 settings can be saved.

The active devices are also displayed in the right-hand combo box of the data manager (section 6.2.6) and can be selected for data output.

In the "Parameter Assignment V24" dialog box, you can:

- Activate and deactivate devices
- Change the configuration of existing devices.

The various input boxes can be selected with the EDIT key. Option buttons can be selected with the ARROW keys and check boxes with the SPACE key.

Devices

10 interface settings are always displayed in the "Devices" list box. When the control is started up for the first time, 10 possible devices are provided with default settings. The settings can be selected and their names and parameters changed.

Device name

You can name an interface setting in this text box. The interface setting is displayed under this name in the "Devices" list box and in the right-hand combo box of the data manager (if active).

Active/Inactive

These option buttons can be used to declare devices as active or inactive. Only the active devices are displayed in the right-hand combo box of the data manager and can be selected for data output.

Port

You can assign the device to a serial port in this combo box.

The first serial port (COM1) is permanently assigned for connection of the NC operator panel and cannot therefore be used for data transfer. Only COM2 is available for data output in the present software version.

Protocol

In this combo box you can choose between three protocols:

- RTS/CTS: With this protocol, the flow of data between the two devices is controlled by additional hardware signals in the transmission cable. These control lines allow the receiver to request the transmitter to stop the data transmission if the receive buffer is in danger of overflowing.

- Xon/Xoff: When this protocol is selected, the flow of data is controlled using control characters. The receiver sends an "Xon" character to signal the transmitter that it is ready to receive. If the receive buffer is full, the receiver can stop the transmitter with the "Xoff" character.

- None: Selecting "None" informs the SINUMERIK 840C that the peripheral device does not use a specific protocol for controlling the flow of data.

The RTS/CTS protocol is generally used for data transmission between the SINUMERIK 840C and a peripheral computer.

Baud rate

The baud rate sets the "speed: bit/s" with which the individual bits are transmitted on the line. The higher the baud rate, the faster the bits are transmitted. If a large number of transmission errors occur, you should reduce the baud rate.

The transmitter and receiver **must** be set to the same baud rate.

Character length

You can choose between 7 or 8 bits for the character length setting. 7 bits are sufficient for the transfer of NCK files in punchtape format, binary files require 8 bits.

Parity

The transmitter can append an extra parity bit to the character to be transmitted. The receiver uses the parity bit to test for possible transmission errors.

The total number of "ones" in the character must be even with positive parity and odd with negative parity.

Stop bits

These bits define the amount of time the transmitter waits before transmitting a new character on the line. Stop bits are, in fact, a time interval rather than actual bits.

The number of stop bits can be set to 1 for modern devices (PCs or similar devices).

DSR/DTR

This check box defines whether or not pin 6, on which the peripheral device signals that it is ready, is evaluated.

The standard setting for this check box is deselected.

Stop at

In this combo box you can choose the character at which the SINUMERIK 840C stops receiving data:

- M02/M03: The SINUMERIK 840C stops data transmission at every program end character it receives.

- End character: The SINUMERIK 840C stops at the character set in the "End of Transmission" input box.

End of block character

Here you can choose between the following two options:

- CRLF: The NC terminates a block in the data output with CR LF. This is the standard end of block identifier used in the PC sector. You should use this end identifier if you want to edit the output files externally with a PC word processing program.

- The NC outputs an LF CR CR at the end of a block. This end identifier is used by older types of printer which do not have an integrated receive buffer and which therefore have to output the received characters directly on paper.

Device type

The following device type is presently supported:

- Punched tape: During data output, each file is provided with a special header (% identifier) and opening and closing characters. When a workpiece is output, the files are transmitted in appended form with no intermediate gap. The transfer is cancelled if one of the following conditions is fulfilled:

  – Time-out: No data were received within the time allocated in the "Time-Out" box.

  – The defined end of transmission character ("Stop At" end character) was received.

Xon character

Defines the "Xon character" for character-controlled data transmission (see "Protocol").

6FC5198-6AA60-0BP0

SINUMERIK 840C, OEM Version for Windows (BA)

Xoff character	Defines the "Xoff character" for character-controlled data transmission (see "Protocol").
End of transmission	Defines the end of transmission character.
Time-Out	This setting defines the time interval for the monitoring of the transmission. If the control does not send a character within the time-out period, the control cancels its ready for transmission status with the error message: "100413 V24: Time monitoring".
Standard	You use this button to reset the settings for a device to the default settings.
Cancel	The "Parameter Assignment V24" dialog box is closed <u>without</u> saving the modified settings.
Exit	The "Parameter Assignment V24" dialog box is closed. All of the settings are saved.

> **!**
>
> **Important**
>
> It is important to note the following because of the internal features of the MS Windows driver for serial interfaces:
>
> • When data entry is completed, the MS WINDOWS RTS signal is not set to LOW but remains in the HIGH state thus signalling to the sender that it is ready to receive.
>
>   Therefore always start the receiver and then the sender.
>
> • When data output is completed, i.e. when ETX has been sent, a few more idle characters are sent. These characters might also be stored by the receiver (PCIN with setting: binary format).

■

Diagnosis

8

8.1 Selecting Diagnosis/Information

To select the data manager, first press the DATA AREA key.
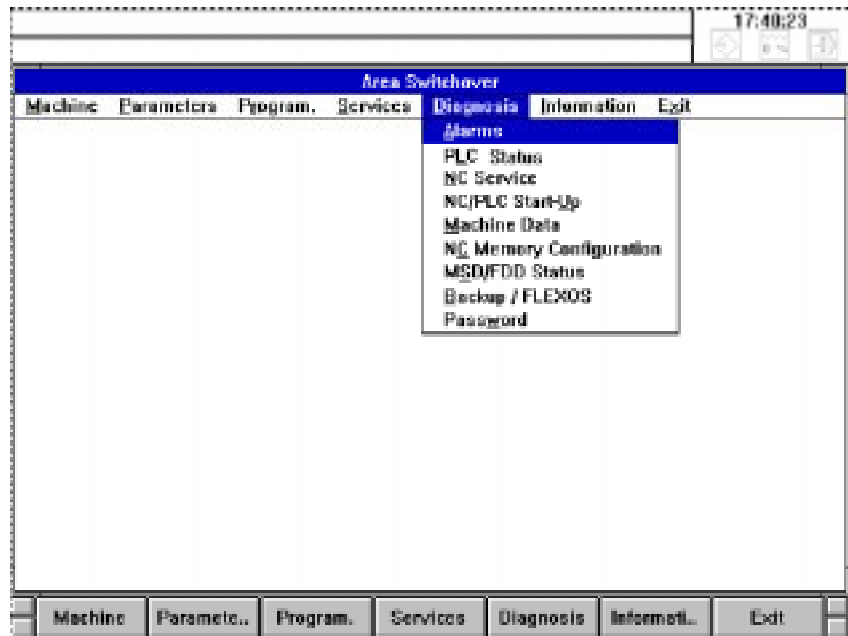
Fig. 8-1   Menu: **Diagnosis**

When Area Switchover
is selected:

- First press the DIAGNOSIS softkey to open the Diagnosis menu.

