Operator's Guide   03/2004 Edition

HT6

sinumerik

SINUMERIK 840D/840Di/810D

Handheld Terminal 6

HT6

SIEMENS

SIEMENS

SINUMERIK 840D/840Di/810D

Handheld Terminal 6
HT 6

Operator's Guide

Valid for

Control	*Software version*
SINUMERIK 840D powerline	7
SINUMERIK 840DE (export version) powerline	7
SINUMERIK 840Di	3
SINUMERIK 840DiE (export version)	3
SINUMERIK 810D powerline	7
SINUMERIK 810DE (export version) powerline	7

03.04 Edition

Introduction	**1**
User interface	**2**
Parameters operating area	**3**
Program operating area	**4**
Machine operating area	**5**
Hand-controlled operation	
Machine operating area	**6**
Automatic mode	
Teach-in operating area	**7**
Services operating area	**8**
Diagnostics operating area	**9**
Start-up operating area	**10**
Appendix	**A**

SINUMERIK® Documentation

Printing history

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

The status of each edition is indicated by the code in the "Remarks" columns.

Status code in the "Remarks" column:

A ....   New documentation.

B ....   Unrevised reprint with new Order No.

C ....   Revised edition with new status.

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

Output	Order No.	Remarks
06.00	6FC5298-0AD60-0BP0	A
11.00	6FC5298-0AD60-0BP1	C
06.02	6FC5298-0AD60-0BP2	C
03.04	6FC5298-0AD60-0BP3	C

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

Edition	Order No.	Remarks
09.04	6FC5298-7CA00-0BG1	C

Trademarks

SIMATIC®, SIMATIC HMI®, SIMATIC NET®, SIROTEC®, SINUMERIK® and SIMODRIVE® are Siemens trademarks. Other product names used in this documentation may be trademarks which, if used by third parties, could infringe the rights of their owners.

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 the contents of this manual for agreement with 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 all recommendations and suggestions.

Subject to change without prior notice

Preface

Structure of the Documentation	The SINUMERIK documentation is organized in 3 parts:	

The SINUMERIK documentation is organized in 3 parts:

- General Documentation
- User Documentation
- Manufacturer/Service Documentation

Audience

This document is designed for machine tool users. The document describes in detail all the technical facts an operator needs to understand to operate the SINUMERIK 840D, 840Di, 810D control systems.

Hotline

If you have any questions, please get in touch with our hotline:

A&D Technical Support

Phone.: +49 (0) 180 5050 - 222

Fax: +49 (0) 180 5050 - 223

If you have any questions about the documentation (suggestions, corrections etc.), please send a fax or e-mail to:

Fax: +49 (0) 9131 98 - 2176

E-mail: motioncontrol.docu@erlf.siemens.de

Fax form: See the reply form at the end of the document.

Internet address

http://www.siemens.com/motioncontrol

SINUMERIK 840D powerline

Since 09.2001, improved-performance versions of

- SINUMERIK 840D powerline and
- SINUMERIK 840DE powerline (export version)

have been available. The hardware description below contains a list of the available **powerline** modules:

References: /PHD/ SINUMERIK 840D Configuration Manual

SINUMERIK 810D powerline

Since 12.2001, improved-performance versions of

- SINUMERIK 810D powerline and
- SINUMERIK 810DE powerline (export version)

have been available. The hardware description below contains a list of the available **powerline** modules:

References: /PHC/ SINUMERIK 810D Configuration Manual

Validity

Thie Operator's Guide applies to the following controls:

SINUMERIK 840D   SINUMERIK 840Di   SINUMERIK 810D

with the handheld terminal (HT) 6, **SW 6.4**.

Standard scope

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

Please contact your local Siemens office for more detailed information about other SINUMERIK 840D, 810D publications and publications that apply to all SINUMERIK controls (e.g. universal interface, measuring cycles, etc.).

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

Validity

Catalog NC 60 is decisive in terms of the validity of the functions /BU/ Machine Tools, catalog NC 60

Layout of the descriptions

Wherever sensible and possible, all functions and operating options have been described using the same internal structure. By structuring the explanations in different information levels, we have made it possible for users to specifically access the required information.
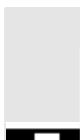
Explanation of symbols

Function

This theoretical part serves as a learning document, particularly for those who are new to the world of NC applications. Important information about understanding operating functions is offered to users at this point.
You should work through the manual at least once to get an overview of the operational scope and performance capability of your SINUMERIK control.

Operating sequence

This point shows the sequence of key operations which need to be executed. Any inputs or additional information required in the individual operating steps are described next to the pictures of the keys.

Other information

For safety reasons some functions are locked against unauthorized access. The machine manufacturer can influence or change the described functional behavior. Always refer to the information and specifications provided by the machine manufacturer.

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

Notes
This symbol is always displayed in this document to draw your attention to an important item of information in a more complex description.

In this documentation you will find the symbol shown on the left with a reference to an **ordering data option**. The function described will only be executable if the control contains the designated option.

Warnings

The following **warning notices** with varying degrees of significance are used in the document:

Danger
This symbol indicates that death, severe personal injury or substantial property damage **will** result if proper precautions are not taken.

Warning
This symbol indicates that death, severe personal injury or substantial property damage **may** result if proper precautions are not taken.

Caution
This warning (with the triangular symbol) means that minor physical injury **can** occur if the appropriate precautions are not taken.

Caution
This symbol (without a warning triangle) indicates that damage to property **may** result if proper precautions are not taken.

Notice

This warning means that an undesirable result or an undesirable state **can** occur if the information is ignored.

Reference notes

You will find this symbol wherever further information on a particular topic can be looked up in specialist literature.

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

Principle

Your SIEMENS 840D, 810D is designed using state-of-the-art technology in conformity with recognized safety regulations, standards and specifications.

Supplementary devices

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

Personnel

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

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

Procedure

Before the control is started up, personnel responsible for its operation must have read and understood the Operator's Guides. The company using this equipment is also obliged to carry out **continuous monitoring** of the overall technical condition of the equipment (with a view to identifying externally visible defects and damage as well as changes in the operating behavior of the control).

Service

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

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

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

Cases where the control is operated **in a technically imperfect condition**, without due provision for safety considerations and/or hazards, or in contravention with any of the instructions in the relevant documentation.

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

Any **modification, bypassing** or **decommissioning** of equipment on the control whose intended purpose is to ensure proper functioning, unrestricted use of equipment and/or active and passive safety.

Unforeseen danger can arise with regard to:
- life and limb of personnel,
- the control, machine or other assets of the owner and the user.

■

Table of Contents

Introduction

1.1 Functionality

1.1.1 General

SINUMERIK 840D, 840Di and 810D are CNC control systems (**C**omputerized **N**umerical **C**ontrol) for machine tools, handling systems and special machines.

The HT 6 has HMI and MCP functionalities and can be connected in place of or in addition to an HMI/MCP.

(HT 6 = **H**andheld **T**erminal with approx. **6**-inch display (measured diagonally),

HMI = **H**uman **M**achine **I**nterface,

MCP = **M**achine **C**ontrol **P**anel)

HMI functions which can be realized:

- Displaying states and messages,
- Programming parts programs via an editor,
- Changing machine data,
- Actual value display,
- Selection of parts programs,
- Displaying the program sequence chart,
- Teaching via teaching formulae
- Program correction, repositioning,
- Program control (SBL1, SBL2),
- Archiving and loading of data via RS-232-C,
- Acknowledging messages/alarms.

MCP functions which can be realized:

Starting and stopping programs,

Changing the mode of operation,

Manual traversing (jogging) (continuous/incremental),

Changing the override

- Resetting programs.

1.2 Operating areas

In the control system, the basic functions are grouped into the following operating areas (light areas in the diagram):
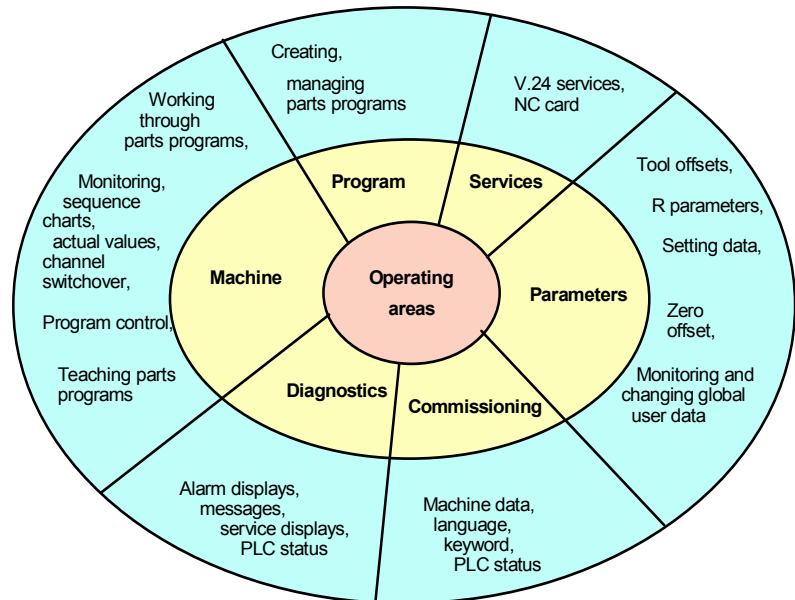
Operating areas chart

All functions can be accessed by the user via the user interface of the HT 6.

The user interface comprises:

- on the front
    - a display unit with 16 - 20 lines, each with 38 - 52 characters (depending on the settings selected by the machine manufacturer),
    - various blocks of input keys,
    - an override rotary button and
    - an EMERGENCY OFF button,

- on the rear
    - two agreement buttons

1.3 Handling information

Caution

The HT 6 must only be opened for service purposes by trained and qualified engineers.
Exception: The user may dismantle the upper part of the housing in order to replace the slide-in labels which are used to label the keys.

References
/BH/Operator Components, Manual,
Section: Handheld Terminal HT 6, Labeling of Slide-in Labels

Warning

Electronic components inside the HT 6 can be electrically destroyed if they are touched inappropriately.

Before using any operating devices on the HT 6:
- Please read through the explanations provided in this document first (particularly Section 2 "User interface", as this knowledge is a pre-requisite for understanding the remaining chapters).
- **Read the documents supplied by the machine manufacturer**.

Machine manufacturer

Please refer to the machine manufacturer's specifications.
Note

The degree of protection IP54 can only be ensured if all of the interfaces of the HT 6 are closed off with a cover.

∎

User interface

2.1 Introduction

User interface
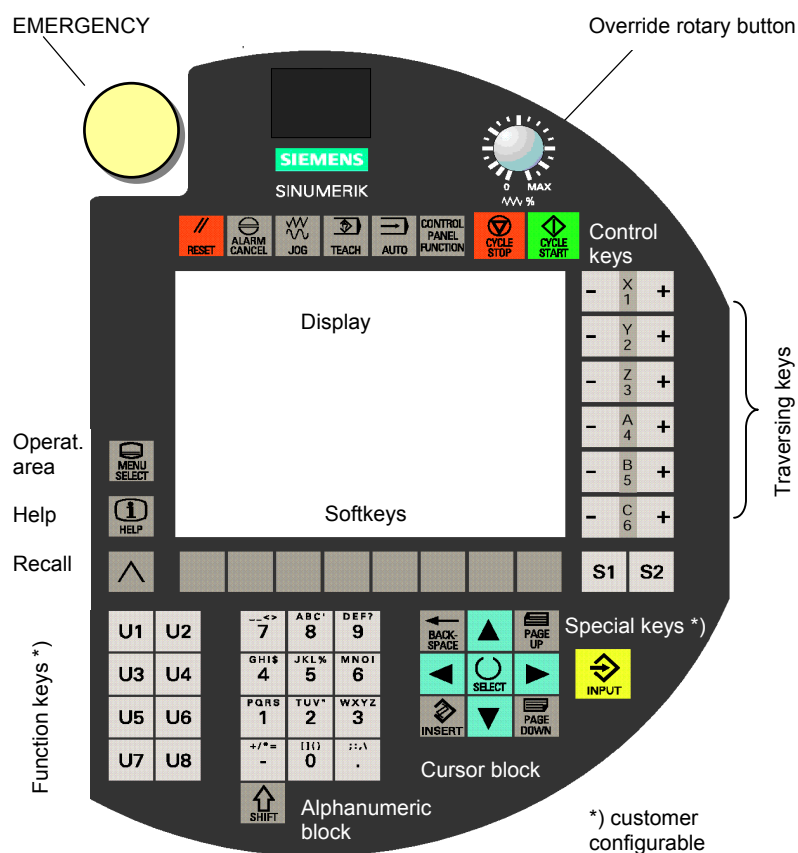
User interface of the handheld terminal HT 6

The user interface of the HT 6 comprises:

- EMERGENCY OFF button
- Override rotary button
- Control key block with machine functions
- Display
- (Jog) traversing keys
- "Operating area", "Help" and "Recall" buttons
- Row of softkeys
- Special keys S1/S2
- Function key block U1, ..., U8
- Alphanumeric block with switchover button
- Cursor block
- Input key

SINUMERIK 840D/840Di/810D Operator's Guide Handheld Terminal 6 (BAH) - 03.04 Edition

- PC memory
  card interface
- MF2 keyboard-
  interface
- Reset button
  (concealed)

V.24 interface
(concealed)

Agreement
buttons

Holding strap

Connecting cable
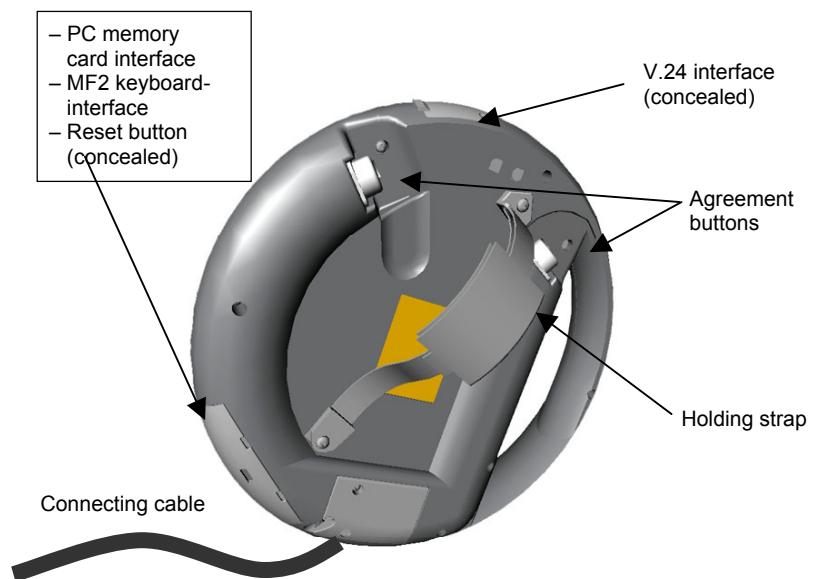
Rear view of handheld terminal HT 6

The following components can be found on the rear:

- underneath the bottom left cover:
  - PC memory card interface
  - PS/2 keyboard interface
  - Reset key
- underneath the top cover:
  - V.24 interface
- Two agreement buttons
- Connecting cable
- Holding strap

2.2 Display

The display consists of:

- the two-line status indicator
- the two-line alarm display
- the 8–13-line dialog window (depending on the displayed font size)
- the one-line dialog and system fault display with "Recall", "Help" and "Shift" symbol
- the two-line softkey display

2.2.1 Status indicator

Structure

```
FieldñNo. 1.   2.      3.   4.   5.   6.    7.    8.
        Ma  Auto  Teach MCS Trafo 100%  ASY2 Chan10   (Reserved
                                                      for machine
        Active       PROG56789012345678  SBL1 NCName  tool manu-
                                                      facturers)
Field No.   9.            10.          11. 12.      13.
```

Explanations 1. Operating area identifier

Ma   Machine
Pa   Parameters
Pr   Program
Di   Services
Di   Diagnostics
Ib   Commissioning
Te   Teaching

2. Operating mode (Auto, Jog, Mda)
3. Machine function (Teach, Ref, Repos)
4. JOG mode (MCS = Machine coordinate system, WCS = Workpiece coordinate system)
5. Transformation activated (Trafo). Only appears when Transformation is activated.
6. Override
7. Axis system on which the traversing keys act (ASY2, ASY3, ASY4).
   In the standard setting, an empty text is displayed for axis system 1 (traversing keys generally act on axes 1 – 6).
8. Channel name
9. Process status of the channel (active, stop, reset)

10. Name of the selected program (max. of 24 characters)
11. Single block mode (SBL1, SBL2). Only appears when Single Block Mode is activated.
12. Name of the NC to which the channel is assigned
13. This area can be used by the machine manufacturer (refer to the machine manufacturer's details).

2.2.2 Alarm display

Line: Alarm text or program messages
1. Line: Alarm text or program messages (e.g. Stop: EMERGENCY OFF active)

2.2.3 Dialog window

Without the frame, the dialog window has 316 x 139 pixels.
Depending on the selected font size, this corresponds to 8 – 13 lines.

2.2.4 Dialog line

Provides messages about currently active actions or actions which
have just been completed, e.g. channel operation messages (stop
conditions; see Note).

Start of line: Recall character **^**, if it is possible to return from the
current screen.