- When the menu is open you can choose the various areas by entering the underlined letter (hotkey) as follows:

  - "A" Alarms

  - "L" PLC Status

  - "N" NC Service

  - "U" NC/PLC Start-Up

  - "M" Machine Data

  - "C" NC Memory Configuration

  - "S" MSD/FDD Status

  - "B" Backup/FLEXOS

  - "W" Password

- You can also call the Information menu by pressing the softkey INFORMATION in Area Switchover.
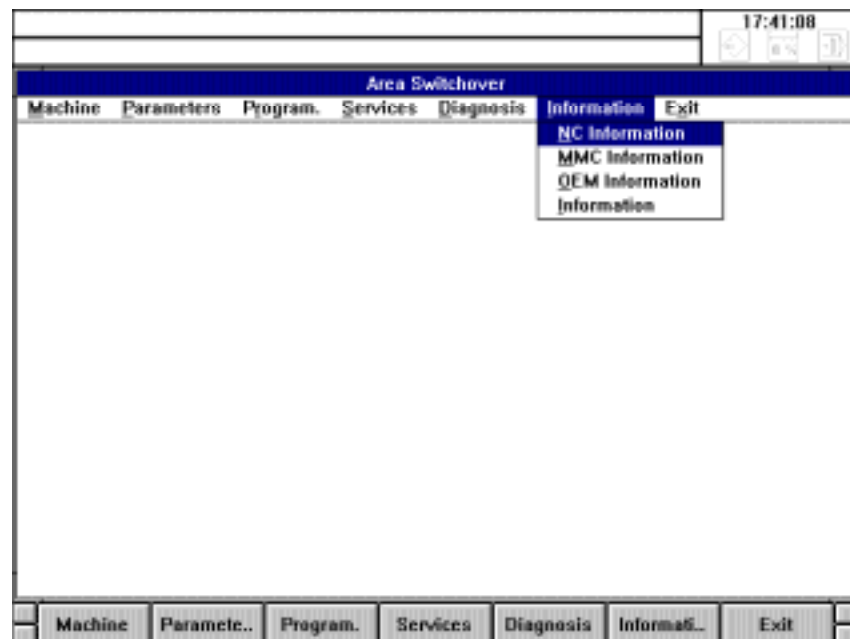
Fig. 8-2   Menu: **Information**

- When the menu is open you can choose the various areas by entering the underlined letter (hotkey):

The individual areas within the DIAGNOSIS menu mainly contain functions related exclusively to start-up or servicing. You will find a detailed description of these functions, in particular those which are password-protected against unauthorized use, in the following publication:

- SINUMERIK 840 C
  Start-Up Difference Description
  OEM Version for Windows

8.2   Alarm and message displays

The alarm/message window is used to display alarms and messages which can arise in the event of a fault.

Only one alarm or message at a time can be displayed in the alarm and message lines of the standard display. If several alarms or messages are output simultaneously, the alarm or message with the highest priority or the last alarm or message to be output is displayed in the alarm/message line.

Other alarms and messages can only be viewed in the alarm window.

You can select the alarm window via the **Area Switchover** and the DIAGNOSIS and ALARMS softkeys or directly using the hotkey:

*   SHIFT + ETC
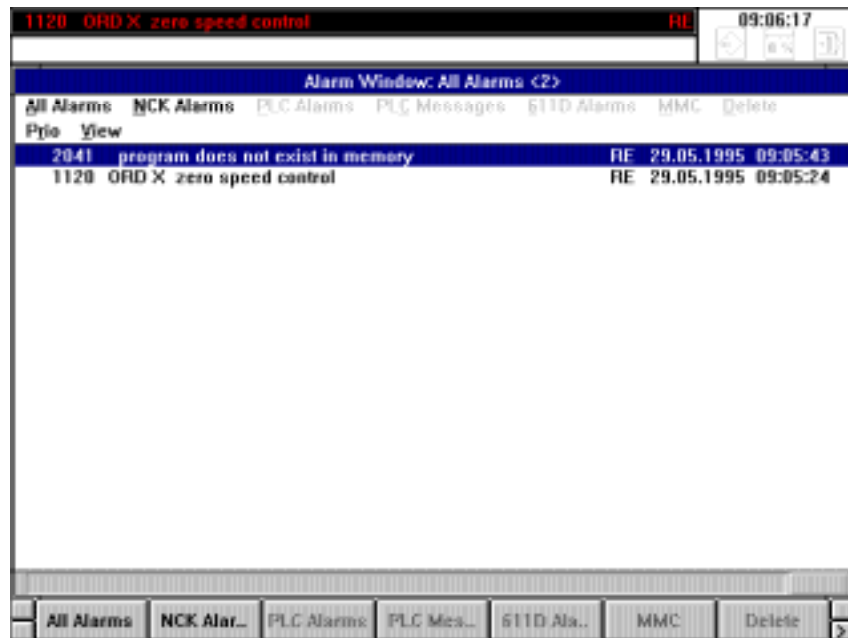
Fig. 8-3   The Alarm window

Alarms and messages are displayed in a single line with their number, text, clearance class, alarm type, date and time. You are also provided with information on the interlocks on the NCK.

The current NCK/PLC/611D and MMC alarms and messages are displayed in the order of their occurrence, i.e. the most recently triggered alarm or message appears at the top of the list.

Via softkey you can select whether you wish to display all alarms or alarms organized into individual alarm groups. You can also list the alarms in order of priority using the softkeys ETC and PRIO.

The window **All Alarms** must be selected to display such a list.
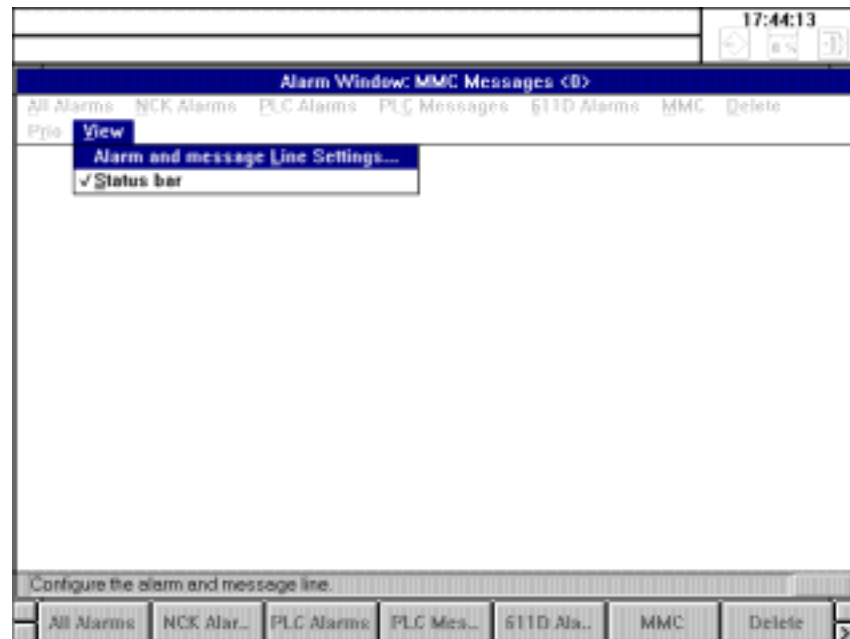
Fig. 8-4   Menu **View**

You can influence the following functions with the menu **Views**:

- The alarm and message line configuration. The following dialog box appears when you select menu item Configure alarm line.

- You can show or hide the status line at the lower edge of the alarm window via menu item **Status line**. The machine-tool manufacturer can configure the status line to display additional information about each alarm or message.

    6FC5198-6AA60-0BP0

Fig. 8-5   Configure alarm/message line

The following settings can be made individually for the alarm and message line in this dialog box.

Page

If this check box is marked the individual alarms and messages in the alarm and message line are paged.
If this box is not marked, either the alarm with the highest priority or the last alarm to be output is displayed.

Paging time

Defines the paging time of the texts in the alarm and message line. Paging is only performed if the check box "Page" is marked..

Display by priority
Display by time

These options specify whether the alarm or message with the highest priority or the last alarm or message to be output are displayed. This selection is only made if the check box "Page" is not selected.
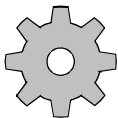
Machine manufacturer

The above settings can be disabled via the PLC.

The above settings are only activated if the function "Page from PLC" is not active. The status of this function is displayed in the top field of the dialog box.

This status display is only updated when the dialog box is opened.

Note

The settings above take effect as soon as the dialog box is closed.

8.2.1   Alarm groups

The alarms are divided into alarm groups.

NC alarms

- General alarms
- Axis-specific alarms
- Spindle-specific alarms
- Channel-specific alarms

Note

At POWER ON RESET (switching on control), all NC alarms are cancelled.

Note

The CANCEL alarms can be deleted in the associated operating area only.

PLC alarms

Note

The PLC alarms are assigned error numbers 6000 to 9999. The alarm text, alarm action and deletion conditions are configured by the machine manufacturer.

MMC alarms

Note

MMC alarms do not interrupt an active NC program. The alarms are acknowledged automatically using the CLEAR softkey.

8.2.2   Alarm numbers/clearing alarms

Table 8-1   Alarm overview

Alarm number	Type of alarm
1 to 999	General alarms
1000 to 1999	Axis-specific alarms
2000 to 3999	General alarms
4000 to 4299 5000 to 5299	Cycle alarms
6000 to 9999	PLC error messages or PLC operational messages
10000 to 12031	Axis-specific alarm
20000 to 209999	Spindle-specific alarms
100000 to 169999	MMC alarms
200000 to 209999	PLC dialogs
210000 to 219999	Unassigned
300000 to 399999	611D alarms
400000 to 999999	Reserved for Siemens
1000000 to 1099999	Reserved for OEM applications

The machine manufacturer can configure the alarm type (clearance class).
Possible alarm types are:

    – PO   POWER ON

    – RE   RESET

    – CA   CANCEL

    – ME   Message

    – PA   PLC ALARM

    – KO   Part program comment

Table 8-2   Possible alarm acknowledgement actions

POWER ON	Execution of the active NC program is aborted. After eliminating the error, the control must be switched off and on again and the reference points must be reapproached. **Caution!** when the control is switched off the contents of the NCK part program memory are lost.
⬜ / RESET	An active NC program is not aborted, but only stopped. After eliminating the error, it is possible to continue execution of the NC program from the point at which it was stopped.
⬜ ⊖ CANCEL	An active NC program is not aborted but only stopped. After eliminating the error, it is possible to continue execution of the NC program from the point at which it was stopped.
SK: DELETE	A message does not interrupt execution of the active NC program. The message is acknowledged in the Alarm window with the CLEAR softkey.

POWER ON means switching off the control and switching it on again.

Please note the information provided by the machine tool manufacturer.

8.2.3   Display of the alarms and messages in the alarm and message lines

The alarm or message with the highest priority is displayed in the alarm/message line. The machine manufacturer can configure whether an alarm/message is displayed in the alarm or message line. Existing comments are overwritten by alarm texts. The alarm line is the first line on the screen.
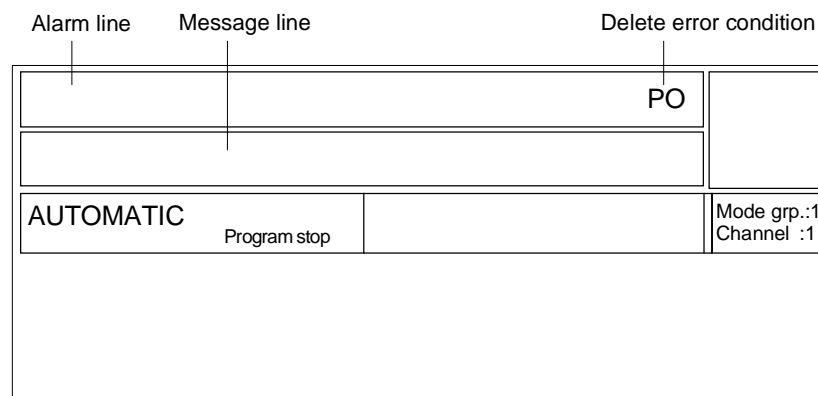
There are three types of display representation for alarm messages: Types A, B and C.

Example of display representation Type A: Alarm display in sequence order

```
┌────────────────────────────────────────────────────────────┬──────┐
│ 10243  ORD  5 X       Illegal pulse multiplication      RE   │      │
├────────────────────────────────────────────────────────────┤      │
│                                                              │      │
└────────────────────────────────────────────────────────────┴──────┘
```

└ Clearance cond.

Approx. 40 characters for
explanatory text
(for single-line alarm)
max. 100 characters
(for two-line alarm)

Max. 5 characters for
ordinal number. The ordinal
number shows the order in
which the alarms have occurred.

Max. 6 characters for alarm number
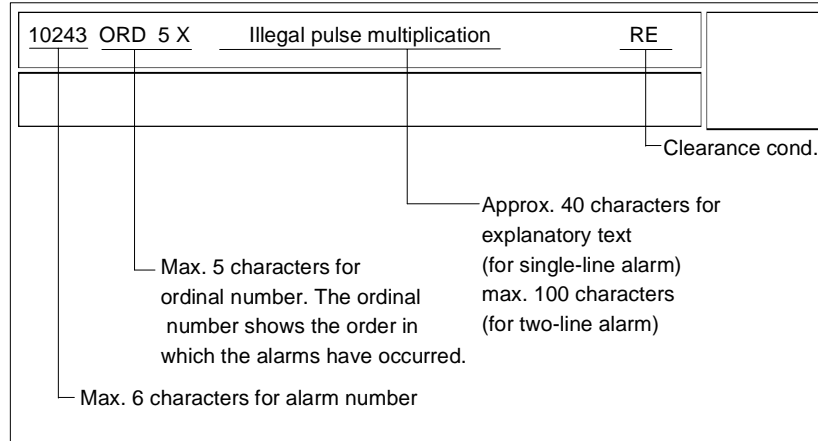
Example of display representation Type B: Alarm display in block number order

```
┌────────────────────────────────────────────────────────────┬──────┐
│ 3000  1  N0045  General programming error              CA    │      │
├────────────────────────────────────────────────────────────┤      │
│                                                              │      │
└────────────────────────────────────────────────────────────┴──────┘
```

└ Clearance cond.
  (2 characters)

Approx. 38 characters for
explanatory text

Max. 5 characters for block
number. E.g.: The error has occurred in block N0045.

1 character for channel number

Max. 5 characters for alarm number

Example of display representation Type C:

```
┌────────────────────────────────────────────────────────────┬──────┐
│ 6000     Hydraulic oil min.                            CA    │      │
├────────────────────────────────────────────────────────────┤      │
│                                                              │      │
└────────────────────────────────────────────────────────────┴──────┘
```

Max. 4 characters for
alarm number

Approx. 47 characters   └ Clearance cond.
for explanatory text
(for single-line alarm)
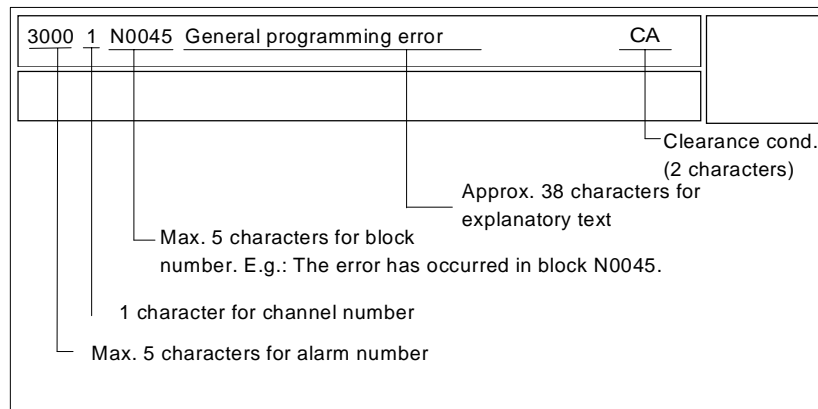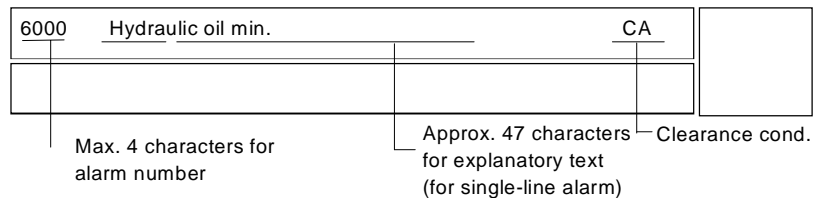
8.2.4 On-line help for alarms and messages