End of line: Help symbol **i**, if Help is available for the current
dialog window.
Shift symbol ⇑, if alpha input is activated.

Other information

The channel operation messages can be overwritten by other outputs.
There is no refreshing.

2.2.5 Softkey display

Displays the assignment of the softkey underneath.

2.3 Operating devices

The description below follows the diagram "User interface of handheld
terminal HT 6".

2.3.1   Machine control keys
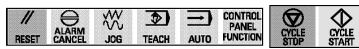
- **Reset** key
  - **Alarm acknowledgement** key
  - **JOG** operating mode key
  - **TEACH** sub-operating mode key
  - **AUTO** operating mode key
  - **CONTROL PANEL FUNCTION** key, see below)
  - **START** and **STOP** key
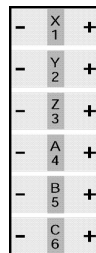
- Traverse (**Jog**) keys (can be labeled by the machine manufacturer in accordance with customer-specific requirements)

- With the Control Panel Function key you can only change the softkey bar:

Chan-nel	Trav.[1] keys	MDI	REPOS	REF	Step/ Go	MCS/ WCS	>

2nd level:

Gen. help						Exit [2]	>

[1]: only at MM_NUM_AX_SEL > 1;   [2]: only with 840Di

Meaning of the softkeys:

- **Channel**   Channel switchover (only if the number of channels > 1)
- **MDA**   → Operating mode *MDA*
- **REPOS**   → Machine function *Repositioning* (not selectable in the operating mode *Automatic*).
- **REF**   → Machine function *Reference position approach* (not selectable in the operating mode *Automatic*).
- **SBL on/**   Switchover between *Step* and *Go* mode.
  SBL off
- **MCS/WCS** Switchover from the machine coordinate system to the workpiece coordinate system for traversing by hand.
- **Gen. Help** General help information for operation of the HT 6.
- **Trav.keys** Axis assignment of the traversing keys (only if no. of axis systems > 1).

Notes

- The **Override** rotary button has 19 positions (amounts in %):
  0, 1, 2, 4, 6, 8, 10, 20, 30, 40, 50, 60, 70, 75, 80, 85, 90, 95, 100.
- Return to the original softkey bar using the **Recall** button.

Machine manufacturer

Please note:
The machine manufacturer may implement a different assignment.

2.3.2 Softkeys
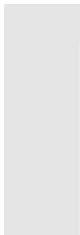
- The eight **softkeys** are assigned specific meanings according to operating area and operating mode. They can be seen from the softkey display.
  The softkey function can be activated by pressing or releasing the button (depending on the settings made by the machine manufacturer).

2.3.3 Free keys

S1	S2

- Special keys **S1/S2** *)

U1	U2
U3	U4
U5	U6
U7	U8
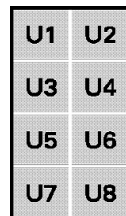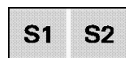
- Function key block **U1**, ..., **U8** *)

*) Can be adjusted and labeled by the machine manufacturer according to individual customer requirements.
During the adjustment it can be specified whether the reaction should take place when the button is pressed or released.

2.3.4 Alphanumeric block
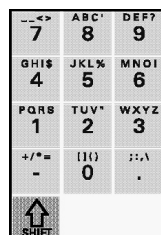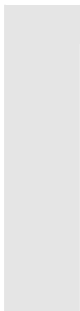
- Number keys
  The keys have multiple assignments. Special characters and letters can be selected by pressing the buttons repeatedly after switching over with the Shift key (see below) (similarly to text input on a mobile phone).

- Switchover (**SHIFT**) key letters $\leftrightarrow$ numbers
  (modal, only becomes effective on release)

2.3.5 Cursor block
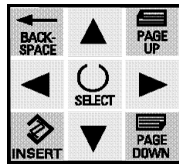
- The cursor block is for the most part self-explanatory.
  Additional explanations:
  - **INSERT**/Edit/Undo key
    In tables and input fields: Switching over to editing mode
    Within the editing mode: Same effect as the Undo key.
    Within an editing field: Switching between Insert and Overtype.
  - **SELECT** key:
    Selection of a value for an input field from specified values or
    from a selection list labeled with the corresponding key symbol.
    Activation or deactivation of a field (so-called toggle function,
    single or multiple selection).

- **INPUT** key:
  Acceptance of an edited value and, if only one dialog field is
  present, closing the dialog.

2.3.6 Other keys

- Operating area key

- Help key

- With the **Recall** key you can jump to the higher-level menu (if
  available).

2.3.7 Other operating devices

- The EMERGENCY-STOP button is laid out with two channels
  (safety category 4).
- Both agreement buttons (on the rear of the HT 6) are two-stage
  with two channels.

Agreement button
The "Agreement button" function is active when one of the two buttons
is pressed.
Example: In some cases one of the agreement buttons needs to be
pressed when the machine is manually traversed.

Note

Press the agreement button quickly as far as it goes to ensure it works properly.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

2.3.8   Menus/operating areas

The HT 6 can be powered up in different ways.
The screen displayed after power-up and the softkey assignment depend on the machine function selected at the controls (REPOS, REF) and the operating mode (JOG, AUTO, MDA).

Machine manufacturer

Please refer to the machine manufacturer's specifications.
After pressing the operating area button the operating areas appear in the softkey display:

Ma-chine	Para-meter	Pro-gram	Services	Diagno stics	IBN		

The softkeys are then used to change to the main menus for the corresponding operating areas.

Options Optional functions are marked in the document.

Note

Under certain circumstances it is possible that the softkey assignment for the main menu might have been changed by the machine manufacturer.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

■

Parameters operating area

3.1 Introduction

Before you start working with the CNC, set up the machine, tools etc. on the CNC:

- Input/change the tool data (tool offsets),
- Input/change the R parameters,
- Input/change the setting data,
- Input/change the zero offset,
- Input the user data.

The setup functions are collected together in the operating area *Parameters*. To get to this area, click on:

- the operating area button,

- the **Parameter** softkey.

When you select this operating area for the first time the *Parameters Operating Area* working window appears. When you subsequently change into this operating area the last selected working window will appear.

The following operating areas are available via the softkeys:

- Tool offsets
- R parameters
- Setting data
- Zero offset
- User data

Press the desired softkey to activate one of the functions described below.

3.2   Tool offset

3.2.1   Overview

Function

The tool offsets consist of a series of data which describe the geometry, the wear, the tool number and the tool type. Depending on the tool type, each tool is assigned a fixed number of parameters. Tools are identified with a T number, cutting edges with a D number.

Precondition

- The **Tool offset** softkey has been pressed in the *Parameters Operating Area* working window. The following prompt appears: "Go to tool:" Enter the known tool number in the input field and press Input to confirm
  or
- The **Overview details** softkey has been pressed.

The following working window appears:

```
Overview of tool offsets
Go to tool:                   0
T no. Type                Name              Cut edges
     1 200    Twist drill                        1
     3 120    End mill (w/o corner roundin        1
     4 510    Finishing tool                     1
     5 120    End mill (w/o corner roundin        1
     6 510    Finishing tool                     1
     7 510    Finishing tool                     1
∧
Detail│      │      │      │Delete│      │New
view  │      │      │      │tool  │      │tool
```

Tool Offset Overview working window

Operating sequence
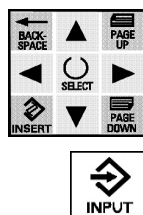
- Select the desired tool with the cursor or enter the tool number in the input field.

- Press Input to go to the *Tool Offset Parameters* working window.

Further buttons

Delete Tool	Deletes the currently selected tool
New Tool	Creates a new tool
	Create New Tool working window

3.2.2 Correcting tool data

Function

Tool data can be displayed and modified.

Prerequisites

- The desired tool has been selected via the **cursor** in the *Tool Offset Overview* working window or by direct input of the tool number and
- the Input button has been pressed to jump to the desired tool.

The following working window is displayed:

```
Tool offsets                    TO area     1
T no.  3       D no.  1
120 End mill (w/o corner rounding)
No. of cuttng edges:   1
                Geometry     Wear        Base
Tool length comp.
Length 1   :     58.181       0.000       0.000 mm
Length 2   :     90.080       0.000       0.000 mm
Length 3   :      0.000       0.000       0.000 mm
Λ
T no. |T no. |D no. |D no. |De-  |      |New  |
-     |+     |-     |+     |lete |      |     |
```

Tool Offset Parameters working window

Operating sequence

- Select the parameters with the cursor.

- Use the numerical input to modify the parameters (permitted characters: digits, decimal point, minus sign).
- Press **Input** to accept the value and jump to the next input field.

Further buttons

T No. -/+	Offset data for the next lowest/highest tool
D No. -/+	Offset data for the next lowest/highest cutting edge
Delete	Selected tool/cutting edge is deleted after a safety prompt
New	Offset data are created for a new tool/cutting edge

3.2.3 Create new tool/new cutting edge

Prerequisites

- The softkey **New** has been pressed in the *Tool Offset Parameters* working window.

The following working window is then displayed:


```
Tool offsets                    TO area    1
T no.  1        D no.  1   .
200 Twist drill
No. of cuttng edges:   1
                Geometry     Wear        Base
Tool length comp.
Length 1  :        0.040      0.000       0.000 mm
Length 2  :      154.217      0.000       0.000 mm

Delete edge or tool?
Λ
Cuttng      Tool
edge
```

Tool Offsets – New working window

Operating sequence

- **Cutting edge** softkey   In the "*Create New Cutting Edge*" working window enter the D no.

- **Tool** softkey   In the "*Create New Tool*" working window enter the T no. and (if appropriate) the tool type

- Click on the **OK** softkey each time to confirm and return to the starting window.

Further buttons

Abort (softkey)　　　Cancel input or

Recall　　　Return to the starting window without
creating a cutting edge or new tool

3.3 R parameters

Function

R parameters can be displayed and modified.

Prerequisites

- The **R Parameters** softkey has been pressed in the *Parameters Operating Area* working window.

The following working window is then displayed:

R variables			
R0	5.00000000	R8	0.00000000
R1	0.00000000	R9	0.00000000
R2	5.00000000	R10	3.00000000
R3	-6.00000000	R11	0.10000000
R4	67.00000000	R12	1.50000000
R5	24.00000000	R13	1200.0000000
R6	12.00000000	R14	123.00000000
R7	0.00000000	R15	2.00000000

Λ

			Delete all	Find		

R parameters working window

Operating sequence
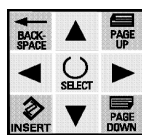
- Select the R parameters with the **cursor.**

- Use the numerical input to modify the values.

- Press **Input** to accept the value (or exit the field to do this).

Further buttons

Delete all　　　Sets all R parameters to 0.000 (after a safety prompt)

Search　　　*R Parameter Search* working window: Search for a desired R parameter by inputting its number

3.4   Setting data

Function

The setting data can also be displayed and modified in a similar manner to the R parameters. There are:

- General setting data
- Channel-specific setting data
- Axis-specific setting data

One of these types can be selected after pressing the **Setting Data** softkey in the *Parameters Operating Area* working window.

Operating sequence

Use one of the softkeys to choose the desired type of setting data:

General SD   *General Setting Data* working window
Channel SD   *Channel-specific Setting Data* working window
Axis SD   *Axis-specific Setting Data* working window

Further buttons

Recall   Return to the *Parameters Operating Area* working window

3.4.1   General setting data

Prerequisites

- The softkey **General SD** has been pressed in the *Setting Data* working window.

The working window *General Setting Data* ($SN_) is opened:

```
General setting data
41010              0.00000000
41050                       1
41100                       0
41110              0.00000000 mm/min
41120              0.00000000 mm/rev
41130              0.00000000 rpm
41200              0.00000000 rpm
41300[0]                    0
∧ JOG_CONT_MODE_LEVELTRIGGRD
                              Find    Find
                                      next
```

General Setting Data working window

Depending on the cursor position, the corresponding name is displayed parallel to the setting data or machine data in the dialog line.

Operating sequence
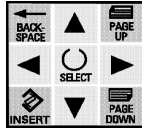
- Select the setting data with the **cursor.**

- Use the numerical input to modify the data.

- Press **Input** to accept the value (or exit the field to do this).

Further buttons

Search	An additional window opens in the lower third of the working window. In this window you can enter the name or partial name (search string) or the number to look for.
Continue search	Searches again for the same search string
Recall	Returns to the *Setting Data* working window

3.4.2 Channel-specific setting data

Prerequisites

- The softkey **Channel SD** has been pressed in the *Setting Data* working window.
The Channel-specific Setting Data working window opens.
All of the channel-specific setting data are listed.
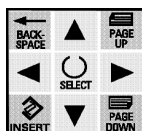
Operating sequence

- Select the setting data with the **cursor.** (The machine data name is displayed in the dialog line).

- Use the numerical input to modify the data.

- Press **Input** to accept the value (or exit the field to do this).

Further buttons

Channel -/+	Previous/next channel number
Search	Search for a number or a (part of a) name
Continue search	Searches again for the same search string
Recall	Returns to the *Setting Data* working window

3.4.3   Axis-specific setting data

Prerequisites

- The softkey **Axis SD** has been pressed in the *Setting Data* working window.

The *Axis-specific Setting Data* working window opens.

All of the axis-specific setting data are listed.
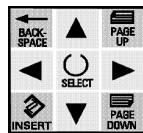
Operating sequence

- Select the setting data with the **cursor.** (The machine data name is displayed in the dialog line).

- Use the numerical input to modify the data.

- Press **Input** to accept the value (or exit the field to do this).

Further buttons

Axis -/+	Previous/next axis number
Search	Search for a number or a (part of a) name
Continue search	Searches again for the same search string
Recall	Returns to the *Setting Data* working window

3 Parameters operating area
3.5 Settable zero offset
03.04 **3**

3.5 Settable zero offset

3.5.1 Overview (ZO)

Function

After the reference point approach the actual value memory and therefore also the actual value display relate to the machine zero point. However, the parts program of the workpiece relates to the workpiece zero point.

This offset should be entered as a settable zero offset (ZO) via the following functions:

- **Offset**
- **Rotation**
- **Scale**
- **Mirroring**

Precondition

- The **Zero offset** softkey has been pressed in the *Parameters Operating Area* working window.

The following working window is then displayed:

```
Overview of all settable WOs
Go to WO:                    2
$P_UIFR[Index]   G code
           1     G54
           2     G55
           3     G56
           4     G57
           5     G505
           6     G506
⋀Press Input key to select
```

Working window: Zero offset overview

Operating sequence
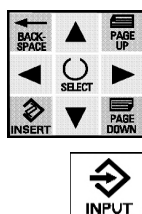
- Select the desired zero offset using the **cursor** keys or input the ZO index into the empty input field.

- Press **Input** to jump to the *Zero Offset (ZO)* working window.

3.5.2   Offset (ZO)

Function

The offset can be entered or modified for each of the three coordinate axes.

Precondition

- The desired ZO has been selected from the *Settable ZO Overview* working window. The working window shows the axis and the offset values of the selected G-identifier.

E.g. $P_UIFR [ ]: This identifier can be used to change a settable zero offset in the program.

Operating sequence
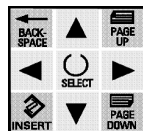
- Select the parameters with the **cursor**.

- Use the numerical input to define the coarse and fine offset values.

- Press **Input** to jump to the next input field.

- Click on the **Save** softkey to accept the entries.

Further buttons

ZO -/+ Previous/next ZO

Rot./Scale   Use this softkey to access the *Rotation/Scale/Mirroring* working window.

Refer to Chapter: Rotation/Scale/Mirroring (ZO).

Cancel   Cancels the entries.

Recall   Returns to the *Settable ZO Overview* operating window

Other information

If any of the values are changed or entered in the working window *Settable ZO Overview* and afterwards in the working window *Rotation/Scale*, then all values are accepted during saving.

3.5.3   Rotation/Scale/Mirroring (ZO)

Function

Rotation, scale and mirroring can be entered or modified for each of the three coordinate axes.

Precondition

- The softkey **Rot./Scale** has been pressed in the ZO – Offset working window.

The following working window is then displayed:

```
Settable work offset
$P_UIFR[   1]  G ID:      G54
Axis     Rotat'n          Scale     Mir-
          (deg.)                     ror
X            0.000        1.000      ⊠
Z            0.000        1.000      ■
CC                        1.000      □
AWZ                       1.000      □
C8                        1.000      □
Λ
WO    WO         Offset              Re-   Save
-     +                              ject
```

Working window: *Settable ZO – Rotation/Scale/Mirroring*

Operating sequence
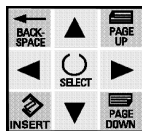
- Select the input field with the **cursor**.

- Change the angle of **rotation** or the **scale** numerically.

- Press **Input** to jump to the next input field.

- Switch the **mirroring** on or off using the **Select** key.

- Click on the **Save** softkey to accept the entries.

Further buttons

ZO -/+	Previous/next ZO
Offset	Use this softkey to access the *Offset* working window. Refer to Chapter: Offset (ZO)
Cancel	Discards the entry.
Recall	Returns to the *Settable ZO Overview* operating window

3.6   User data

3.6.1   Overview

Function

User data (Global User Data, GUD) can be used to globally define certain states for a machine, e.g. the return plane or the clearance distance.
The user data are stored in the directory "/_N_DEF_DIR".

References

More information on the subject of "User data" can be found in /BA/ SINUMERIK Operating Instructions, User Data and /PGA/ SINUMERIK Programming Guide, Job Planning

The following user data can be displayed and modified:

- **global**
- **channel-specific**
- **local**
- **program-global**

Precondition

- The **User data** softkey has been pressed in the *Parameters Operating Area* working window.

Operating sequence

- Use the softkey to select the desired type of user data.

Global	*Global User Data* working window
Channel-spec.	*Channel-specific User Data* working window
Local	*Local User Data* working window
Program	*Program-global User Data* working window

Machine manufacturer

Please refer to the machine manufacturer's specifications.

SINUMERIK 840D/840Di/810D Operator's Guide Handheld Terminal 6 (BAH) - 03.04 Edition   3-41

3.6.2 Displaying and modifying user data

Function

The procedure for displaying and modifying global, channel-specific, local and program-global user data is similar and is therefore dealt with jointly here for all of the user data types.

Precondition

In the *User data* working window you can opt to select the **Global**, **Channel-spec., Local** or **Program** softkey.

Analogously to: Working window - channel spec., local, program-global user data.

Exception: With local and program-global user data the data number in the title bar and the softkeys **GUD-** and **GUD+** are omitted, as there is only one data list.

Operating sequence
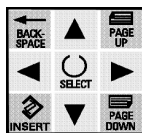
- Select the desired variable with the **cursor**.

- Use the numerical input to modify it.

- Press **Input** to accept the value (or exit the field to do this).

Further buttons	(Does not apply to *local/program-global user data*)
GUD -/+	Previous/next data record
Search	*Search User Data* working window
Continue search	Continues the search
Recall	*User Data Overview* working window

3.6.3 Search user data

Function

In order to find a user variable you can either enter the name complete or in part (as a search string).

Precondition

- The **Search** softkey in the *Global/Channel-specific/Local or Program-local User Data* working window has been pressed.

An additional *User Data Search* working window opens in the lower half of the screen.

Operating sequence

- Enter the search string or number after "Name or number of the user data".

- Click on **OK** to start the search.

Further buttons

∧	**Cancel** or **Recall**	Cancel the input and jump back to the starting window

■

Program operating area

4.1   Directories

When you select the *Programming* operating area for the first time, an overview of the standard directories is displayed.
Within these directories you can create new files for programs required to machine the workpiece, as well as organizing, copying, renaming, deleting and editing existing files.

Notes

- For reasons of space the workpiece name and file name are not always displayed in full.
- If you leave the programming level for the *Programming* operating area and then call it up again afterwards you automatically jump to the last selected directory.

Prerequisites

To execute the functions for organizing and editing part programs described in this section, click on

MENU
SELECT

- the operating area button,

Pro-
gramm

- the **Program** softkey.

The following working window is then displayed:

```
Program operating area
WPD: Workpieces
MPF: Global part programs
SPF: Global subprograms
CST: Standard cycles
CUS: User cycles
CMA: Manufacturer cycles


WPD   MPF   SPF   CST   CUS   CMA
```
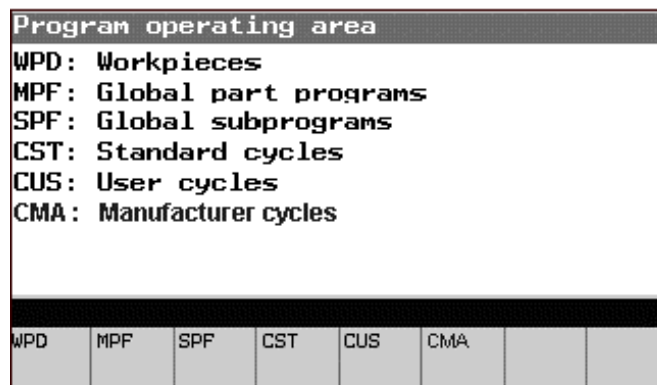
Standard Directory Overview working window

To call up the contents of the individual directories, select one of the softkeys **WPD**, **MPF***,* **SPF**, **CST, CUS** or **CMA**.
If, for example, you select **WPD**, the working window *Workpiece Overview* will pop up.

After pressing the 8th softkey, the expansion key, you will be presented with another softkey bar containing the softkeys **CLP** and **Save Info**.

CLP   Calls up the contents of the clipboard

Save Info   Displays the NC memory

```
Program operating area




CLP: Clipboard
MemoryInfo: NC memory information



                                        CLP   Memory
                                              info
```
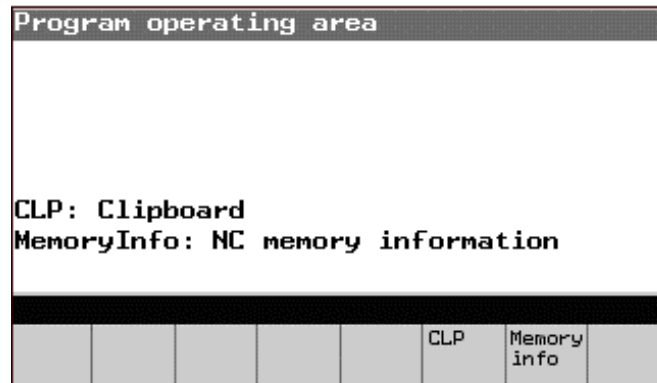
4.1.1 Workpiece overview

Function

After you select the **WPD** softkey the *Workpiece Overview* working window is displayed.. All of the existing workpieces are listed with their name, type, date and release.

Other information

The following softkeys are available:

New   Creates a new workpiece directory

Copy   Copies the selected workpiece to the clipboard.

Paste   Inserts a workpiece which was previously copied to the clipboard

Delete   Deletes the selected workpiece (after a safety prompt)

Renaming   Allows the currently selected workpiece to be renamed.

Enable   The selected file receives/loses the enable identifier. This is indicated with an "X"; the program can be worked through.

Selection   *Workpiece Selection* working window

Further buttons

Recall *Standard Directories Overview* working window

Input *Workpiece Selection* working window

4.1.2 Create new workpiece

Function

To create a new workpiece, a working window opens up requesting the input of the workpiece name.
In this new, empty directory you can create files for the programs which are required for machining of the workpiece.

Precondition

The softkey **New** has been selected in the working window *Workpiece Overview*. An additional *Create New Workpiece* working window then opens in the lower half of the screen.

Operating sequence

• Type in the new name.
The type has already been set by default to "workpiece" (WPD).
• Click on the softkey **OK** to create the workpiece directory.
In this empty directory you can create a new file and edit it.

Further buttons

Abort (Softkey) or

Recall Exits the working window <u>without</u> creating a workpiece

Other information

Creating a workpiece template If there is no "_TEMPL_" for a workpiece then a workpiece needs to be created with the name "_TEMPL_ ".

If a workpiece is already present with the name "_TEMPL_" then this workpiece is copied and renamed when the "New" function is used.
All of the files in this directory are also copied into the new directory in the process.

Example Workpiece: _TEMPL_

        TEMPL.MPF

        DATEN.INI

        TEST.MPF

Press the softkey **New**

Name: Specify "ACHSE" (*Axis*)

A new workpiece called "Achse" (*Axis*) is then created with the following files:

        ACHSE.MPF

        DATEN.INI

        TEST.MPF

4.1.3 Create new file

Function

To create a new file under a workpiece directory, a working window is opened which asks you to input the file name. In this new, empty file you can edit the programs which are required for machining of the workpiece.

Prerequisites

A workpiece has been selected with the **cursor** keys in the working window *Workpiece Overview* and the **Input** key has been pressed to move to the directory of the selected workpiece.
Call up the working window *Create New File* by clicking on the **New** softkey (initially without a file name).

Operating sequence

- Use the **Select** button to determine the file type

- Type in the new name.
- Click on the softkey **OK** to create a new file.

This file is empty, and you can use the *Editor* to write in it.

Further buttons

Abort   (Softkey) or

Recall   Exit the working window <u>without</u> creating a file.

4.1.4   Parts programs/subroutines

Function

The Program main screen contains a complete overview of all parts programs and subroutines.

Precondition

In the *Standard Directories Overview* working window one of the following softkeys has been selected:

MPF   for global parts programs or, analogously,

SPF   for global subroutines.

Operating sequence

- Use the corresponding softkey to activate one of the following functions:

New   An additional *Create New File* working window opens in the lower half of the screen:

```
Part programs
Name               Type  Size  Date     Enable
CMM_MDA            MPF      5 02.05.02  X
CMM_SINGLE        MPF     51 02.05.02  X
E_C950_LOG        MPF   5796 02.05.02  X
INPUT_DATA_MM     MPF   3617 03.05.02  X
LOG_F_SP_RP       MPF    198 02.05.02  X
NRKDUMP           MPF   8849 02.05.02  X
Create new file
Name:    test
Type:    Part program (MPF)
∧
                                         Abort  OK
```

Parts Programs, New File working window

Enter the file name (up to a maximum of 24 characters).
The type is set by default.

Copy	Copies the selected parts program or subroutine onto the clipboard
Paste	Inserts a parts program or subroutine which was previously copied to the clipboard.
Delete	Deletes the selected parts program or subroutine.
Renaming	Allows the selected parts program or subroutine to be renamed.
Enable	The selected parts program or subroutine receives/loses the enable identifier. This is indicated with an "X": The program can be worked through.
Selection	The parts program is selected on the current channel.

Further buttons

∧	**Recall**	*Standard Directories Overview* working window
⬦ INPUT	**Input**	*Directory of the Selected Parts Program* working window

4.1.5 Cycles

Function

Cycles are subroutines which are used to perform repetetive machining processes on the workpiece.

Precondition

In the *Standard Directories Overview* working window one of the following softkeys has been selected:

CST for the standard cycles or, analogously,
CUS for user cycles
CMA for manufacturer cycles.

The Cycles main screen contains a complete overview of all standard cycles (CST) and, analogously, all user cycles (CUS) and manufacturer cycles (CMA).

Operating sequence

The following softkeys are available:

New	Creates a new cycle directory
Copy	Copies the selected cycle to the clipboard
Paste	Inserts a cycle which was previously copied to the clipboard
Delete	Deletes the selected cycle (after a safety prompt)
Rename	Allows the currently selected cycle to be renamed
Enable	The selected file receives/loses the enable identifier. This is indicated with an "X"; the program can be worked through
Selection	*Cycle Selection* working window

Further buttons

Recall *Standard Directories Overview* working window

Input *Cycle Selection* working window

4.1.6   Memory information

Function

This function enables you to have the entire available memory displayed.

Operating sequence

In the *Standard Directories Overview* working window the softkey **Mem Info** has been selected:

The following working window opens:

```
Program operating area
WPD: Workpieces
MPF: Global part programs
SPF: Global subprograms
```

NC main memory			
	Total	Assigned	Free
NC memory	2064384	1836524	227860
Directories	30	29	1
Files	400	262	138

Memory Info working window

4.2   Editor in normal mode

Function

You can make alphanumeric changes in the editor. The standard operator functions of an editor are available.
The selected program can also be opened when active. Blocks which are not disabled, i.e. ones which have not yet been interpreted, can be changed online.
If the selected program has already been opened by another application then the following message will appear when an attempt is made to open it again:
"This program has already been opened by a different application".
The following message appears if an attempt is made to edit a disabled block: "No write access to the data".

Prerequisites

- The **Input** button has been pressed in a directory.

If for example the starting point was a program directory (**MPF** or **SPF)**, the following working window could appear:

```
Editor LOG_F_SP_RP.MPF                    1
--------------------↵
Logfile F_SP_RP: 14.3.2 11:56:57↵
Wertefolge: X/2↵
RP: 17/2/1EX+300/1EX+300/XR:1EX+300↵
Letzte Richtung: -2↵
WWP: 279.04/442.6892/0(0=WKS;1=MKS)↵
Zum Werkzeugwechsel: 220.33/398.38924↵
==eof==



Λ
Edit            Recom-  Go to  Find   Find   Close
                pile                  next
```
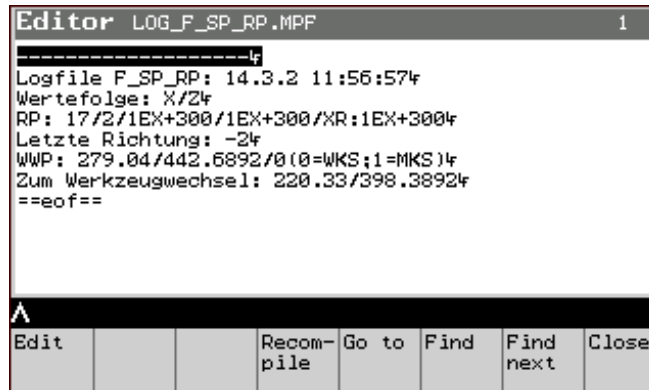
Working window *Editor in Normal Mode*

Operating sequence

Activate the editor functions with the softkeys as described below:

Edit	Editor mode (*Edit Program* working window)
Recompile	The changes are not saved. (This softkey only appears if there is a corresponding entry in the Comm.com file).
Go to	Exact details of the row and column
Search	*Text Search* working window: Asks for a string and then searches for it in the file.
Continue search	Searches for the last string again
Close	Exits the editor and returns to the previously selected directory. The content of the last row is saved. The content of the other rows is saved as soon as you leave the row.

4.2.1   Edit program

Function

The working window *Editor in Normal Mode* is displayed unchanged; only the softkey bar changes to the editing functions:
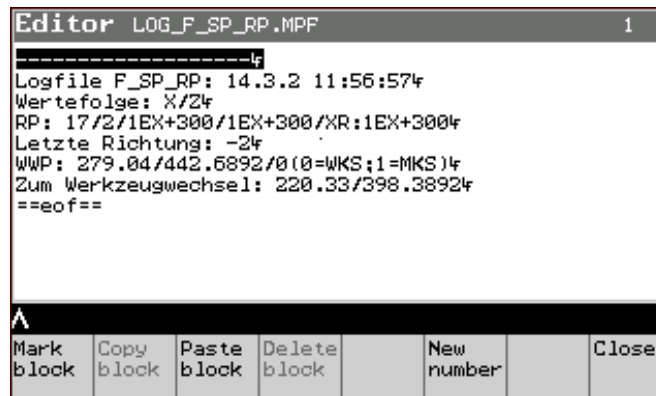
```
Editor LOG_F_SP_RP.MPF                                    1
--------------------↲
Logfile F_SP_RP: 14.3.2 11:56:57↲
Wertefolge: X/Z↲
RP: 17/2/1EX+300/1EX+300/XR:1EX+300↲
Letzte Richtung: -2↲
WWP: 279.04/442.6892/0(0=WKS;1=MKS)↲
Zum Werkzeugwechsel: 220.33/398.3892↲
==eof==
```

| Mark | Copy | Paste | Delete | | New | | Close |
| block | block | block | block | | number | | |

Edit Program working window

Operating sequence

Activate the editor functions with the softkeys as described below:

Mark block	Start/discard text marking; softkey is inverted as long as marking mode is switched on;
Copy block	Copy the selected block to the clipboard; Exit marking;
Insert block	Insert the saved block (letters are lighter when no block is saved);
Delete block	Delete the selected block (after it has been saved),
New number	Renumber blocks. Select block number input in stages.

Other softkeys: same as in the *Editor in Normal Mode* working window

Further buttons

| **Recall** | Discard markings; → *Editor in Normal Mode* working window |
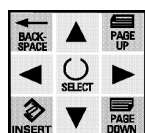
Cursor keys	Move within the text
Backspace	Deletes the character to the left of the cursor.
Insert	Switch between insert/overtype modes

Input   New line after the cursor

Any changes and inputs are accepted whenever you leave a line.

4.2.2   Text search

Function

You can search for any sequence of characters (string) in the program to be edited.

Prerequisites

The softkey **Search** has been pressed in the *Editor in Normal Mode* working window.
The *Text Search* working window opens in the lower half of the screen.
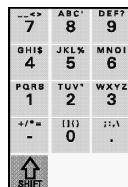
Operating sequence

- Enter the string you wish to search for in the **Search** line.

- Click on the **OK** softkey to confirm.
The editor searches for the entered string and jumps to the point where it finds it.

Further buttons

Abort   (Softkey) or
Recall   Cancels the search and returns to the *Editor in Normal Mode* working window.

4.2.3 Positioning the cursor

Function

If the target row and column of a program are known then this function can be used to jump there directly.

Prerequisites

The softkey **Go To** has been pressed in the *Edit Program* working window.
The *Go To* working window opens in the lower half of the screen.

Operating sequence
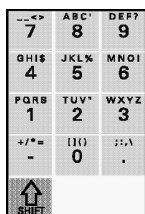
- Enter the target row and column in the corresponding fields.

- Click on the **OK** softkey to confirm.

The editor searches for the desired row and column and then jumps there.

Further buttons

Start of file	Positions the cursor at the start of the file
End of file	Positions the cursor at the end of the file
Abort	(Softkey) or
Recall	Cancels the positioning process and returns to the *Editor in Normal Mode* working window.