The user can call up additional information on alarms by positioning the selection bar on the alarm with the ARROW keys and pressing the HELP key:
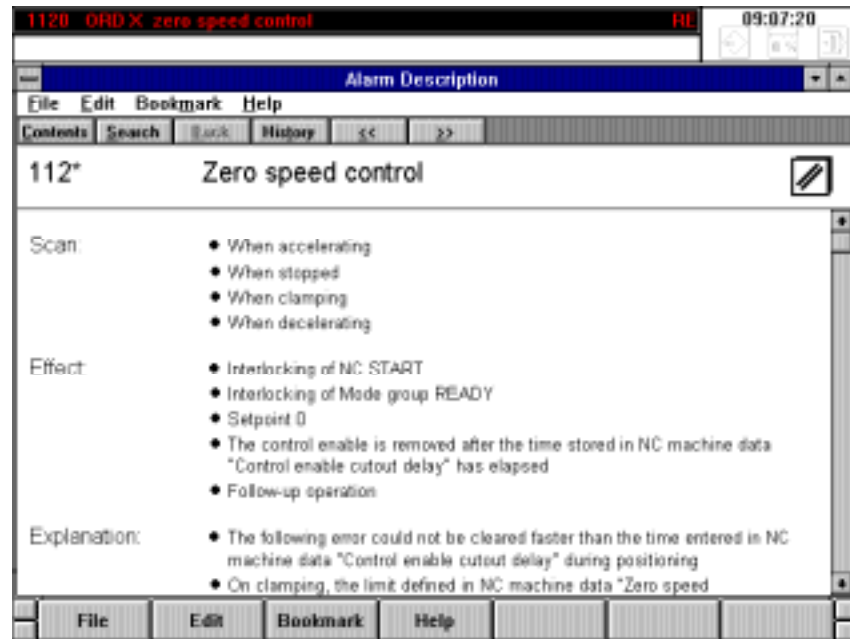
Fig. 8-6   On-line help for alarms

The on-line help for alarms is presented in a standard format. The header of each alarm description provides information on the alarm number, the alarm text and the clearance mode.

Table 8-3   Overview - On-line help

Alarm number	Alarm text	Clear key
Query:	Details of the state in which the alarm occurred.	
Effect:	Details of the effect of triggering the alarm.	
Explanation:	Details of the reasons for the occurrence of the alarm.	
Remedy:	Instructions for eliminating the alarm state.	

The help menu can be closed again by selecting the **Exit** command in the **File** menu.

The **Contents**, **Search**, **Back**, **History** and **Browse** (**<<** and **>>**) commands in the toolbar shown above can be selected using the appropriate HOTKEYS on the operator panel.

8.3 PLC Status

The PLC Status display is used to provide the commissioning engineer with additional information on error codes and on the signal status on the interface.
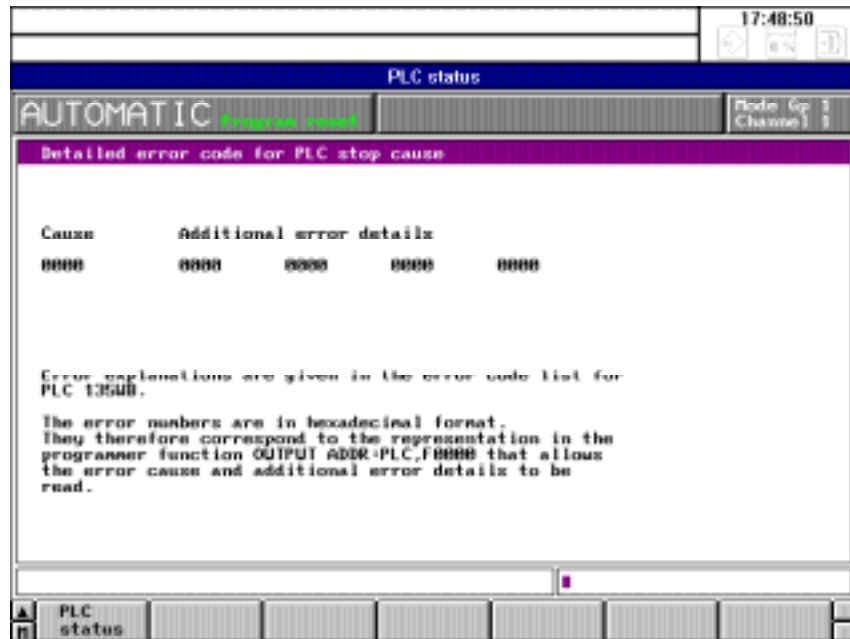
Fig. 8-7   Error coding in the Diagnosis area

8.4 NC Service

In addition to other start-up and diagnostic functions, the NC Service area can be used to perform a drift compensation on the axes.

The cause of the drift is attributable to the temperature dependency of semiconductor components in the motor actuator unit. The axes drift from the setpoint position and a permanent offset results.

If the < or > indicators remain visible following a traversing movement with stationary axes, the drift has exceeded the values defined in MD 204* or MD 208* and a drift compensation is required.

The drift compensation is performed semi-automatically and is activated by softkey.

Preconditions
- The < or > indicator (position not reached) is still visible following a traversing movement.
- The axes of the NC and the drives are in closed-loop control mode.
- The axes are stationary.
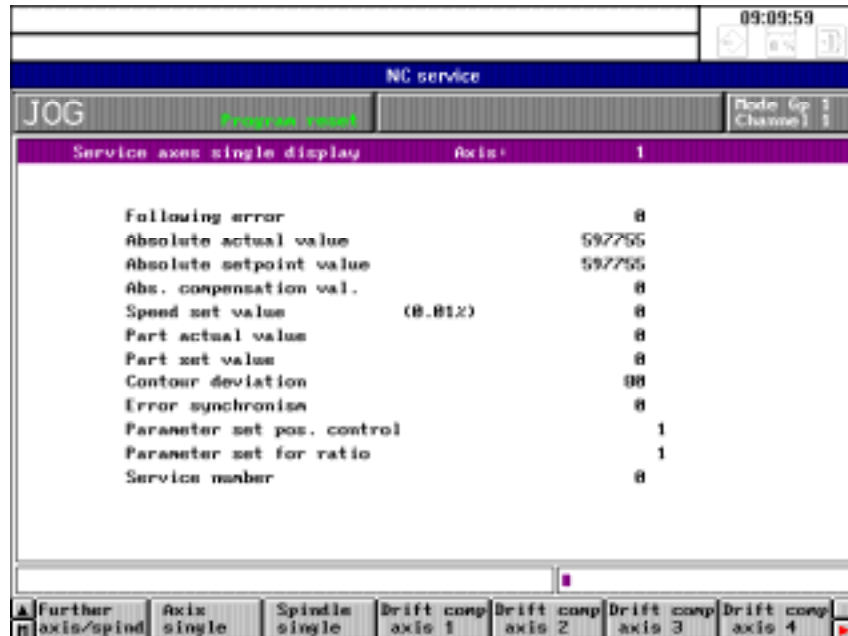
Fig. 8-8   Drift compensation for axis 1 in the **Diagnosis** area

Press the following softkeys consecutively:

NC SERVICE

> NC
> service

AXIS SINGLE

> Axis
> single

DRIFT COMPENSATION AXIS 1 etc.