■

Manual Mode – Operating Area Machine

5.1 Introduction

The NC can be operated by hand control to

- traverse the axes step-by-step (*JOG* operating mode),
- synchronize control and machine (*reference point approach* function), reposition the axes (after they have been traversed, e.g. during a program interruption, *REPOS* function),
- modify a program or subroutine (*TEACH-IN* operating mode).
- In manual mode you can
- adjust the step width of the **Jog** (traversing) buttons (*incremental traversing* function),
- adjust the traversing speed (*Override* function).

Precondition

System status either

- **Stop** or

- **Reset**

5.2 Jog operating mode

Function

Following a program interruption in automatic mode (e.g. to take a measurement on the workpiece and correct the tool wear values or because of tool breakage), manual repositioning of the tool away from and back towards the contour path is possible after changing to *Jog* mode.

Prerequisites

- **Jog** function key has been pressed:
The second field in the first status bar of the working window changes to **Jog**.
If you then press
- the **operating area** key and

- the **machine** softkey

the *WCS Actual Values* or *MCS Actual Values* working window will normally appear, depending on the chosen coordinate system.

If you change into this area later on then the above working window is not necessarily displayed, as the last selected working window pops up instead.

Caution

When the system is switched on, the actual value display will not correspond to the actual positions of the axes.
You should therefore ignore the actual value display until the axes have been referenced.

```
WCS        POSITION        Repos offset
   X      -111.000   mm        0.000
   Z         0.000   mm        0.000
   CC        0.000   Deg.      0.000
   AWZ       0.000   Deg.      0.000
   C8        0.000   Deg.      0.000
   Z2        0.000   mm        0.000




         ActVal                         INC
         MCS
```

WCS Actual Values working window

The first six axes in the WCS coordinate system are visible. It is possible to scroll up and down if there are more than six axes.
The names of the axes are defined by the machine manufacturer in the machine data.

- **St** (status) and **Tu** (turn) indicate the position of the axis and the sign of the axis rotation, both as 8-bit information.
  They only appear
  - in the status "WCS actual values" and
  - if PTP (point-to-point) traversing is supported by the NC.
- **Increment** only appears when incremental traversing is active.
- *Automatic* and *Mda* operating modes:
  - The **distance-to-go** is displayed instead of the **repos. displacement** (provided the selected machine function is not *Reposition*).
  - The recall sign **^** appears in the dialog line on the left.
  - The symbol ⤒ appears at the top left ("all axes in exact positioning mode").

Operating sequence

CONTROL PANEL FUNCTION

After pressing the **Control Panel Function key**, press the softkey

Traverse to select the axis system.

Machine manufacturer

The machine manufacturer configures which axes are assigned to the axis system.
Please refer to the machine manufacturer's specifications.

| – | X 1 | + |
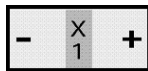
- Use the **Jog** key to traverse the desired axis by pressing the + or - side.

The axis is traversed, and the corresponding value is updated on the display.

The traversing process stops when the key is released or when a preselected stop is reached.

In contrast to MCP, it is possible for several axes to be traversed simultaneously.

You can influence the traversing speed with the **Override** rotary button.

Softkeys

Act.Val. MCS	→ *MCS Actual Values* working window
INC	→ *Increment Selection* working window

5.3   Reference point approach

Function

After the system is switched on the actual value display will not correspond to the actual positions of the axes. The *Ref* function (reference point approach) is used to synchronize the control and the machine.

Prerequisites

After pressing

CONTROL
PANEL
FUNCTION

- the **Control Panel Function** key and

REF

- the **REF** softkey

the following working window appears (= startup window after booting, see Section "Start-up"):

```
Reference point approach
        MCS      POSITION
   ⊕    X1          0.000   mm
   ⊕    Z1          0.000   mm
   ⊕    C1          0.000   Deg.
   ⊕    AWZ1        0.000   Deg.
        C8          0.000   Deg.
        Z2          0.000   mm
```

Reference Point Approach working window

In addition, the following symbols may also appear next to the axes:

⊕ - Reference point approach already performed for this axis.

○ - Reference point approach yet to be performed for this axis.

No symbol - Reference point approach does not need to be performed for this axis.

The ***User Agreement*** sub menu only appears if the *Safety Integrated* option is selected and at least one axis is present which needs to be approved.

Operating sequence

Use the **Jog** key to traverse the desired axis by pressing the + or - side.

However, the axis will only react if the chosen direction matches the configuration. If it does then the axis is traversed to the reference point and the actual value display is updated.
Several axes can be traversed at the same time here as well.

Softkeys

User   Calls up the *User Agreement* working window (only
Agreem.   if the *Safety Integrated* option is selected):
The display shows the machine axes in the MCS, the current position and a check box for "Activate/deactivate approval".

Operating sequence
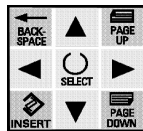
- Position the cursor on the desired machine axis.

- Use the **Select** button to activate/deactivate approval for the selected machine axis.

Check whether a reference point approach has been performed on the relevant axis. If not, the error message "Perform reference point approach first" is displayed. Approval cannot be granted for the axis until the reference point approach has been performed on the axis. The User Approval function is only offered if user approval is required on at least one axis of the channel.

References   For more information, please refer to the following documents:
/BA/ SINUMERIK Operating Instructions, Machine Operating Area
and
/FBSI/ SINUMERIK Description of Functions, Safety Integrated.

5.4 Repositioning

Function

In the *Jog* operating mode the control saves the coordinates of the interruption point. The path differences of the axes traversed in *Jog* mode can be displayed.

After the interruption the displacement is reversed with the **Jog** keys (i.e. the axis is repositioned).

Precondition

CONTROL PANEL FUNCTION

- **Control Panel Function** key and

REPOS

- **REPOS** softkey have been pressed.

Operating sequence
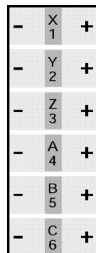
- Use the **Jog** key to traverse the desired axis by pressing the + or - side.

The motion stops when the *Repos* value reaches zero.

Several axes can be traversed at the same time here as well.

5.5 Increment

Function

The increment is the step width by which the selected axis is traversed each time the associated **Jog** key is pressed. It can also be modified either via the keyboard or via a softkey.

Precondition

INC

- The **Increment** softkey has been pressed in the *MCS Actual Values /WCS Actual Values* working window.

The following working window is then displayed:

```
 MCS      POSITION         Dist-to-go
 X1         0.000  mm         0.000
 Z1         0.000  mm         0.000
 C1         0.000  Deg.       0.000
 AWZ1       0.000  Deg.       0.000
 C8         0.000  Deg.       0.000
 Z2         0.000  mm         0.000
Increment selection
Variable increment val.:        0

1    10    100   1000  10000         Abort  OK
```

Increment Selection working window

Operating sequence

- Input the value for the increment numerically.

Alternatively:

- Use the desired **softkey** to select an increment value (e.g. 1, 10, 100, 1000 or 10000),
- Click on the **OK** softkey to accept the value and return to the previously active working window.

Further buttons

Recall or	Returns to the previously active working window
Cancel softkey	To cancel without accepting the value.

5.6   MDA operating mode

Function

This operating mode is intended for use in semi-automatic operation for traversing an axis to a certain position with individual NC commands, calling programs or executing M functions, without needing to write a full NC program.

Precondition

The status of the system is

- **Stop** or **Reset**.

- The **Control Panel Function** key and the **MDA** softkey have been pressed.

The following working window is then displayed:

```
MDI program
Def INT TEST↵
F1000↵
loop↵
Y12 = 0↵
Y1 = 10↵
endloop↵
==eof==




                  Act.  Contrl Delete              Save
                  val.  progr. MDI                 MDI
```
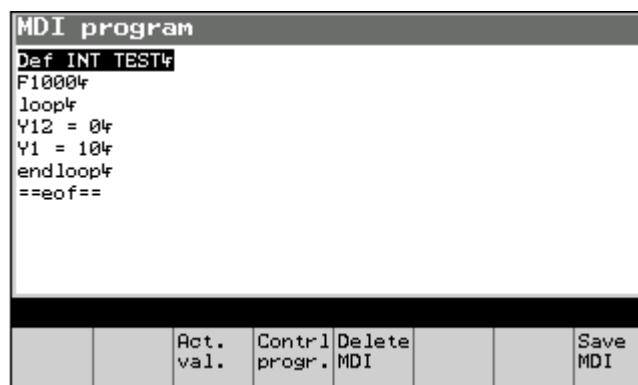
MDA working window

Operating sequence

- Type in the program line.

- Finish by pressing the **Input** button.

To execute the NC commands

- Start the NC.

Caution

The control can perform unwanted motions if incorrect inputs are entered.
Therefore this operating mode should only be used by trained personnel.

Softkeys

Actual values	Actual value display, depending on the selected coordinate system
Progr.Ctrl	Program control
Delete MDA	Deletes the MDA program (without safety prompt)
Save MDA	Saves the MDA program.

The following working window appears when this softkey is pressed:

Save MDA program

Save MDA Program working window

Operating sequence

- Type in the program name.
- Click on the **OK** softkey to save the MDA program and return to the previously active working window.

Further buttons

	Recall or	Returns to the previously active working window
	Cancel softkey	To cancel without accepting the value.

■

Automatic Mode – Operating Area Machine

6.1 Introduction

Function

The *Automatic* mode represents the normal mode of operation for parts machining. In this mode you can run through parts programs fully automatically.

Within this mode you can have information displayed and influence the operation. These options are described below.

Prerequisites

- The relevant parts program has been saved in the control and selected.
- The measurement system of the control has been synchronized with that of the machine (reference point approach).
- All of the necessary offset values have been checked or entered, e.g. zero offsets or tool offsets.

- The **Automatic** function key has been pressed.

The second field in the first status bar of the working window then changes to *Auto* or *Mda*.

If you then press

- the operating area button and then

- the **Machine** softkey, the *Program Blocks* working window normally appears.

If you change into this area later on then the above working window is not necessarily displayed, as the last selected working window pops up instead.

6.2 Program blocks

Function

A section of the program currently being edited is displayed in the program block display. The editor is in read-only mode, which means that no changes can be made to the displayed program. This prevents accidental changes to the program.

The current block is shown inverted.

You can specify in the *Program Control (2)* working window which types of blocks are displayed (i.e. only machine function blocks or all).

Softkeys

Channel	Switches to the next channel (only displayed if two or more channels are present)
	Please read the warning below!
Actual values	*MCS/WCS actual values*
	If the machine function "Reposition" is active then the REPOS displacement is displayed, otherwise the distance-to-go is shown.
Curr.block	*Current block*
Search	*Block search*
Progr.level	*Program levels*
Progr.corr	*Program correction*
>	2nd softkey level (toggle button)
Overstore	*Correction block overstore*
Progr.ctrl	*Program control*

Warning

When the channel is switched over, one of the following may occur depending on the PLC programming:

- The position of the override rotary switch may no longer match the override on the channel (e.g. when the rotary switch only acts on channel 1) or
- The override on the channel may change when the channel is switched over.

Machine manufacturer

The manufacturer of the installation must ensure that the installation is configured in a way which prevents dangerous situations from arising.

6.2.1 Current block

Function

The *Current Block* working window shows the commands which have actually been performed or are to be performed (jump commands etc. have already been taken into account), while the *Program Blocks* working window shows a section of the programs.

```
Actual block          TEST_ULI.MPF
ᴸᶠ
LABEL:ᴸᶠ
G1 G90 X100  Z300 F2000ᴸᶠ




              Act.  Progr. Search Progr. Edit
              val.  blocks        level  progr.   >
```
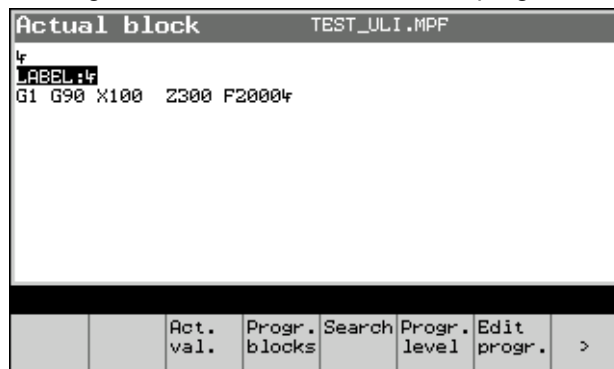
Current Block working window

Softkeys

Progr.blocks   *Program blocks*
Otherwise the same as in the section "Program blocks".

6.2.2 Block search

Function

The block search function allows the program to be advanced to the desired point in the parts program. Two different search methods are available:
1. Search with calculations:
   During the search all the same calculations are carried out as during normal program operation mode, but the axes do not move.
2. Search without calculations:
   No calculations are carried out during the search.
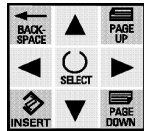
- You can directly position the cursor to specify the search target. When the search target has been reached it is adopted as the current block. After the NC startup the program is launched, and processing of the program continues from this point.

- You can abort the search by pressing the **Reset** key.
The *Block Search* function is not available in the sub-operating mode TEACH IN.

Precondition

The **Search** key has been pressed in one of the working windows *Program Blocks*, *Current Block* or *Program Level*.
The following working window is then displayed:

```
Search position      TEST_ULI.MPF
G1 G90 X100  Z300 F2000↵
X0 Z0 F4000↵
GOTOB LABEL↵
M30↵
↵
==eof==
```

| | With calc. | | | Plane – | Plane + | W/o calc. |
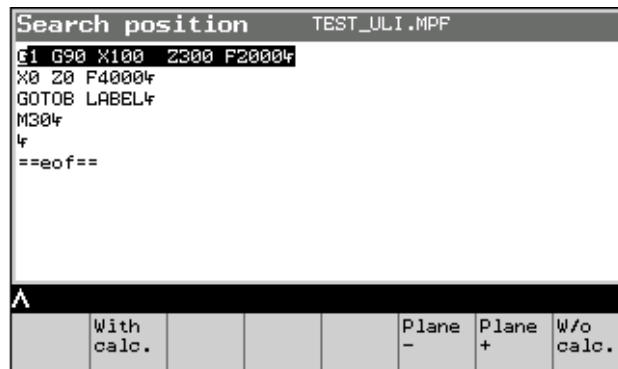
Block Search working window

Operating sequence
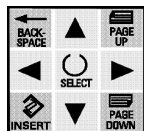
- Position the **cursor** on the desired block,

- Use the desired **softkey** to start the search.

with calc.   Starts the search from the start of the program to the cursor block with calculations,

w/o calc.   Starts the search from the start of the program to the cursor block without calculations.

6.2.3   Program level

Function

The **Progr. level** softkey has been pressed. All of the programs can be displayed in the following working window.

Operating sequence
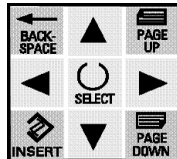
- Use the ↑ and ↓ cursor keys to display all programs.

Use of the **softkeys** is the same as for the *Program Blocks* working window.

6.2.4   Program correction

Function

This function allows you to correct faulty program lines.
When the correction mode is selected from the *Program Blocks* working window, the editor attributes are changed from "read-only" to "read/write". This enables you to make changes in the program.

Prerequisites

- The **Progr.corr.** key has been pressed in one of the working windows *Program Blocks*, *Current Block* or *Program Level*.

The correction editor pops up and the cursor is positioned at the faulty block.

Operating sequence
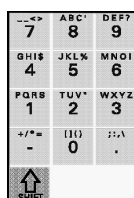
- Correct the error(s),

- Click on the **Close** softkey to accept the changes and return to the starting window.

The remaining softkeys and functions correspond to those of the "editor in normal mode".

Notes

1. Entries are not only accepted after closing, but also (depending on the settings) after each character or after leaving the block.
2. After making the selection with the cursor, you can also modify other commands in addition to the faulty block provided this does not generate a conflict with the interpreter status.

6.2.5 Overstoring

Function

The overstore buffer is opened in the editor with a write access (only effective in the operating mode **Automatic**).
It is possible to e.g. change program data when the parts program has been stopped (current speed, position etc.). When the start key is pressed the program is executed in the overstore buffer and the new data are taken over directly.
The accepted data are only valid until they are changed again in the parts program or via the "overstore" command. The overstored data are not saved in the program. The old values become active again the next time the program is started.
The overstore buffer is automatically deleted on closing.

Precondition

- System set to **Stop** or **Reset**.

- The **Overstore** softkey has been pressed in the *Program Blocks working window* or the *Current Block* working window.

In this working window you can enter the NC blocks you want to be worked through.

Operating sequence
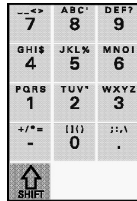
- Enter the command alphanumerically.
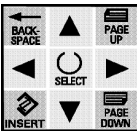
- Transfer the command into the buffer memory with the **Enter key** or
- by moving to a different line with one of the **cursor** keys.

Softkeys

Delete all   Deletes the overstore buffer.

Further buttons

Start   Executes the entered commands and returns to the calling working window.

Recall   Returns to the calling working window <u>without</u> executing the commands.

6.2.6 Program control

Function

The program operation mode can be changed using this function. The following program controls can be activated or deactivated:

SKP   Skip block
DRY   Dry run feed
ROV   Rapid traverse override
M01   Programmed stop
DRF   Selection of DRF offset
PRT   Program test
SBL1   Stop after every machine function block
SBL2   Stop after every block

Precondition

- The **Progr.Ctrl** softkey has been pressed in the *MDA Program* or *Program Blocks* or *Current Block* working window.