> Drift comp
> axis 1...

The drift compensation is performed for the selected axis.

> **Caution**
>
> On high-precision machines, the drift compensation should be performed several times a day during operation, as the drift directly affects the following error.

8.5   Password

For start-up and service, it is necessary to enter the password to increase the access rights. The SINUMERIK 840C provides a special menu for this purpose:

Fig. 8-9   Password in the **Diagnosis** area

Please refer to the machine manufacturer's documentation for further information.

8-14   SINUMERIK 840C, OEM Version for Windows (BA)

8.6   NC Information

In the NC Information area, you receive information on the various versions of the control software and UMS (user memory submodule)
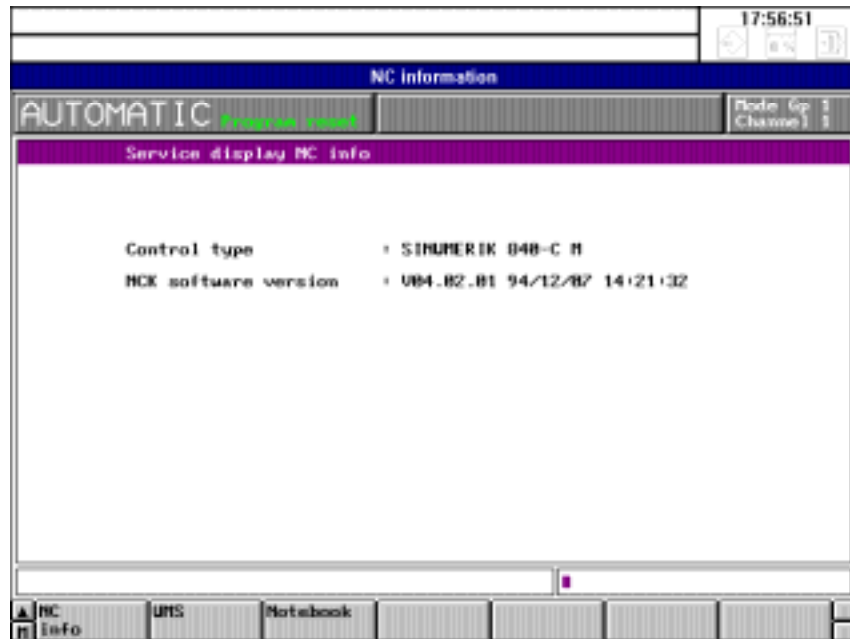
Fig. 8-10 NC information in the **Information** area

NC
info

Press the NC INFORMATION softkey to display an information menu providing you with details of the NCK software version and the control type. This information is important for servicing.

UMS

Pressing the UMS softkey displays information on the contents of the user memory submodule (languages, menus, etc.).

Notebook

The NOTEBOOK function is intended for servicing and start-up and is protected by password.

6FC5198-6AA60-0BP0

8.7   MMC Information

The message box **MMC information** contains information about the MMC software version of the control.
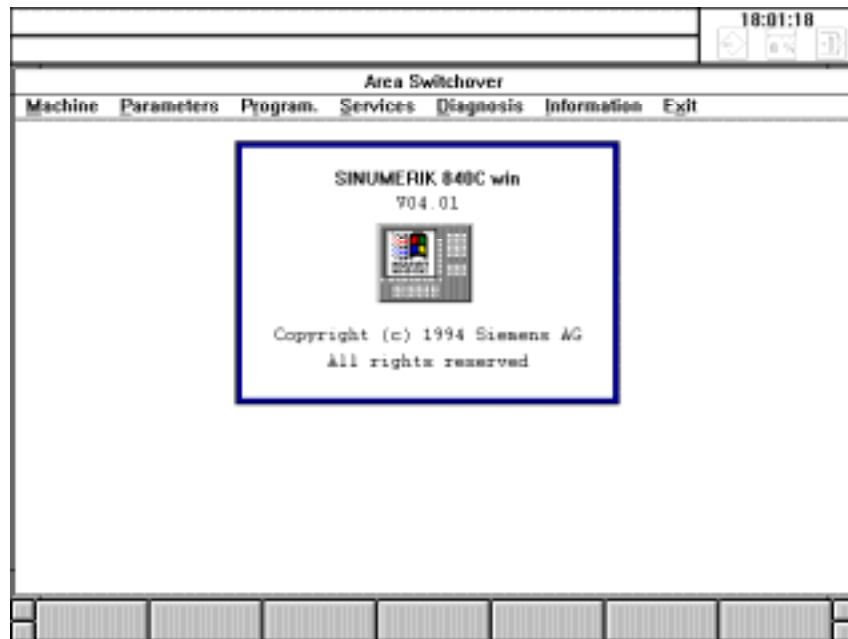
Fig. 8-11   MMC information

The copyright message disappears after approximately 5 seconds.

In the same way, the OEM user can display information about his own applications under menu item **OEM information**.

8.8 Information

The message box **Information** contains information about the software version of the individual components, MMC Windows, NCK, PLC, 611 FDD/MSD and MMC FlexOS.

Note

The contents of this figure does not correspond to the current status.
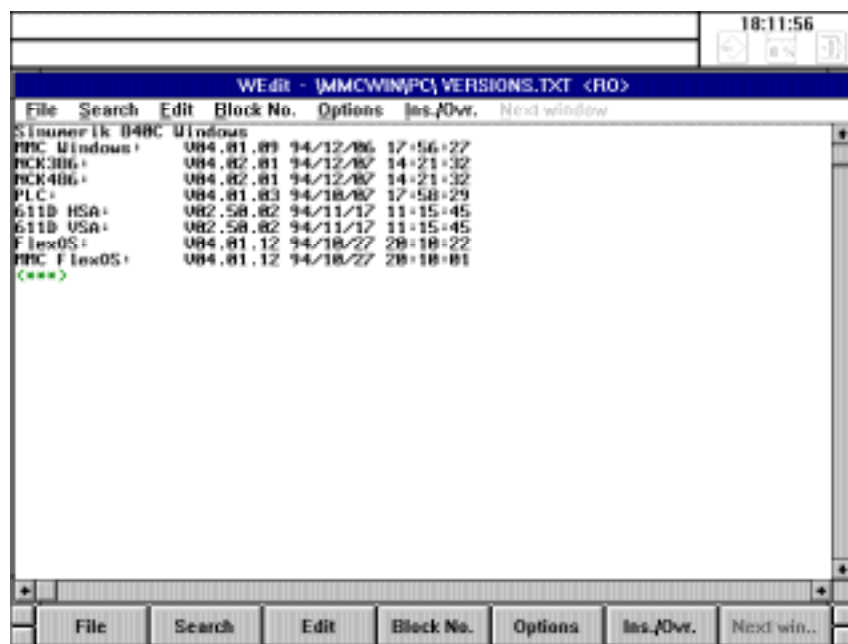
Fig. 8-12 Menu **Information**

8.9   Exit

The SINUMERIK 840C control must not be simply switched off but "powered down", in order to assure the integrity of the files on the hard disk and avoid loss of data.

The control can be powered down by pressing the "Exit" key in the Area Switchover menu.