The following working window is then displayed:

```
Program control
■ SKP: Skip block
☐ DRY : Dry run feedrate
☒ ROV : Rap. trav. o/ride
☐ M01 : Program stop
☐ DRF: Select DRF offset
☐ PRT : Program test

Select with selection key



        Part 1 Part 2              OK
```

Program Control (1) working window

Operating sequence
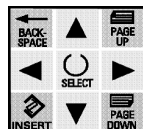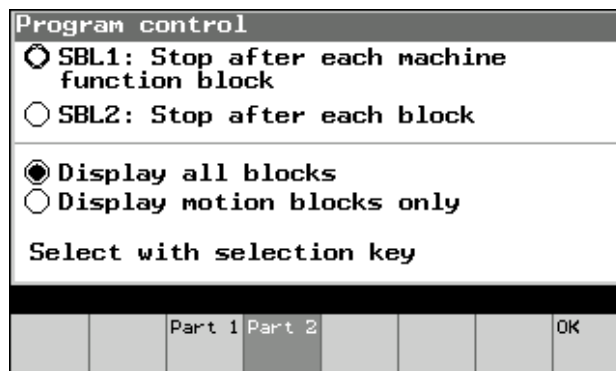
- Position the **Cursor** on the desired function (the selected symbol is inverted)
- Activate or deactivate the selected function with the **SELECT** key.
  ☒ means: Function activated
  ■   indicates the cursor position

The **Recall** function is not available here as the inputs are accepted directly via the **Select** key. This means that it is not possible to exit the working window without accepting.

Softkeys

Part1   *Program Control (1)* working window (inverted if already active, in which case ineffective)
Part2   *Program Control (2)* working window (see below)
OK   Exits the working window and accepts the inputs.

```
┌─────────────────────────────────────────┐
│ Program control                          │
│ ○ SBL1: Stop after each machine          │
│     function block                       │
│ ○ SBL2: Stop after each block            │
│                                          │
│ ◉ Display all blocks                     │
│ ○ Display motion blocks only             │
│                                          │
│  Select with selection key               │
│                                          │
├──────────┬──────┬──────┬────┬────┬───────┤
│          │Part 1│Part 2│    │    │  OK   │
└──────────┴──────┴──────┴────┴────┴───────┘
```

Program Control (2) working window

Operating sequence

Analogous to *Program Control (1)*: see above

■

Teach-In

7.1 Introduction

Function

In the **TEACH IN** operating mode you can
Select program blocks,
Insert program blocks,
Change program blocks,
Replace program blocks,
Correct programs using the editor.

Up to 12 axes can be taught-in in one block.

Prerequisites
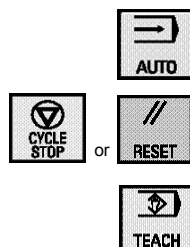
- The operating mode **Automatic** is selected.

- The system has been set to **Stop** or **Reset**
  (otherwise a message will appear saying "Program running").

- The **TEACH** function key has been pressed.

Then the third field in the first status bar of the working window
changes to "Teach-in" and the *Program Blocks* working window
appears.
If the *Program Blocks* working window has already been selected,
only the softkey bar changes.

```
Program blocks        PTP.MPF
↳
⯈
trafoof ↳
g0 y-400 z-20 a1=20 f10000 ↳
traori ↳
g0  g60  y1=-410 z1=-200 a1=0 ↳
y1=-500 z1=-350 ptp ↳




Paste Change Re-    Corr.              ->Int.
            place  editor             block.
```
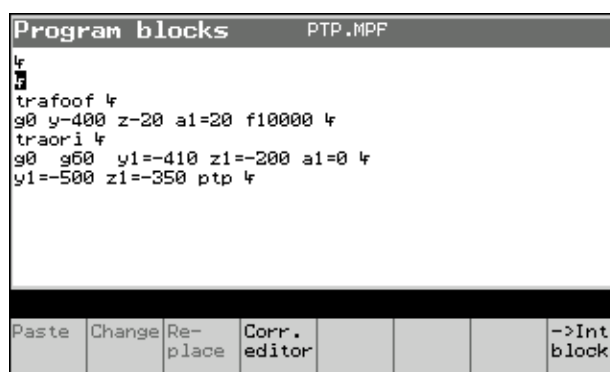
Program Blocks working window

Operating sequence

Use the appropriate softkey to activate the desired function.

Insert	Insert mode (*Insert Block* working window*)*
Change	Change mode (*Change block* working window*)*
Replace	Replace mode
Corr. Editor	Program correction with the editor (*Correction Editor* working window)
Interr.block	Starts a block search for the interruption block.

Further buttons
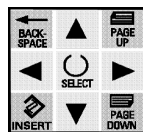
Cursor up/down → *Block Selection* working window

Note

All working windows are closed when the program is reset. The *Program Blocks* working window is then displayed again.

7.2   Block selection
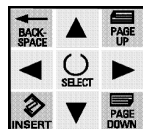
Precondition

The **Cursor up** or **down** button was pressed in the *Program Blocks* working window.
The program blocks are displayed in the working window which pops up.

Operating sequence
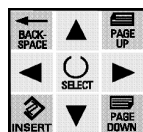
- Use the **cursor** to select the block with which the program is to be continued,

- Press the **Recall** or **Start** key to perform the block selection**.**

or

Softkeys

<-Curr.block   Repositions onto the current block

Notes

A block is <u>always</u> selected when the block selection screen is exited
(via Recall, operating area changeover etc.).
The program is then only continued at the selected point after the next
start-up. If the start-up is performed with the block selection screen
active (previously blocked), then the block selection is implicitly
carried out before the start-up is executed.
It is possible to change to other teach-in functions.
Preconditions:
Program not running.
Operating mode set to *Teach-in*.

7.3   Inserting a block

Function

The working window contains an input field for the motion type and
the actual values in the WCS. Only the axes which are assigned to
the current channel according to the machine data are displayed.
When it is selected for the first time the input field is not pre-assigned,
as it is also possible to create blocks without G code. Inputs can only
be made via softkey (see below).
Instead of "G00", "G01" or "CIP", the names in the machine data may
have been redefined by the user.
The displayed actual values cannot be overwritten. Alphanumerical
corrections of positions are possible in the *Editor*.
The teach-in block is inserted after the current block.

Precondition

The **Insert** softkey has been pressed in the *Program Blocks* working
window.

The following working window is then displayed:

```
Insert block          PTP.MPF
Motion type: G00      Rapid traverse
  Y1    -410.000   mm    U1       0.000   Deg.
  Z1    -200.000   mm    V1       0.000   Deg.
  A1       0.000   Deg.
Motion                  ○CP      ○PTP

STAT          00000000000000000000000000000000
TU            00000000000000000000000000000010

∧
G00    G01    CIP    ASPLIN        Teach  INC    STAT/
                     E             para.         TU dec
```

Teach-in – Insert Block working window

Notes

Concurrent teaching-in of Status (STAT) and Turn (TU) only takes place if "point-to-point" traversing (PTP) is selected and the associated teach-in parameters have been set. They are only visible if PTP traversing is supported (machine data set). Otherwise only the axes are displayed.
The Increment display only appears when incremental traversing is active.

The STAT and TU displays can be masked via the machine data.

References   /IAM_IM2 Start-up Guide, Section 11

Operating sequence

- Press **G00** if rapid traverse is required,

- Press **G01** if feed motion is required,
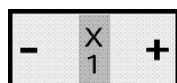
- Approach the point with the **Jog** keys,

- Accept the actual position by pressing the **Input** button, use it to create a block and then return to the starting window.

Notes

- The teach-in block comprises the selected motion type, the selected axes with position values and the selected teach-in parameters.

Example:
X = 59.012 Y = 49.246 Z = 140 B = 0 C = 0 STAT = 'B10' TU = 'B1' PTP

- The motion type setting is modal, but it can be deselected again for the corresponding motion type by pressing the softkey again, or it can be re-selected from new again by pressing another key.
- If the G codes for G00, G01, CIP, ASPLINE, PTP, CP, STATUS and TURN are re-defined then the corresponding display is also changed (also applies to the associated softkey).

Further buttons

CIP	*Insert Circular Motion* working window,
ASPLINE	*Insert A-Spline* working window additional softkey; available as an option.
TeachPara.	*Specifying Parameters* working window
INC	Incremental traversing
STAT TU dec	Decimal representation of **Status** and **Turn** Softkey toggles to **STAT TU bin**
STAT TU bin	Binary representation of **Status** and **Turn** (toggle key).

7.3.1 Inserting a block – circular motion

Function

Circular blocks can also be taught-in. To do this, the auxiliary point and the destination point need to be entered. Only the input of the auxiliary point is described here, as the input of the destination point is analogous.

The following working window appears after the **CIP** softkey is pressed in the *Teach-in – Insert Block* working window:

```
Insert block          PTP.MPF
Motion type: CIP      Circ. constr. pnt.
  Y1      -500.000    mm
  Z1      -400.000    mm

Motion              ◉CP     ○PTP
                       □ Targ.pnt. taught
STAT        0
TU          2

Λ
G00    G01    CIP    ASPLIN Target Teach   INC    STAT/
                     E      point  para.          TU bin
```

Circular Motion – Auxiliary Point working window

Other information

Teach-in is only performed for the geometry axes. The other axes are not selectable.

Operating sequence
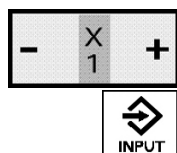
- Approach the auxiliary point with the **Jog** keys,

- Press the **Input** key.

The system then responds according to the previous history:

The complimentary point has not yet been taught-in.	The complimentary point has already been taught-in.
The actual position is accepted,	The actual position is accepted, a block is created and inserted after the cursor block,
→ complimentary working window	→ return to the calling working window

Further buttons

Destination point   Goes to new complimentary working window
                          without *accepting* the settings

Teach-Para.   *Teach-in Parameters* working window

Recall   Goes to the starting window without accepting
                          the settings

7.3.2   Inserting a block - A-Spline (option)

Function

The spline interpolation function can be used to link series of points
along smooth curves.

Precondition

The softkey **A-SPLINE** has been pressed in the working window
Teach-in – Insert Block.

Keys

same as for the working window *Insert Block*

7.3.3   Specifying parameters

Function

At the start of an editor session all of the axes are selected. A
selection can be made as required:

After pressing the **Teach Para.** softkey in the *Insert Block* or *Change
Block* working window the following working window will appear:

Specifying Parameters working window

Operating sequence

- Select the desired parameter with the corresponding softkey <u>or</u>

- move the **cursor** to the previous/next input field,

- select or deselect the axis with the **SELECT** key and

- click on the **OK** softkey to accept the settings and return to the starting window.

- Recall $\rightarrow$ Returns to the starting window without accepting the settings

7.4 Changing a block
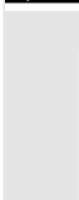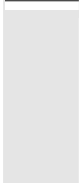
Function

To change a block which has already been programmed you proceed in a similar way to the procedure for *inserting* a block,

- except that you press the **Change** softkey.

The block to be changed is displayed.

Operating sequence

With the **Input** button you can swap the programmed values in the block for the actual values for the **selected** axes. The cursor is then positioned at the start of the changed block.

Only the actual values are overwritten. It is not possible to insert new axes or delete existing axes.

Notes

- The motion type and the motion parameters cannot be changed.
- Only the axis values already contained in the traversing block can be changed, but no new ones can be added and none can be deleted, not even if they have been selected via "Axis selection". The axis selection chooses which of the existing axes in the teaching-in block are to be newly taught-in.
- The displayed axis values are actual values, not the values contained in the block.
- If you wish to change any variable except the position in the *Editor in Correction Mode* working window, then we recommend the alphanumerical input.
- With circular blocks, every time you press the **Input** button you are changing the block, as both points are already programmed.
- If a block does not contain any allocations to "teachable" axes then no changes are made to the block, but all the other actions (block selection, return) are carried out nonetheless.
- The block can only be modified if the block layout corresponds to that of a block which has been taught-in by the HPU, otherwise the error message "Teach-in not permitted" will appear.

7.5 Replacing a block

Function

The function *Replace Block* is a mixture of *Insert Block* and *Change Block*.
In contrast to *Insert Block*, the new block is inserted in place of the current block, not after it (the current block is therefore deleted).
For a description refer to the function *Insert Block*.

7.6 Correction mode

Function

You can make alphanumerical corrections or teach-in, unless the same program is being processed on a different channel.
A warm restart is only possible when the editor has been closed with the **Close** softkey (see below).

Operating sequence

Similar to *Editor in Normal Mode*

- Edit the block alphanumerically (with the cursor key block),
- Press **Input** to confirm the entry; → new line.

Further buttons

Edit	*Text Edit* working window
Search	Searches for a sequence of characters (string)
Continue search	Searches for the last string again
Close	Closes the editor; returns to the starting window
Cursor keys	Standard editor functions (entries are accepted whenever the line is changed).

7.7 Example: Creating a new program with teaching

Programming example

Option A

1. Create a new program with the following contents:
   ```
   N10 F5000   ; Speed at G01
   N20 M0      ; Programmed stop
   N30 M30     ; End of program
   ```
2. Select the program
3. Switch on the single block: SBL1
4. Press "NC-Start" twice: Program execution is at "MO".
5. Teach the block: The block is inserted after "M0".
6. Teach the next block and the other blocks. The block is inserted after the last taught block.

Option B

1. Create a new program with the following contents:
   ```
   N10 F5000   ; Speed at G01
   N20 M30     ; End of program
   ```
2. Select the program
3. Switch on the single block: SBL2
4. Press "NC-Start" twice: Program execution is at "F5000".
5. Teach the block: The block is inserted after "F5000".
6. Teach the next block and the other blocks. The block is inserted after the last taught block.

Option C

1. Create a new program with the following contents:
   ```
   N10 F5000   ; Speed at G01
   N20 X10 Y10 ; E.g. basic position of the machine
   N30 M30     ; End of program
   ```
2. Select the program
3. Switch on the single block: SBL1
4. Check that the machine is not in the basic position. If the position has already been reached during execution of the program, then the block is treated as a "control block" by the control, and on the second NC start the end of the program would already be reached.
5. Press "NC-Start" twice: Program execution is at "X10 Y10".
6. Teach the block: The block is inserted after "X10 Y10".
7. Teach the next block and the other blocks. The block is inserted after the last taught block.

■

Data Transmission – Operating Area Services

8.1 Introduction

Function

You can read data and programs in or out via the RS-232-C data interface of the HT 6. There are:

- Commissioning data
- Display machine data
- Workpieces
- Parts programs
- Subroutines
- User cycles
- Standard cycles
- Manufacturer cycles
- Comments
- Definitions
- Feed drives
- Main spindle drives
- OEM (data and programs of the machine manufacturer)
- System
- Logbook
- Communications fault log
- Texts
- Machine configuration

To access the interface, open the RS-232-C cover on the housing. When reading in data you can initially save the files to a temporary clipboard before deciding whether they should be deleted or saved in a selected target directory.

Reading data out is performed analogously, but there is no interim storage option on the clipboard.

Transmission parameters like the baud rate, parity etc. need to be set in order for the data communications to take place.

8.2 Read in data

Function

The process of reading data into the HT 6 comprises the following steps:

1. Selecting the data (*Read in Data* working window),
2. Deciding where to transfer the data to (*Select Target Directory* working window),
3. The actual transfer itself.
4. If data have been transferred to the clipboard then a decision also needs to be taken as to whether they should be deleted or organized in a directory.

Prerequisites

- A connection to the RS-232-C interface has been established.

- The operating area button has been pressed,

- The **Services** softkey has been pressed.

The following working window is then displayed:

```
Read in data :
Name:
Data
Start-up data
Display machine data
Workpieces
Part programs
Subprograms
User cycles
Standard cycles
Comments

Data   Data   Clip-   Error  NC     RS232C          Start
in     out    board   log    card   param.
```
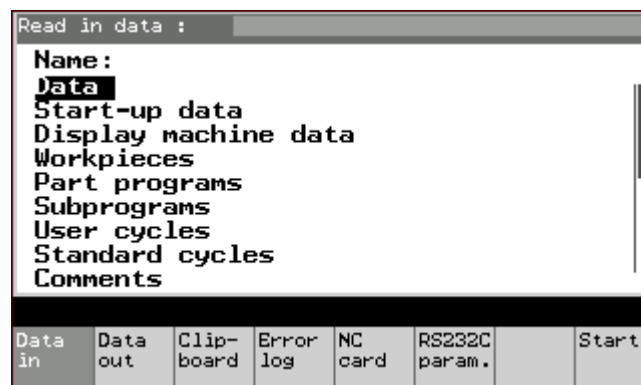
Read in data working window

Operating sequence
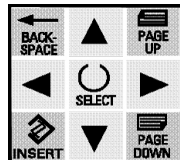
- Use the **cursor** to select a directory (in the example: "Daten" (*Data*))

- Use the **Start** softkey to call the working window in which you can select the target directory.