Fig. 8-13 SYSTEM DOWN dialog box

The power supply must not be switched off until the exit command has been confirmed by pressing the "Yes" button in the SYSTEM DOWN dialog box (see Fig. 8-13 above) and the "System is down" display appears at DOS level after leaving Windows.

Please refer to the machine manufacturer's documentation for further information. ■

Maintenance

9

9.1 Operating data

Table 9-1 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 • Rear of operator panel • Front of machine control panel • Rear of machine control panel	 IP 54 IP 00 IP 54 IP 00

Complete data on use and operation can be found in the documentation "Interface Part 2, Connection Conditions".

9.2　　Replacing the battery

There is a backup battery (beim CSB bis Erzeugnisstand C 6LR61, 550mA, 9V) on the central service board (CSB) 6FC5114-OAAO2-OAAO. The following data is backed up with this battery voltage: PLC program, RPA, TOA, SEA, SEA4, ZOA.

Note

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.

The period between warning steps 1 and 2 of battery monitoring is approximately 4 weeks.

Note

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

Replacing the battery
- Store and transport the battery in its packaging.
- Prior to installation, check the battery for any damage. Do not use damaged batteries.

Procedure
- Undo the screw on the battery cover
- Remove both cover and battery carefully
- Use the screwdriver to press the battery out of the battery cover
- Remove the contact piece from the battery
- Fix the new battery to the contact piece
    - for CSB from product version D:　Lithium batteries (Varta, type 6127 503 074)
    - for CSB up to product version C:　Alkaline batteries (Type 9LR61, 550mAh)

Note   On the CSB, product version D, the operating points of the battery monitor
                                 have been adapted to the lithium battery.

                                 If you combine the CSB, product version C, with a lithium battery or the CSB,
                                 product version D, with an alkaline battery, it is possible that the advance
                                 warning time of 4 weeks or the backup time of 1 year cannot be maintained
                                 under adverse environmental conditions (temperature).

                                 • Press the battery into the battery cover

                                 • Fix the battery cover with the screw to the CSB.

 ❗   **Important**

                                 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.

9.3 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.
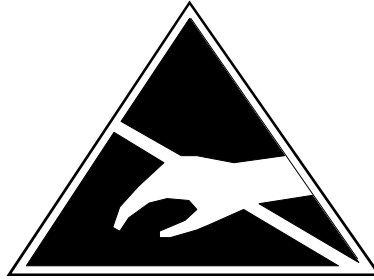
ELECTROSTATIC **S**ENSITIVE **D**EVICES (ESD) can be destroyed by voltages and energy levels well below those perceived by the human body.

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 can be used.

9.4 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.

Table 9-2   Tips for fault elimination

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	• Contactors not connected	• Connect contactors (coils)
	• Adjacent machines producing sparks (welding sets)	• Lay cable to the control differently (increase distance to cause of fault)
	• Radio transmitter, centralized multi-station control system	• Additional screening
	• Cable with wrong screen connection, cable with wrong wire combinations or wrong characteristics	• Use original cable and check core assignment
	• Interruption in cable screening (e.g. if a cable distributor is interposed)	• Connection of cable screening, screening the point of interruption (e.g. the cable distributor)
	• Bonding conductor laid wrongly	• Re-lay bonding conductor
	• Control dirty	• Clean control (and modules) and ensure that cleaner air is supplied
Permanent axis offset	See possible causes as for 'Sporadic failure'.	See elimination of fault as for 'Sporadic failure'. The actual value cable acts as the fault sink.

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)

9.5   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.

Note

Solvents such as chlorinated hydrocarbons, benzene, esters and ethers should be avoided!

■

Abbreviations / Glossary

10

10.1 Abbreviations

ASCII	American Standard Code for Information Interchange
BCD	Binary Coded Decimal
BRK	Break (program control)
CLR	Clear, delete blocks (program control)
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 Ready
DW	Data Word
EIA code	Special tape code; number of perforations per character always odd
EPROM	Erasable Programmable Read Only Memory

ETC	ETC key; Extension To Text Line
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
ISO code	Special tape code; number of perforations per character always even
JOG	Jogging
KYRU	Key Request Unit
K1...K4	Channel (max. 4 channels with SINUMERIK 840C)
LED	Light Emitting Diode
LF	Line Feed
MD	Machine Data
MCP	Machine Control Panel
MCS	Machine Coordinate System
MDA	Manual Data Automatic
MMC	Man Machine Communication (PC area in SINUMERIK 840C)
MPF	Main Program File
NC	Numerical Control
NCK	Numerical Control Kernel
OPI	Operator Panel Interface
PLC	Programmable Logic Controller
PST	Procedure step, program step (program control)

RAM	Random Access Memory
ROV	Rapid Override
RPA	R Parameter Active, NCK memory for R parameter numbers
RTS	Request to Send
SBL	Single Block
SEA	Setting Data Active, NCK memory for setting data
SD	Setting Data
SKP	Skip
SPF	Sub Program File
SW limit switch	Software Limit Switch
TEA	Testing Date Active, refers to machine data
TO	Tool Offset
TOA	Tool Offset Active, memory for tool offsets
TRC	Tool Radius Compensation
UMS	User Memory Submodule
WCS	Workpiece Coordinate System
ZO	Zero Offset
ZOA	Zero Offset Active, memory for zero offsets

10.2   Glossary

Access rights   The following possibilities are available for preventing unauthorized access to NC data and functions

- Entry of password

- Locking with keyswitch

- Cycle disable

Alarms   All -> messages and alarms are displayed on the operator panel in plaintext with the corresponding abbreviation for the clearance criterion. The display is separated into alarms (alarm line) and messages (message line).

If the alarm window is selected, additional help can be displayed for the alarms and messages by pressing the info key.

Approach reference point   If the position measuring system used is not an absolute value encoder, it is necessary to approach the reference point to ensure that the actual values supplied by the measuring system are in keeping with the machine coordinate values.

Archiving   Reading out of files and/or directories to an **external** storage unit.

Area switchover   Central program that can be selected by means of the data area key. Area switchover (REGIE.EXE) allows the operator to start further applications.

ASIC   (Application Specific Integrated Circuit): Integrated circuit which, as opposed to standard circuit is designed and manufactured in accordance with application-specific characteristics.

AUTOMATIC   Mode of the control (block sequence mode in accordance with DIN): Mode in NC systems where a part program is selected and continuously executed.

Auxiliary function   With auxiliary functions, additional information can be transferred in part programs to the PLC where they initiate responses that have been defined by the machine manufacturer.

Back up   Copying of the contents of memory to an external storage unit.

Backlash compensation   Compensation for mechanical backlash in a machine, e.g. when reversing ball screws. The backlash compensation can be entered separately for each axis.

Backup battery   The backup battery ensures that the PLC user program is stored in such a way that it will not be lost if there is a power failure and that defined data areas (R parameters, tool offsets etc.), flags, timers and counters are held remanent.

Baud rate   Rate of data transfer (bit/s).

Blank   Workpiece in the unmachined state.

Block   Part of a part program, concluded by a Line Feed. A distinction is made between main blocks and subblocks.

Block search	When verifying part programs or after processing has been interrupted, the block search function can be used to select any point in the part program at which processing is to start or be continued. Block search can be performed with or without calculation.
Boot	Loading of the system program after Power On.
C axis	Axis about which a controlled rotary motion and positioning take place with the workpiece spindle.
Channel	A channel is characterised by its ability to execute a part program independently of other channels. Part program sequences in different channels can be co-ordinated by synchronisation.
Clipboard	When copying and reading in data, the data can, if required, always be stored first of all in the clipboard (buffer) from which it can then be inserted/arranged in a specific directory.
CNC	-> NC
COM1, COM2	Serial interfaces on the CPU module
CPU	Central Processing Unit
Cursor	Marker used in editors, input boxes etc. to identify the position.
Cutter radius compensation	When programming a contour, the tool is assumed to be pointed. Because this is not a real life situation, the radius of curvature of the tool used by the control is specified and allowance is made for this. The centre of curvature is shifted by the radius of curvature equidistantly around the contour.
Cycle	Protected subroutine for executing a repetitively occurring machining operation on the workpiece.
Data	Individual value(s), e.g. machine data.
Data block	Data unit of the PLC to which STEP 5 programs can access.
Data selector	Aid for selecting and displaying files and directories.
Data transfer program PCIN	PCIN is a universal program for sending and receiving CNC user data through the serial interface, e.g. part programs, tool offsets etc. The PCIN program will run under MS-DOS on standard industrial PCs.
Data word	Two-byte data unit within a data block.
Diagnosis	Operating area in the control
Dialog	Dialog text shown on the screen in accordance with a menu and to which a reply is expected by means of softkey functions or by pressing a key.
Directory	Organisation unit for the hierarchical management of the data on a hard disk. Files and/or further (sub)directories can be stored within a directory.

Directory path	This specifies the position of a file in the directory hierarchy.
DRAM	(Dynamic Random Access Memory): Volatile memory chip with direct access to each individual memory location. The locations retain their contents as long as they are supplied with electrical power.
DRF	Differential Resolver Function: NC function that generates an incremental zero offset in the Automatic mode with an electronic handwheel.
Drift compensation	SINUMERIK 840C has a semi-automatic drift compensation function which compensates the drift of the drive due to temperature changes. Drift compensation is needed in order to eliminate the permanent contour offset which results from the accumulative following error.
EDIT key	Key on the operator panel: In NCK areas: Edit existing values In MMC areas: Same function as tabulator key on the MF2 keyboard
Editor	The editor allows texts/program blocks to be edited, added to, pushed together, and inserted.
EMC	Electromagnetic compatibility is understood to mean the capability of an item of electrical equipment to function without fault in a given environment without having an unacceptable influence on the surroundings.
Enter box	Predefined area for entering values and texts.
EPROM	(Erasable Programmable Read Only Memory): Non-volatile programmable memory which retains its contents even if the power is switched off.
Feed override	The programmed feedrate is modified from the machine control panel or from the PLC (0 - 120%).
Feedforward control	By means of the feedforward control, the velocity setpoints calculated by the interpolator are output directly to the drive, i.e. without looping through the position control. Inaccuracies in the contour, caused by the following error, can be almost entirely eliminated by the dynamic, acceleration-dependent feedforward control. This results in good machining accuracy even at high contouring velocities.
File	Collection of data and/or text
File type	Possible types of files, e.g. part programs, zero offsets, R parameters, etc.
Flash EPROM	Non-volatile programmable memory with high memory density that retains its contents even if the power is switched off.
Helical interpolation	Helical interpolation is a great help in producing internal or external threads with form cutters and for milling lubrication groups. The helical path is made up of two motions: • Circular motion in a plane • Linear motion perpendicular to this plane.

Icons	Graphic symbols on the screen in the "Symbols" field that display statuses with fixed meaning in all channels, e.g. switch on lubricants.
Inch system of units	A system of units that defines distances in inches and fractions of inches.
Incremental dimension	The distance traversed represented by number of increments x increments length. The number of increments can be stored in the form of setting data or it can be selected by means of special keys with the inscriptions 10, 100, 1000, 10 000.
Initialisation files *.INI	Initialisation files are special ASCII files. They contain value assignments that are read out before program execution as such. Initialisation files are used in particular predefined data.
Input	Input key, enter key: Transfer (acceptance) of an edited value
Intermediate blocks	Traversing motions with selected tool offset (G41/G42) may be interrupted by a limited number of intermediate blocks (blocks without axis motion in the offset plane). The tool offset can still be allowed for correctly in the calculation.
JOG	Mode of the control (set-up mode). In the JOG mode the machine can be set up. Individual axes and spindles can be traversed in the JOG mode using the direction keys. Other functions in the JOG mode are reference point of approach and preset (set actual value).
Keyswitch	The keyswitch on the machine control panel has four positions that are interpreted by the operating system of the control. Furthermore three keys with different colours fit in the key switch and can be withdrawn in the specified positions.
KV	Servo-gain factor: A term used in closed-loop control systems. This determines the dynamic response (rise time) in the position control loop.
Languages	The display texts for operator guidance, the system messages and the alarm/messages are available in 5 system languages: **English, French, German, Italian** and **Spanish**. The above languages can be installed in the control and then selected by means of the Diagnosis/Change Language softkeys.
Leadscrew error compensation	Compensation of mechanical inaccuracies in a ball screw involved with a feed, provided by the control on the basis of stored measured values for the deviations.
Machine	Operating area of the control.
Machine axes	Axes that exist physically in the machine tool.
Machine coordinate system (MCS)	Coordinate system referred to the axes of the machine tool.
Machine zero	Machine zero point, machine datum: Point that has been uniquely defined by the machine tool, e.g. reference point.

Main memory	The main memory is a RAM in the CPU on which the processor accesses the user program at the time of program execution.
MCP	Machine tool control panel with operator control elements such as keys, rotary switches etc. and symbol indicating elements such as LEDs. It is used for controlling the machine tool directly via the PLC.
	The MCP is linked to the PLC CPU (135 WD) and interface PLC (135 WB/2) through the operator panel interface.
MDA	Mode of the control: Manual Data Automatic. In the MDA mode, individual program blocks or block sequences can be entered without reference to a main program or subroutine and then be executed immediately by means of the NC-Start key.
Menu, Softkey menu	Displayed control functions on the screen that can be selected by means of softkeys.
Messages	All messages and alarms are displayed on the operating panel in plaintext with date and time and the appropriate symbol for the clearance criterion. The display distinguishes between alarms and messages.
Mirroring	In mirroring, the signs of the coordinate values of a contour are interchanged with respect to an axis. Mirroring is possible with respect to several axes at the same time.
MLFB No.	Order number: Machine-readable product designation
MMC	Man Machine Communication, operator interface. Another designation for the integrated PC of a machine-tool control.
Mode	Execution concept for operating a SINUMERIK control. The following modes are defined: -> JOG, -> MDA, -> TEACH IN -> AUTOMATIC.
Module	• A self-contained assembly unit or functional group which as a rule can become effective in a wider function relationship.
	• A self-contained software program
NC	Numerical Control: This includes all the components for machine tool control: NCK, PLC, MMC,
	Note: For the SINUMERIK 840C controls the term CNC would be more accurate: computerized numerical control.
NCK	Numeric Control Kernel: Component of the NC control that executes part programs and co-ordinates the motion sequences of the various axes.
Network	A network represents the linking together of several computers or control units through connecting cable. Communication takes place through the network, i.e. the data exchanged between the linked devices.
OEM	Original Equipment Manufacturer: An OEM makes his own operator interface or includes technology-specific functions in the control.