Further buttons

Data out	*Data Output* working window
Clipboard	*Clipboard* working window
Fault log	*Fault Log* working window
NC card	*Select NC Card Data* working window
RS-232-C param.	*RS-232-C Settings* working window

8.2.1 Selecting a target directory

Function

Files can be copied

- to the directory selected in the *Read in Data* working window, or
- to the directory shown in the header of the file, or
- to the clipboard.

Precondition

- The **Start** softkey has been pressed in the *Read in data* working window.

The following working window is then displayed:

Select Target Directory working window

Operating sequence
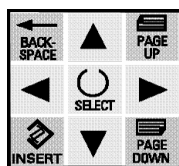
- Use the **cursor** to select the desired target directory ("path/workpiece from archive file" and/or "read in to clipboard")

- Use the **SELECT** key to activate or deactivate the directory.

Effect:

No field checked:	First field checked:	Second (or both) field(s) checked:
The files are copied to the directory selected in the *Read in Data* working window.	The files are copied to the directory shown in the header of the file.	Files are copied to the clipboard.

- Start the transfer by clicking on the **OK** softkey.

The *Transfer* working window will then appear.

Further buttons

Cancel softkey or

Recall   Returns to the starting window *Read in Data*

8.2.2 Transfer

Function

The current status of the file transfer is shown on the screen.

Precondition

- The **Start** softkey has been pressed in the *Select Target Directory* working window.

The following message appears: "Transfer active"
The following are displayed:

- – when reading in the data: only the number of transmitted bytes
- – when outputting the data: the path and name of the last transmitted file as well.

Further buttons

Softkey **Stop** or

Recall   Transfer is aborted, screen returns to the calling working window *Read-in data.*

8.2.3 Save to clipboard

Function

Data stored on the clipboard can either be deleted or organized in a directory.

Precondition

- The **Clipboard** softkey has been pressed in the *Read in Data* working window.

Another working window opens showing a selection of all files, with name, type, length and date.

Operating sequence

Select the file(s) and

press the corresponding softkey to start the desired action:

Organize	Moves the file from the clipboard to the directory selected in the *Read in File* working window.
Trans. Dir.	→ Transfer directory (2nd clipboard) Only appears when Teleservice is active.
Delete	Deletes the file from the clipboard.

A safety prompt appears before the file is deleted or organized into a directory.

8.2.4 Transfer directory (2nd clipboard)

Function

The transfer directory is active for the "remote diagnosis" function. It acts as a second clipboard.

Precondition

- The **Trans.Dir** softkey has been pressed in the *Clipboard* working window.

Another working window appears, in which the "trans-files" can be selected.

SINUMERIK 840D/840Di/810D Operator's Guide Handheld Terminal 6 (BAH) - 03.04 Edition
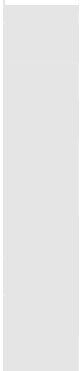
Operating sequence

Press the corresponding softkey to start the desired action:

Organize	Moves the file from the transfer directory to the directory selected in the *Read in File* working window.
Gener.archive	→ new working window in which a prompt asks for the archive name. Afterwards the current file is copied to the transfer directory.
Delete	Deletes the file from the clipboard.
Refresh	Updates the contents of the transfer directories

A safety prompt appears before the file is deleted or organized into a directory.

8.2.5 Fault log

Function

In the event of a fault during the data transfer, a log is created and shown in the *RS-232-C Fault Log* working window under "User Abort".

Further buttons

Press the **Recall** softkey to return to the calling working window.

8.2.6 Read in data from NC card

**Function**

Files can also be read in from an NC card.

Precondition

- The **NC Card** softkey has been pressed in the *Read in data* working window.

The following working window is then displayed:

```
Copy and paste da
  Name:
  Data
  Start-up data
  Display machine data

NC card      : TARGET

  Name                        Type Length  Date
  NC01                        ARC  26112 01.03.02



∧
Data   Data                      De-           Abort Start
in     out                       lete
```

Select NC Card Data working window

Operating sequence

- Use the **cursor** to select the desired source.

- Use the **Start** softkey to call the startup window for reading in from the NC card:

The working window opens, e.g. *Commissioning Data*.

- Click on the **OK** softkey to start the transfer, the commissioning data are read in.

Further buttons

Recall   "Cancel read-in": Cancels the transfer and returns to the starting window.

8.2.7   RS-232-C settings

Function

Standard values for the parameterization of the interface are read from the display machine data.
You can adapt the standard values to the current transfer in the following working windows.
The changes are applied to the machine data.

Precondition

- The **RS-232-C Param.** softkey has been pressed in the *Read in Data* working window or in the *Output Data* working window.

The following working window is then displayed:

```
RS232C settings: Parameter (1)
 Interface              COM1
 Log:                  ⊍ RTS/CTS
 Baudrate:             ⊍ 9600
 Stop bits:            ⊍ 1
 Parity:               ⊍ none
 Data bits:            ⊍ 8
 XON / XOFF:            11   13   Hex
 End of transmission:   1a
∧
          Part 1 Part 2                    Save
                                           sett.
```

RS-232-C Settings working window

Operating sequence
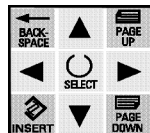
- Use the **cursor** to select the desired line.

- Change the upper six values with the **SELECT** key.
- Change the lower values alphanumerically, and if the special functions (*RS-232-C Settings* working window*, Special functions)* are not to be parameterized as well:
- Transfer the changes to the working memory using the **Save settings** softkey.

Further buttons

Part 2   → *RS-232-C Settings* working window: *Parameters (2)* Special functions

Save settings Save settings and return to calling window.

Recall   → Returns to starting window without accepting the changes.

8.2.8   RS-232-C settings, special functions

Function

Expanded parameterizing functions can be found in the *Special Functions* working window.

Precondition

- **Part 2** softkey has been pressed in the working window *RS-232-C Settings*.

The following working window is then displayed:

```
RS232C settings: Parameter (2)
 U  ■ Start with XON
 U  □ Overwrite with confirmation only
 U  ⊠ Read-in: Block end with LF only
 U  ⊠ Stop with EOT character
 U  □ Evaluate DSR signal
 U  □ Leader and trailer
 U    Punched tape
 U  □ Timeout
Λ
          Part 1 Part 2              Save
                                     sett.
```

RS-232-C Settings, Special Functions working window

Operating sequence
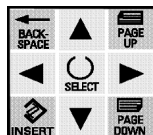
- Use the **cursor** to select the desired line.

- Change the selected value with the **SELECT** key.

- Transfer the changes to the working memory using the **Save settings** softkey.

Further buttons

Part 1   → *RS-232-C Settings* working window:
                   Parameters (1)

Save settings Save settings and return to calling working window.

Recall   → Returns to starting window without accepting the changes.

8.3 Output data

Function

The data output runs analogously to the reading in process. However, it is not meaningful to save in a temporary storage location (i.e. clipboard), so this is omitted.

Precondition

The **Data out** softkey has been pressed in the *Read in Data* working window.
The same working window opens up as for *Read in Data*.

Operating sequence   *Same as Read in Data* working window.

Further buttons

Data in   → *Read in Data* working window
NC card   → *Output Data to NC Card* working window

8.3.1 *Output Data to NC Card* working window

Function

Data can also be output to an NC card analogously to the section "Read in data from NC card".

Precondition

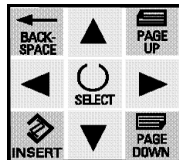- The **NC** softkey has been pressed in the *Output data* working window.

The following working window is then displayed:

```
Copy and paste da
  Name:
  Data
  Start-up data
  Display machine data


NC card     : TARGET

  Name               Type Length  Date
  NC01               ARC  26112 01.03.02




Λ
Data  Data            De-         Abort Start
in    out             lete
```

Output Data to NC Card working window

Operating sequence

- Use the **cursor** to select the desired file.

- Use the **Start** softkey to jump to the *Archive* working window.

The working window *Create Archive* appears.

Operating sequence

- Enter the desired archive name.
- Start the transfer with the **Start** softkey ($\rightarrow$ Working window analogous to the "Transfer" procedure)

Further buttons

Delete Deletes the selected file

Abort Cancels the transfer

■

Messages, Service Displays, PLC Data – Operating Area Diagnosis

9.1 Introduction

Function

Functions which provide information about the system are grouped together in the Diagnostics operating area. In some cases these functions also allow data to be modified. There are:

- Alarms,
- Messages,
- Service displays,
- PLC status, date/time,
- Remote diagnosis (optional).

Prerequisites

You have pressed the

- operating area button and then

MENU SELECT

Diag- nose

- the **Diagnostics** softkey.

When you select this operating area for the first time the *Alarms* working window appears. When you subsequently change into this operating area the last selected working window will appear.

9.2 Alarms

Function

All pending alarms are displayed in the alarm overview.
The **Alarms** softkey has been pressed.
A working window appears containing a listing of all pending alarms with alarm number, date, cancel criterion and explanation.

Operating sequence

Use the corresponding softkey to activate the desired function:

Alarms	Lists the alarms which have occurred
Messages	*Messages* working window
Serv. displays	*Service Displays* working window
PLC	*PLC* working window
Rem.diag.	*Remote diagnosis* working window (only if the "Remote diagnosis" option is enabled)

9.3 Messages

Function

Similarly to the alarms, messages can be displayed after pressing the **Messages** softkey in the *Alarms* working window.

Operating sequence

There are no operations other than reading the messages.

Further buttons

Alarms	*Alarms* working window
Messages	Updates the working window

9.4 Service displays

Function

This is a pure display function, which means that data cannot be changed. There are:

- Service axes display,
- System resources (NC utilization)
- Communications log
- Action log
- NCU and HMI version information

The following working window appears once the **Service Displ.** softkey has been pressed in the *Diagnostics Operating Area* working window.

```
Service displays
Service axes


System resources

Communications log
Action log
Version
Λ
Serv.        System      Comm. Action Ver-
axis         res         log   log    sion
```
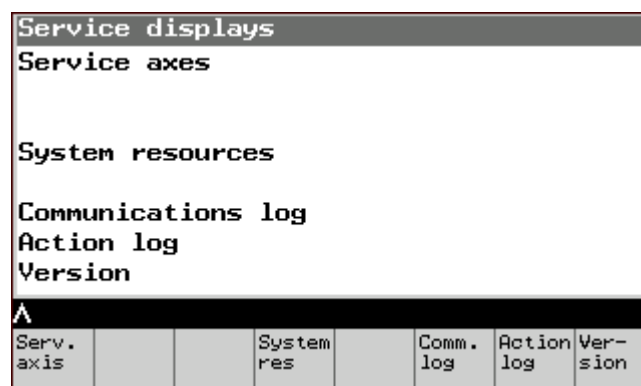
Service Displays working window

Operating sequence

- Select the desired function via the corresponding softkey.

9.4.1 Service axes

Precondition

- The **Serv. axis** softkey has been pressed in the *Service Displays* working window.

```
Service axes                          X1  1
Follow'g error               0.001 mm
Control deviation            0.001 mm
Contour dev.(axial)          0.001 mm
Kv factor(calculated)        0.000 1000/min
Act. meas. system                1
Pos.act.val.meas.sys1       74.879 mm
Pos.act.val.meas.sys2        0.000 mm
Position setpoint           74.880 mm

∧
Axis -│Axis +│   │   │   │   │   │
```

Service Axes working window

Operating sequence
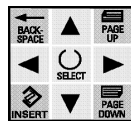
- Use the cursor keys to select the desired line.

Further buttons

Axis -/+   Displays information about the previous/next axis

Recall   → Returns to the *Service Displays* working window.

9.4.2 NC utilization

Precondition

The **System Res.** softkey has been pressed in the *Service Displays* working window.
The following working window is then displayed:

```
NC utilization                              chan1  1
Net runtime (pure calculation time)
                 Actual        Minimum        Maximum
Pos.contr.       0.480 ms      0.181 ms       1.311 ms
Interpolator     4.330 ms      0.444 ms       6.225 ms
Fwd. motion      0.204 ms      0.164 ms      52.352 ms

NC utilization (pos.contr./IPO)   53.8% 10.8% 102.%
Space used in interpolator buffer         0 %

Λ
Chan-  Chan-         Net  Gross        Stop  Start
nel -  nel +
```

Net NC Utilization working window

Function

In this window the **net** runtime (pure computation time) is displayed for the NC.

Further buttons

Channel -/+   Displays information about the previous/next channel

Stop   Number refresh is stopped, numerical values stop.

Start   Number refresh is restarted (inverted on selection).

Recall $\rightarrow$ *Service Displays* working window

9.4.3 Communications log

Function

The following working window logs the communications between the HMI and NCK/PLC. Date and time are output at the start of communications.

Softkeys

Same as *Service Displays* working window.

9.4.4 Action log

Function

You can display an existing action log.
This function is only intended for service purposes.

Softkeys

Same as *Service Displays* working window.

Further buttons

Curr. data The displayed data are updated
Data RS-232-C Transfer of the data via RS-232-C

9.4.5   NCU version data

Precondition

- The **Version** softkey has been pressed in the *Service Displays* working window.
- The **Vers.NCU** softkey has been pressed.
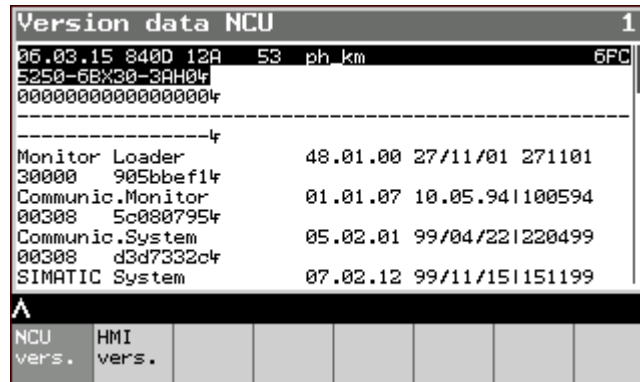
The following working window is then displayed:

```
Version data NCU                                    1
06.03.15 840D 12A   53  ph_km                     6FC
5250-6BX30-3AH04↵
00000000000000004↵
---------------------------------------------------
----------------↵
Monitor Loader             48.01.00 27/11/01 271101
30000    905bbef1↵
Communic.Monitor           01.01.07 10.05.94|100594
00308    5c080795↵
Communic.System            05.02.01 99/04/22|220499
00308    d3d7332c↵
SIMATIC System             07.02.12 99/11/15|151199
Λ
NCU   HMI
vers. vers.
```

NCU version data working window

Operating sequence
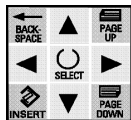
- Use the cursor keys to select the desired line.

Further buttons

Vers. HMI   *HMI version data* working window

Recall   Returns to the calling working window

9.4.6 HMI version data

Precondition

The **Version** softkey has been pressed in the *Service Displays* working window.
The **HMI Vers.** softkey in the *HMI Version Data* working window has been pressed.
The HMI data are displayed (analogously to the NCU version data).

Operating sequence
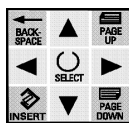
Use the cursor keys to select the desired line.

Further buttons

Vers. NCU → *NCU version data* working window

Recall Returns to the calling working window

9.5 PLC

Function

User-specific expansions can be entered in the *Diagnostics* operating area for clear illustration of PLC states.

- Reading and modifying PLC data
- Entering the date and time

Precondition

The **PLC** softkey has been pressed in the *Alarms* working window.

The working window which pops up offers the following selection:

- PLC status
- Set date and time of the PLC

Operating sequence

Select the desired function via the corresponding softkey.

9.5.1 PLC status

Function

Here, any PLC data (markers, inputs, outputs, data blocks) can be read and modified.

Precondition

- The **PLC Status** softkey in the *PLC* working window has been pressed.

The following working window is then displayed:

PLC status		Active
Operand	**Format**	**Value**
DB21.DBB5	B	0001 1010
DB32.DBB19	D	14
DB33.DBB2	H	12

| Oper. – | Oper. + | Input Help | De-lete | | Change | Abort | Accept |

PLC Status working window

Operating sequence

- with softkey 1 or 2:

Oper. - Decrement operand – address. Increase by 1.

Oper. + Increment operand – address. Reduce by 1.
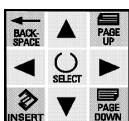
- Use the cursor keys to change between operand and format.
- Write to the input field either directly or with the input aid.

You need to activate the **Change** softkey to change the value of a piece of PLC data.

The marking then jumps to the "Value" area.

- After selecting and modifying the values they are adopted by the system via the **Accept** softkey.

Further buttons

Input aid	Assistance for entering values
Delete	Deletes all display lines after a safety prompt **Yes/No**
Abort	Cancels changing the PLC values and changes from "Values" to "Operand" mode.

Input aid

Instead of entering the operands and formats directly in the *PLC Status* working window, you can also generate them by pressing the **Input aid** softkey and then the softkeys which appear subsequently.

Depending on the previous history, the following softkey bars appear:

"Operand" is selected:

DB	M Marker	I Input	Q Output	T Time	C Countr		

Input aid softkeys

After softkey selection **DB**:
Select .DBX, .DBB, .DBW and .DBD.

After softkey selection **M**, **I** or **Q**:
Select B byte, W word, D double-word.

If the cursor is in the "Format" input field then the following values appear in the softkey bar:
B binary, D decimal., H hexadecimal, F float

Further buttons

∧	**Recall**	Exits the input aid and accepts the inputs.