Operating area	The main operating areas are: Machine, Parameters, Programming, Services, Diagnosis.
Operator interface	The operator interface is the means of providing a display in a CNC control and is provided in the form of a screen. It has 7 horizontal and 7 vertical softkeys.
Operator panel	The operator panel comprises a video display unit and an NC keypad. On the slimline operator panel (TFT screen) the NC keypad is integrated in the operator panel unit, as opposed to the CRT version. The operator panel is linked to the operator panel interface with the first serial interface of the MMC in the control.
Overall reset (general reset)	The purpose of an overall reset is to clear specific areas of NCK and PLC memory before starting up (again).
Overall reset mode	A special mode for installation and start-up of the control. In the overall reset mode no axes can be traversed or part programs executed.
Parameter	• Operating area of the control • Value for setting or for controlling specific characteristics or functions of physical modules and software programs.
Part program	A series of instructions to the NC control for producing a specific workpiece part.
PART program manager	• The part program manager on the MMC of the control is workpiece oriented. Each workpiece is assigned a directory in which the data required to produce a workpiece (e.g. part programs, tool offsets, R parameters etc.) are stored. • On the NCK side, the part program manager is linear. Within the part program memory (DRAM) the various part programs are linear, i.e. they are stored without any kind of hierarchy cross-references.
Path	Specifies the position of a file in the directory hierarchy on a hard disk.
PG	Programming unit
PLC	Programmable Logic Controller: Component of the NC control; interface control for handling the control logic in machine tools.
PLC	Electronic controllers whose function is stored as a program in the control unit. The design and wiring of the unit do not therefore depend on the function of the controller. The programmable logic controller has the same structure as a computer: it comprises CPU (central controller) with memory, input/output modules and internal bus system. The I/O devices and the programming language (STEP 5 -> user program) are designed to suit the needs of control technology.
PLC block	-> User program

Preset	With the preset function the control zero can be redefined in the machine coordinate system. With preset, no movement of the axes takes place, a new positional value is simply entered for the momentary axis positions.
Program	• Operating area of the control • Sequence of instructions to the control.
Program block	-> User program
Program control	In the Automatic and MDA mode, this function can be used to change the program mode, for instance skip block can be selected.
Programming key	Characters and character strings that have a defined meaning in the programming language for part programs (see Programming Guide).
Quadrant error compensation	Contour errors at quadrant transitions resulting from varying frictional conditions (static friction) on guideways can be largely eliminated with the quadrant error compensation function. Parameterization of the quadrant error compensation takes place by a circularity test.
R parameter	Arithmetic parameter; can be set or called in the part program for any purpose by the programmer.
Rack	Mounting frame of the control.
Recall	Jump back into the next higher horizontal menu.
Reference point	Point on the machine tool to which the measuring system of the machine axes refers.
Remanence	Data areas in data blocks are remanent as well as timers, counters and flags if their contents are not lost at restart or power off.
Repos	Reposition on the contour by operator control. With the Repos function and by using the direction keys, it is possible to return to the point of interruption.
Reset	Reset key: Press the Reset key: processing of the current part program is stopped. Messages from the monitoring system are cleared provided they are not Power On or Acknowledgement Alarms. The control is put into the "Reset" state, i.e. the NC control remains in synchronism with the machine, all buffers and working memories are cleared (but the contents of the part programs memory are retained). The control is in the initial state and ready for a new program run.
Reset state	Setting of attributes and parameters for the time of beginning of program.
Rotary axis	Rotary axes allow a workpiece or tool to be rotated into a specific angular position.

Rotary axis rotating endlessly	Depending on the particular application, the traversing range of a rotary axis can be selected to be either limited to less than 360° or rotate endlessly in both directions. Endlessly rotating axes are used, for example, for unround machining, grinding work and winding jobs.
Rounding axis	Rounding axes allow a workpiece or tool to rotate into an angular position corresponding to an index grating. On reaching a grating the rounding axis is "in position".
S5	Abbreviation for S5 -> user program, PLC.
Safety functions	The control includes constantly active monitoring functions which detect disturbances in the CNC, the interface control (PLC) and in the machine early enough to largely prevent damage from occurring to workpiece, tool or machine. If a disturbance occurs, the machining operations are interrupted, the drives are stopped, the cause of the disturbance is stored and a corresponding alarm is displayed. At the same time, the PLC is notified that a CNC alarm has occurred.
Screen darkening	When the screen darkening function is active, screen and back lighting on the operator panel are switched to dark after a settable time. The screen darkening function is not implemented in the OEM version for Windows -> screen saver.
Screen saver	MS-WINDOWS application that can be activated via the Control Panel. After a parameterizable time varying figures appear on the screen and thus prevent screen burn-in.
Scroll bar	Slider that shows which section (beginning, middle, end) of a program or a listing has been selected.
Serial interface V.24	Two RS232C (V.24) interfaces are provided by the MMC module of the control for data input/output as standard. The first interface is linked to the operator panel interface on the operator panel. The second interface can be used by the operator for data backup. Part programs as well as manufacturer and user data can be loaded and backed up through this interface.
Services	Operating area of the control for data transfer
Setting data	Setting data represent attributes of the tools used or of the workpiece to be produced (not of the machine tool -> machine data). They are entered through the operator panel or through V.24 into the control and allowed for accordingly at the time of program execution.
Simulator module	A simulator module is a module • on which digital qualities entered through operator control elements can be simulated, and • digital output quantities can be displayed

Softkey	A key whose inscription is represented by a field in the screen and which varies dynamically to suit the current operating situation. Softkeys are function keys for which the functions are defined in the software.
Software limit switch	Software limit switches limit the traversing range of a axis and prevent the table from contacting the hardware limit switches. Two pairs of values can be specified for each axis, and these can be activated separately by means of the PLC.
Spindles	The spindle functionality is divided into two performance steps: Spindles: Speed or position closed-loop controlled spindle drives analog +/- 10V or digital Auxiliary spindles: speed open-loop spindle drives, e.g. for driven tools.
Spline interpolation	With spline interpolation, the control can produce a smooth curve on the basis of just a few intermediate points that have been given on a set contour.
Standard cycles	For frequently recurring machining tasks, standard cycles are available: • for drilling/milling technology • for turning technology
Subroutine (subprogram)	A series of instructions in a part program that can be repeatedly called with different supply parameters. The subroutine is called from the main program. Every subroutine can be locked to prevent unauthorized output and display. Cycles are one form of subroutine.
TEACH IN	Mode of the control. Part programs can be created or corrected with **TEACH IN**. The various program blocks can be entered thought the keyboard and be executed immediately. Positions approached by the direction keys or handwheel can also be stored. Additional information such as G functions, feeds or M functions can be entered in the same block.
Text editor	-> Editor
Tool	The working part on the machine tool responsible for the machining of the workpiece, e.g. turning tool, milling cutter, drill, laser beam.
Tool offset	The tool is selected by programming a **T function** (4 decade integer) in the block. Selection of the offset values associated with the tool is by means of **D numbers**.
Tool radius compensation	In order to be able to directly program a required workpiece contour, the control must traverse a path which is equidistant from the programmed contour making allowance for the radius of the tool used (G41/G42).
User memory	All programs and data such as part programs, subroutines, tool offsets, zero offsets etc. can be stored in the mutual CNC user memory.

User program   User programs for the PLC in the 840 C are written in the STEP 5 programming language. The user program is of modular structure and consists of various blocks.

The fundamental block types are:

Program blocks (OBs, PBs, FBs, FXs, SBs): these blockss contain the STEP 5 commands

Data blocks (DBs, DXs): these blocks contain constants and variables for the STEP 5 program.

Vocabulary words   Words used in a programming language with defined notation and meaning

Words with defined notation within initialisation files. The values assigned to the various vocabulary words are used as initialisation parameters for the associated application.

Workpiece
- Part to be made or machined by the machine tool.
- A workpiece is the name given to a directory in which programs and other data such as main programs, initialisation file, tool offsets (see data structure description) are stored. The workpieces are in turn stored in a directory.

Workpiece contour   Desired contour of the workpiece to be made/machined.

Workpiece coordinate system   The workpiece coordinate system has its origin in the workpiece zero point. When programming in the workpiece coordinate system, dimensions and directions refer to this system.

Workpiece zero   The workpiece zero (workpiece zero point, workpiece datum) represents the origin for the workpiece coordinate system. It is defined by distances to the machine zero.■