9.5.2 Date/time

Precondition

- The **Date/time** softkey in the *PLC* working window has been pressed.

The following working window is then displayed:

```
Set date and time of PLC
Current time:
  HMI  Thursday, 06.June 2002            09:37:53
  PLC  Thursday, 06.June 2002            09:37 10
New PLC time:
  Day    Month    Year    Hour   Minutes  Second
   6       6      2002      9       37       10

        Synchronization time (0-99min) 10

                        Sync.  Autom. Autom.        OK
                        manual ON     OFF
```

Date and time working window (inverted softkey: currently active function)

Operating sequence
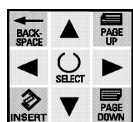
- Use the cursor keys to move to the desired input field.

- Input the alphanumerical changes.
- Click on the **OK** softkey to accept the changes.

Further buttons

Manual sync.	Synchronizes the PLC and HMI times.
autom. on	The time after which synchronization is automatically performed is entered in the "Synchronization time" field.
autom. off	No automatic synchronization ("Synchronization time" line disappears).
Recall	Goes to the *PLC* working window without accepting the changes.

9.6 Remote diagnosis

Function

If no modem was active during startup then this function can be used to manually start the remote diagnosis. The remote diagnosis can be parameterized in the PowerOn startup menu (key 6 during startup) (e.g. automatic starting of the remote diagnosis during startup). The baud rate etc. are set using a third-party product, e.g. "Reach Out".

Precondition

Once the **Remote diagn.** softkey has been pressed in the *Messages* working window, the *Remote Diagnosis* working window appears.

Operating sequence

- **OK**   The remote diagnosis is started and a restart is performed.

If the remote diagnosis has already been started the message "Remote diagnosis has already been started" is output and the working window is closed.

Further buttons

Recall Goes to the *Messages* working window without starting the remote diagnosis.

References Information about the installation and operation of the remote diagnosis can be found in the following documentation:
/FBFE/ Description of Functions, Remote Diagnosis, FE1 (ReachOut)

■

Machine Data, NC Start-Up, System Parameters – Operating Area Start-Up

10.1   Introduction

Danger

Changes in the startup operating area have a significant influence on the machine. Incorrect parameterization can endanger lives and cause the destruction of the machine.

Access rights to certain menus in the startup operating area may be blocked by means of a keyswitch or password.

The functions are described which can be executed by the machine operator with his access rights.

For more detailed information on the subject of startup for
- System personnel
- Machine manufacturers
- Service personnel
- Machine users (machine setters)

please refer to the following documentation:

References   /IAM/IM2, Start-up Guide, HMI Embedded, Section 11
/IAD/Start-up Guide, SINUMERIK 840D
/IAC/Start-up Guide, SINUMERIK 810D

Function

The following functions are available during startup:
- Changing machine data and setting them effective
- Startup of NC or HT 6
- PLC status and time
- Setting the system parameters

Prerequisites

- The operating area button has been pressed,

- The **IBN** softkey has been pressed.
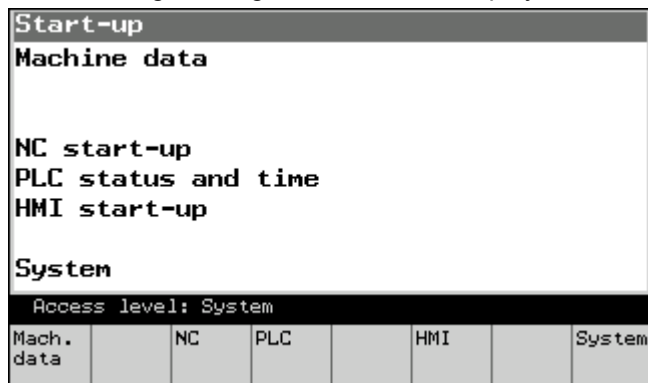
The following working window is then displayed:

```
Start-up
Machine data


NC start-up
PLC status and time
HMI start-up

System

Access level: System
Mach.         NC    PLC        HMI        System
data
```

Startup Overview working window

Operating sequence

Call up the desired function with the corresponding softkey.

Mach. Data   *Machine Data* working window
NC   *NC Startup* working window
PLC   *PLC Status and Time* working window
HMI   *HMI startup* working window
System   *System* working window

10.2   Machine data

Function

Machine data can be displayed and modified. There are:

- General machine data
- Channel-specific machine data
- Axis-specific machine data
- Drive configuration
- Drive-specific machine data
- Display-specific machine data

Precondition

- The **Mach. data** softkey has been pressed in the *Startup Overview* working window.

The following working window is then displayed:

```
Machine data
General machine data
Channel-specific machine data
Axis-specific machine data
Drive configuration
Feed-specific machine data
Main-spindle-specific machine data
Display-specific machine data



Λ Access level: System
Gen-  Chann-Axis-  Drives              Dis-
eral  spec. spec.                      play
```

Machine Data Overview working window

Operating sequence

Call up the desired function with the corresponding softkey.

General MD	*General machine data*
Channel MD	*Channel-specific machine data*
Axis MD	*Axis-specific machine data*
Drive Conf.	*Drive configuration*
Drive MD	*Drive-specific machine data*
Display	*Display-specific machine data*

10.2.1 General machine data

Precondition

The **General MD** softkey in the *Machine Data Overview* working window has been pressed.
The following machine data information is displayed from left to right in the window which pops up:

- Machine data number
- Machine data value
- Unit (if present)
- Effectiveness:
    po = effective at power on,
    so = effective immediately,
    cf = via "set effective" softkey activation
    re = set effective after program reset

The selected machine data name is displayed in the dialog line.

Operating sequence
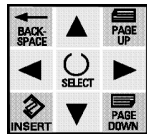
Use the cursor or the search function to select the desired machine data.

- Make changes with the keyboard and
- Set the machine data effective with the **set effective** softkey (with the attribute **cf** = confirm).
- The message "MD set effective" will then appear in the dialog line.

Further buttons

Search   Search for a character string in a number or name → *Search Machine Data* working window

```
General machine data
10930[1]          0.00000000 mm|degree    re
10930[2]          0.00000000 mm|degree    re
10930[3]          0.00000000 mm|degree    re
10930[4]          0.00000000 mm|degree    re
10930[5]          0.00000000 mm|degree    re
Search for machine data
Name or number of machine data:

Λ
                                    Abort OK
```

Search Machine Data working window

Continue search   Search for another occurrence of the string

Operating sequence
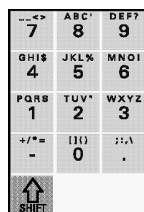
- Enter the string you wish to search for in the **Search** line.

- Click on the **OK** softkey to confirm.

The editor searches for the entered string and jumps to the point where it finds it.

Further buttons

Abort   (Softkey) or

Recall   Returns to the *Machine Data* working window

10.2.2 Channel-specific machine data

Precondition

- The **Channel MD** softkey in the *Machine Data Overview* working window has been pressed.

The channel-specific machine data information is displayed from left to right in the window which pops up.

See section "General machine data"

Operating sequence

As described in the section "General machine data".

Further buttons

Channel -/+   Displays the previous/next channel

The remaining buttons are the same as the ones described in the section "General machine data".

10.2.3 Axis-specific machine data

Precondition

- The **Axis MD** softkey in the *Machine Data Overview* working window has been pressed.

The axis-specific machine data information is displayed from left to right in the window which pops up.

See section "General machine data"

Operating sequence

As described in the section "General machine data".

Further buttons

Axis -/+   Displays the previous/next axis

The remaining buttons are the same as the ones described in the section "General machine data".

10.2.4 Drive configuration

Precondition

- The **Drive Conf.** softkey in the *Machine Data Overview* working window has been pressed.

The following working window is then displayed:

```
Drive configuration
StP1 DrvNo Activ   drive    Module    PowerSec
  1     3   N☐    FDD☐    1 axis☐    0000H
  2     1   N☐    FDD☐    1 axis☐    0000H
  3     2   N☐    FDD☐    1 axis☐    0000H
  4     0    ☐       ☐         ☐       #
  5     0    ☐       ☐         ☐       #
  6     0    ☐       ☐         ☐       #
  7     0    ☐       ☐         ☐       #

∧ Access level: System
Gen-   Chann- Axis-  Drives              Dis-
eral   spec.  spec.                      play
```

Drive Configuration working window

Softkeys

As described in the section "General machine data".

10.2.5 Drive-specific machine data

Function

This function enables

- Changes to drive-specific machine data
- Calculation of controller data
- Saving/deleting of boot files

Precondition

The feed-specific machine data information is displayed from left to right in the window which pops up.

See section "General machine data"

The selected machine data name is displayed in the dialog line.

Operating sequence

Select the machine data and activate the desired function with the corresponding softkey:

Motor controller	New softkey bar is displayed (See subsection: "Further buttons")
Boot file	New softkey bar is displayed (See subsection: "Further buttons")

Further buttons

Drive -/+   Displays the previous/next drive

Motor controller The following softkeys are available:

Calculate	Calculate parameter(s)
Calc. contr.	Convert control paramaters (with safety prompt: **No, Yes**)

Boot file The following softkeys are available:

Del. boot file	Deletes the boot file (with safety prompt: **Cancel, Delete All, Yes**)
Save boot file	Saves the boot file (with safety prompt: **Cancel, Save All, Yes**)

Operating sequence

- Select or deselect via the SELECT key
- Press the **OK** softkey

The following safety prompt is then displayed:

> **Attention!**
> Calculating the controller data will stop the drives.
> Any setting adjustments will be reset.

Press the **OK** softkey again to start the calculation.

10.2.6 Display-specific machine data
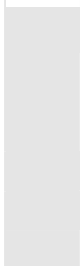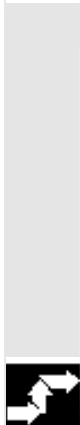
Function

Allow display machine data to be changed. If they are not saved in the NC, the data are only valid until the next boot-up of the HT 6.

Precondition

- The **Display** softkey in the *Machine Data Overview* working window has been pressed.

The display-specific machine data information is displayed from left to right in the window which pops up.

See section "General machine data"

The selected machine data name is displayed in the dialog line.

Operating sequence
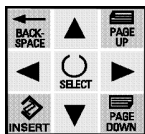
- Use the cursor or the search function (see below) to select the desired machine data.

- Make changes with the keyboard and

- Save them in the NC via the **Save** softkey.

The message "MD set effective" will then appear in the dialog line.

10.3 NC startup

Function

The NC startup offers the following function:

- NCK address

Precondition

The **NC** softkey in the *Startup Overview* working window has been pressed.

The following working window is then displayed:

```
NC start-up
Start-up switch
NCK address




A Access level: System
Start- NCK
up NC addr.
```

NC Startup Overview working window

Operating sequence

Call up the desired function with the corresponding softkey.
NCK addr. *NCK Address* working window

10.3.1 NCK address

Function

With this function you can assign a new address to the NCK.

Precondition

- The **NCK addr.**softkey has been pressed in the *NC Startup Overview* working window.

The following working window is then displayed:

```
NC start-up
Addresses of NCK
Current NCK address:              13
New address of NCK:            ████
Addresses of active nodes:
6
13


∧ Access level: System
                              Save  Abort  OK

```

NCK Address working window

Operating sequence

- Write a new NCK address into the input field,
- Click on **OK** to save it just for the current session, or
- Click on **Save** to save it permanently.

There is a safety prompt before saving:

Attention!
Do you want to change the NCK address?
The changes will not become effective until an NCK power on reset
has been performed.

Close this prompt by clicking either **Cancel** or **OK**.

Further buttons

Cancel,   Returns to the calling working window without
Recall   changing the NCK address.

10.4 PLC

These functions are described in Section 9, Subsection "PLC".

10.5 HMI startup

Function

The following individual settings are available on the HMI:

- Language selection
- Action log
- DOS editor

Precondition

The **HMI** softkey in the *Startup* working window has been pressed.

The following working window is then displayed:

```
HMI start-up

Language selection

Action log

DOS Editor

Λ
       Lang.      Action              DOS
       selec.     log                 Editor
```

NC Startup Overview working window

Operating sequence

You can call the following functions with the corresponding softkey:

Lang. Selec. → *Language Selection* working window (SK only appears if more than two languages are loaded)

Action log → *Action Log* working window

DOS editor → *DOS Editor* working window

10.5.1 Language selection

Precondition

The **Lang.select.** softkey is only available when more than two languages are loaded.

Function

With the aid of this softkey you can access a list of the foreign languages available in the system. You can choose two languages from this list.

Operating sequence

After pressing the **Lang.selec.** softkey you are given a list of the available languages. Choose your required language for languages 1 and 2.

- Click on **OK** to save the selection.

The **Change Language** softkey is used to switch between the two languages. This SK appears after you press the **System** softkey.

Further buttons

Cancel   Returns to the calling working window without accepting the selected languages.

10.5.2 Action log

Precondition

With the appropriate user access level you can access the **Action Log** softkey.

Function

The softkey offers a window for parameterizing and activating the action log. The following settings can be made and saved by clicking on **OK**:

- Log on
- Alarms
- Keys
- Channel status/override
- Windows IDs

10.5.3 DOS editor

Precondition

If the user access level is set accordingly you can access the **EditorDOS** softkey.

Function

The selected DOS shell file opens after you click on the **EditorDOS** softkey, although access is restricted to read-only.

Operating sequence

- The **Open** command opens an input window where you can enter the file name.
- Click on **OK** to open the file.
- Click on **Close** to close the file.

Further buttons

Cancel   Returns to the previous window.

10.6   Setting the system parameters

Function

System-specific settings can be displayed and, in some cases, changed here.

- LCD contrast settings
- Backlighting
- Language change
- Password processing

Precondition

- The **System** softkey in the *Startup Overview* working window has been pressed.

The following working window is then displayed:

```
System
LCD contrast setting
Background lighting
Language changeover
NCK Reset

Set password
Delete password
Change password

∧ Access level: System
Con-    Brigh- Change NCK              Set     Delete Change
trast   tness  lang.  Reset           passwd  passwd passwd
```

System working window

Operating sequence

Call up the desired function with the corresponding softkey.

Contrast	Changes the contrast settings for the screen of the HT 6
Brightness	Changes the backlighting of the HT6
Change lang.	Toggles between the two online languages
Set password	Sets a password
Del. password	Deletes a password
Change password	Changes a password

10.6.1 LCD contrast
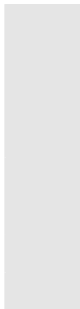
The **Contrast** softkey in the *System* working window has been pressed.
A new softkey bar appears.

Softkeys

brighter   Reduces the contrast
darker   Increases the contrast

10.6.2 Backlighting
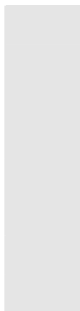
The **Brightness** softkey in the *System* working window has been pressed.
A new softkey bar appears.

Softkeys

brighter   Backlighting is made brighter
darker   Backlighting is made darker

10.6.3 Changing the language
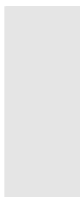
Two of the available languages (German, French, English, Spanish, Italian, Dutch and Hungarian) are loaded into the HT 6; e.g. German and English.

- Pressing the **Change lang.** softkey immediately switches to the second selected language.

10.6.4 Processing the password

Function

With this function you can set a new password or delete or change an existing password.

Precondition

The softkey **Set passw.** (for example) has been pressed in the *System* working window.
The following working window is then displayed:

```
System
LCD contrast setting
Background lighting
Language changeover
NCK Reset
Set password
Please enter password:      ████████
Current access level:
  System
Λ
                                          Abort  OK
```

Set Password working window

Operating sequence

Write the password in the input field,
click on the **OK** softkey to accept it and return to the *System* working window.
A password with an equal or lower user access level can only be set if none exists, i.e. if it has been deleted or not yet set.
The Operating Area softkey bar is displayed once the user access level has been successfully changed. The new access rights can then be checked when a new operating area is selected.
The functions *Delete password* and *Change password* should be handled analogously.

■

SINUMERIK 840D/840Di/810D Operator's Guide Handheld Terminal 6 (BAH) - 03.04 Edition   10-131

Appendix

A   Abbreviations

A	Output
ARM	Asynchronous Rotary Motor
AS	Automation System
ASCII	American coding standard for the exchange of information
ASIC	Application Specific Integrated Circuit:
ASUB	Asynchronous subroutine
AuxF	Auxiliary function
AV	Job planning
BA	Operating mode
BB	Ready
BCD	Binary Coded Decimals: Decimal numbers encoded In binary code
BCS	Basic coordinate system
BIN	**Bin**ary Files
BIOS	Basic Input Output System
BOT	Boot files: boot files for SIMODRIVE 611 D
C Bus	Communication bus
CAD	Computer-Aided Design
CAM	Computer-Aided Manufacturing
CLP	**Clip**board
CNC	Computerized Numerical Control
COM	Communication
COR	Coordinate rotation

CP	Communication Processor
CPU	Central Processing Unit:
CR	Carriage Return
CRC	Cutter radius compensation
CRT	Cathode Ray Tube
CSB	Central Service Board: PLC module
CST	**C**ustomer **St**andard Cycle
CTS	Clear To Send: Signal from serial data interfaces
CUS	User cycles (**C**ustomer **Us**er)
CUTOM	Cutter radius compensation: Tool radius compensation
DAC	Digital-to-Analog Converter
DB	Data block in the PLC
DBB	Data block byte in the PLC
DBW	Data block word in the PLC
DBX	Data block bit in the PLC
DC	Direct Control: Movement of the rotary axis across the shortest path to the absolute position within one revolution
DCD	Carrier Detect
DDE	Dynamic Data Exchange
DIN	Deutsche Industrie Norm (German Industry Standard)
DIO	Data Input/Output: data transfer display
DIR	Directory
DLL	Dynamic Link Library
DOE	Data transmission equipment

DOS	Disk Operating System
DPM	Dual Port Memory
DPR	Dual-Port-RAM
DRAM	Dynamic Random Access Memory
DRF	Differential resolver function (DRF)
DRY	Dry Run: Dry run feedrate
DSB	Decoding single block
DTE	Data Terminal Equipment
DW	Data word
EIA code	Special punched tape code, number of holes per character always odd
ENC	Encoder: Actual value encoder
EPROM	Erasable Programmable Read Only Memory
ERROR	Error from printer
FB	Function block
FBS	Slimline screen
FC	Function Call: Function block in the PLC
FD	Feed Drive
FDB	Product database
FDD	Floppy Disk Drive
FEPROM	Flash-EPROM: Read and write memory
FIFO	First In First Out: Memory that works without address specification and whose data are read in the same order in which they were stored.
FIPO	Fine interpolator

FM	Function Module
FM-NC	Function module – numerical control
FPU	Floating Point Unit
FRA	Frame block
FRAME	Data record (frame)
FST	Feed stop
FUP	Function plan (PLC programming method)
GP	Basic Program
GUD	Global user data
HEX	Abbreviation for hexadecimal number
HD	Hard disk
HHU	Handheld unit
HMI	Human Machine Interface
HMS	High-resolution Measuring System
HSK	Horizontal softkey
HPU	Handheld programming unit
HT	Handheld Terminal: Programming device (see PD)
HW	Hardware
I	Input
IBN	Startup
IF	Drive module pulse enable
IK (GD)	Implicit communication (global data)

IKA	Interpolatory compensation
IM	Interface Module Interconnection module
IMR	Interface Module Receive: Interconnection module for receiving data
IMS	Interface Module Send: Interconnection module for sending data
INC	Increment
INI	Initializing data
IPO	Interpolator
I/O	Input/Output
I/RF	Infeed/regenerative feedback unit (power supply) of the SIMODRIVE 611(D)
IS	Interface signal
ISA	International Standard Architecture
ISO	International Standard Organization
ISO code	Special punched tape code, number of holes per character always even
JOG	Jogging: Setup mode
K1 .. K4	Channel 1 to channel 4
K$_v$	Servo gain factor
K$_ü$	Speed ratio
LAD	Ladder diagram (PLC programming method)
LCD	Liquid-Crystal Display
LEC	Leadscrew Error Compensation
LED	Light Emitting Diode:
LF	Line Feed
LMS	Position Measuring System

LR	Position controller
LUD	Local User Data
MB	Megabyte
MD	Machine data
MDA	Manual Data Automatic: manual input
MC	Measuring circuit
MCP	Machine control panel
MCS	Machine coordinate system
MLFB	Machine-readable product designation
MMC	Man-Machine Communication: user interface on numerical control systems for operator control, programming and simulation
Mode group	Mode group
MPF	Main Program File: NC parts program (main program)
MPI	Multi Port Interface
MS-	Microsoft (software manufacturer)
MSD	Main Spindle Drive
NC	Numerical Control
NCK	Numerical Control Kernel: NC kernel with block preparation, traversing range, etc.
NCU	Numerical Control Unit: Hardware unit of the NCK
NRK	Name for the operating system of the NCK
NURBS	Non-Uniform Rational B-Spline

OB	Organization block in the PLC
OEM	Original Equipment Manufacturer
OP	Operator panel
OPI	Operation Panel Interface: Interface for connection to the operator panel
OPT	Options
OSI	Open Systems Interconnection: Standard for computer communications
P bus	Peripheral bus
PC	Personal Computer
PCIN	Name of the SW for data exchange with the control
PCMCIA	Personal Computer Memory Card International Association: Standard for plug-in memory cards
PLC	Programmable Logic Control: Interface controller
PLC	Programmable Logic Controller
POS	Positioning
Programming device	Programming device
RAM	Random Access Memory: program memory that can be read and written to
REF	Reference point approach function
REPOS	Reposition function
RISC	Reduced Instruction Set Computer: type of processor with small instruction set and ability to process instructions at high speed
ROV	Rapid override: Input correction
RPA	R-Parameter Active: Memory area in NCK for R-NCK for R parameter numbers

RPY	Roll Pitch Yaw: Rotation type of a coordinate system
RS-232-C	Serial interface (definition of the exchange lines between DTE and DCE)
RTS	Request To Send: Control signal sent by serial data interfaces to request that the send part is switched on
SBL	Single block
SD	Setting data
SDB	System data block
SEA	Setting Data Active: Identifier (file type) for setting data
SFB	System Function Block
SFC	System Function Call
SK	Softkey
SKP	SKiP: Skip block
SM	Stepper motor
Software limit switch	Software limit switches
SPF	Sub Program File: Subprogram
SRAM	Static RAM (non-volatile)
SRM	Synchronous Rotary Motor
SSI	Serial Synchronous Interface: Synchronous serial interface
STL	Statement list
SW	Software
SYF	System Files

T	Tool
TC	Tool change
TEA	Testing Data Active: Identifier for machine data
TLC	Tool length compensation
TNRC	Tool Nose Radius Compensation
TO	Tool Offset
TOA	Tool Offset Active: Identifier (file type) for tool offsets
TRANSMIT	TRANSform Milling Into Turning: Coordinate conversion on turning machine for milling operations
TRC	Tool Radius Compensation
UFR	User Frame: Zero offset
UI	User interface
UP	Subroutine
VSK	Vertical softkey
WCS	Workpiece coordinate system
WOP	Workshop-oriented programming
WPD	Work Piece Directory
ZO	Zero offset
ZOA	Zero Offset Active: Identifier (file type) for zero offset data
µC	Micro Controller

B   Terms

Key terms are given in alphabetical order. Terms which appear in the explanatory part and for which there is a separate entry are referred to with the "->" symbol.

A

Absolute dimension

A destination for an axis movement is defined by a dimension that refers to the origin of the currently active coordinate system. See also -> incremental dimension.

Acceleration with jerk limitation

In order to optimize the acceleration response of the machine whilst simultaneously protecting the mechanical components, it is possible to switch over in the machining program between abrupt acceleration and continuous (jerk-free) acceleration.

Access authorizations

The CNC program blocks and data are protected via a 7-stage access authorization procedure.
- three password levels for system manufacturers, machine manufacturers and users, plus
- four keyswitch positions which can be evaluated via the PLC.

Address

An address is the identifier for a certain operand or operand range, e.g. input, output etc.

Alarms

All alarms and -> messages are output on the operator panel in plain text with the date and time and a symbol indicating the cancel criterion. The display is divided into alarms and messages.
1. Alarms and messages in the parts program
   Alarms and messages can be displayed as plain text directly from the parts program.
2. Alarms and messages from the PLC
   Alarms and messages from the machines can be displayed as plain text from the PLC program. No additional function block packages are required to do this.

Analog input/output module

Analog input/output modules are signal formers for analog process signals.
Analog input modules convert analog measured values into digital values which can be processed in the CPU.
Analog output modules convert digital values into analog output signals.

Approach fixed machine point	Approach motion towards one of the predefined -> fixed machine points.
Archiving	Reading out data and/or directories to an **external** memory device.
A-Spline	The Akima-Spline runs under a continuous tangent through the programmed interpolation points (3rd order polynomial).
Asynchronous subroutine	A parts program which can be started asynchronously to (independently of) the current program status by an interrupt signal (e.g. "rapid NC input" signal).
Automatic	Operating mode of the control (block sequence operation according to DIN): Operating Mode in NC systems in which a -> parts program is selected and continuously executed.
Auxiliary functions	Auxiliary functions can be used to transfer -> parameters to the -> PLC in -> parts programs, where they trigger reactions which are defined by the machine manufacturer.
Axes	In accordance with their functional scope, the CNC axes are subdivided into: • Axes: interpolating path axes • Auxiliary axes: non-interpolating feed and positioning axes with an axis-specific feed rate. Auxiliary axes are not involved in the actual machining, and include for example tool feeders and tool magazines.
Axis address	See -> axis identifier
Axis identifier	Axes are labeled in accordance with DIN 66217 (for a clockwise orthogonal -> coordinate system) with the letters X,Y, Z. -> Rotary axes which rotate around are labeled with the letters A, B, C. Additional axes parallel to the above can be identified with further address letters.
Axis name	See -> axis identifier
B	
Backlash compensation	Compensation for mechanical machine backlash, e.g. backlash on reversal for feed screws. Backlash compensation can be entered separately for each axis.
Backup	Saving the memory contents to an external memory device.

Backup battery	The backup battery ensures that the -> user program is reliably backed up in the -> CPU against mains failure and that fixed data areas and markers, times and counters are kept in non-volatile memory.
Backup memory	The backup memory enables buffering of memory areas of the -> CPU without a buffer battery. Buffering can be performed for a configurable number of times, counters, markers and databytes.
Basic axis	Axis whose setpoint or actual value position forms the basis of the calculation of a compensation value.
Basic coordinate system	Cartesian coordinate system which is mapped by transformation onto the machine coordinate system. In the -> parts program, the programmer uses the axis names of the basic coordinate system. The basic coordinate system exists in parallel to the -> machine coordinate system when no -> transformation is active. The difference between the systems relates to the axis identifiers.
Baud rate	Rate of data transfer (Bit/s).
Blank	Workpiece as it is before a part is machined.
Block	"Block" is the term given to any files required for creating and processing programs.
Block	A section of a -> parts program terminated with a line feed. A distinction is made between -> main blocks and -> subblocks.
Block search	For testing part programs or following interruption of machining, it is possible to select any point in the part program using the block search function in order to start or resume at this point.
Booting	Loading the system program after power on.
B-spline	With the B-Spline, the programmed positions are not interpolation points, as they are just "control points" instead. The generated curve only runs near to the control points, not directly through them (optional 1st, 2nd or 3rd order polynomials).
Bus connector	A bus connector is an S7-300 accessory part which is supplied together with the -> I/O modules. The bus connector expands the -> S7-300 bus from the -> CPU or an I/O module to the neighboring I/O module.

C

C axis
Axis around which the tool spindle describes a controlled rotational and positioning movement.

C spline
The C-spline is the most well-known and widely used spline. The transitions at the interpolation points are continuous, both tangentially and in terms of curvature. 3rd order polynomials are used.

Circular interpolation
The -> tool is required to travel in a circle between defined points on the contour at a specified feedrate while machining the workpiece.

Channel
A channel is characterized by its ability to execute a -> parts program independently of other channels. A channel exclusively controls the axes and spindles assigned to it. Parts programs run on various channels can be coordinated by -> synchronization.

Channel structure
The channel structure enables the -> programs of the individual channels to be executed simultaneously and asynchronously.

CNC
-> NC

CNC programming language
The CNC programming language is based on DIN 66025 with high-level expansions. The -> high-level CNC language and programming allow, among other things, macros to be defined (groupings of individual instructions).

COM
Component of the NC control for the implementation and coordination of communication.

Compensation axis
Axis with a setpoint or actual value modified by the compensation value

Compensation table
Table containing interpolation points. It provides the compensation values of the compensation axis for selected positions on the basic axis.

Compensation value
Difference between the axis position measured by the position sensor and the desired, programmed axis position.

Continuous-path mode
The purpose of continuous-path mode is to prevent excessive deceleration of the -> path axes at the part program block boundaries (in terms of the control, machine and other properties of the operation and the user) and to effect the transition to the next block at as uniform a path speed as possible.

Contour
Outline of the -> workpiece

Contour monitoring	The following error is monitored within a defined tolerance band to ensure contour precision. An impermissibly high following error might be caused by a drive overload, for example. In this case an alarm is triggered and the axes are stopped.
Coordinate system	See -> machine coordinate system, -> workpiece coordinate system
CPU	Central Processor Unit, -> Programmable Logic Controller
Cutter radius compensation	Contour programming assumes that the tool is pointed. As this is not actually the case in practice, the curvature radius of the tool used must be communicated to the control, which then takes it into account. The curvature center is maintained equidistantly around the contour offset by the radius of curvature.
Cycle	Protected subroutine for executing a repetitive machining process on the -> workpiece.
Cycle support	In the "Program" operating area, the available cycles are listed under the menu "Cycle Support". After selecting the desired machining cycle the required parameters for the value assignment are displayed in clear text.

D

D number	Number for the tool offset memory
Data block	1. Data unit of the -> PLC, which the -> HIGHSTEP programs can access. 2. Data unit of the -> NC: Data blocks contain data definitions for global user data. These data can be initialized directly when they are defined.
Data transmission program PCIN	PCIN is an auxiliary program which is used to send and receive CNC user data via the serial interface, such as e.g. parts programs, tool offsets etc. The PCIN program can be executed under MS-DOS on standard industrial PCs.
Data word	A data unit, two bytes in size, within a -> data block.
Diagnostics	1. Control operating area 2. The control has both a self-diagnostics program and testing aids for service. Status, alarm and service indicators.
Digital input/output module	Digital modules are signal formers for binary process signals.

Dimensions in metric units and inches	Position and gradient values can be entered in the machining program in inches. The control can be set to a basic system regardless of the programmed measuring system (G70/G71).
Drive	• SINUMERIK FM-NC offers an analog +10V interface to the SIMODRIVE 611A converter system. • The SINUMERIK 840D control system is connected to the SIMODRIVE 611D converter system by means of a high-speed digital parallel bus.
DRF	Differential Resolver Function: An NC function, which generates an incremental zero offset in automatic mode in conjunction with an electronic handwheel.
Drift compensation	During the constant traversing phase of the CNC axes an automatic drift compensation is performed for the analog speed control. (SINUMERIK FM-NC).
Dynamic feedforward control	Inaccuracies in the contour caused by following errors can be almost entirely eliminated with the aid of dynamic, acceleration-dependent feedforward control. The result is extraordinary machining precision even at high tool path feed-rates. The feedforward control can be individually selected and deselected for each axis in the parts program.
E	
Editor	The editor is used to create, modify, add to, compress and insert programs/texts/program blocks.
Electronic handwheel	The electronic handwheels can be used to simultaneously traverse selected axes manually. The meaning of the lines on the handwheels is defined by increment weighting.
Exact stop	With a programmed exact stop instruction, the position stated in a block is approached precisely and very slowly, if necessary. In order to reduce the approach time, -> exact stop limits are defined for rapid traverse and feed.
Exact stop limit	When all path axes reach their exact stop limits, the control responds as if it had reached its destination point precisely. The -> part program continues execution at the next block.
External zero offset	Zero offset specified by the -> PLC.

F

Fast retraction from contour	When an interrupt arrives a motion can be initiated via the CNC machining program which allows the tool to be quickly retracted from the workpiece contour currently being machined. The angle of retraction and the absolute value of the path can also be parameterized. An interrupt routine can also be executed following the fast retraction of the tool. (SINUMERIK FM-NC, 840D).
Feed override	The programmed velocity is overridden by the current velocity setting via the machine control panel or by the PLC (0-200 %). The feedrate can also be corrected by a programmable percentage factor (1 - 200%) in the machining program.
File type	Possible types of data, e.g. parts programs, zero offset, R parameters etc.
Finished-part contour	Contour of the finished workpiece. See also -> blank.
Fixed machine point	A point defined uniquely by the machine tool, e.g. the reference point.
Fixed-point approach	Machine tools can approach fixed points such as a tool change point, loading point, pallet change point, etc. in a defined way. The coordinates of these points are stored in the control. Where possible, the control moves these axes in -> rapid traverse.
Focus	Frame (bold surround) which indicates the windows which can be edited.
Frame	A frame is an arithmetic rule that transforms one Cartesian coordinate system into another Cartesian coordinate system. A frame contains the components -> zero offset, -> rotation, -> scaling, -> mirroring.

G

General reset	During a general reset the following memories of the -> CPU are deleted:

- the -> working memory
- the read/write area of the -> load memory
- the -> system memory
- the -> backup memory

Geometry	Description of a -> workpiece in the -> workpiece coordinate system..
Geometry axis	Geometry axes are used to describe a 2- or 3-dimensional range in the workpiece coordinate system.

Global main program/subroutine	Every global main program/subroutine can only appear once under its own name in the directory, and it is not possible to have the same program name in different directories with different contents as a global program.
Ground	Ground is taken as the total of all linked inactive parts of a device which will not become live with a dangerous contact voltage even in the event of a malfunction.

H

Helical interpolation	Helical interpolation is especially suitable for easy machining inside or outside threads with form cutters and for milling lubrication grooves. The helix consists of two motions: 1. A circular movement in one plane 2. A linear movement perpendicular to this plane
HIGHSTEP	Summary of the programming options for the -> PLC in the AS300/AS400 system.
High-level CNC language	The high-level language offers: -> user variable, -> pre-defined user variable, -> system variable, -> indirect programming, -> mathematical and trigonometric functions, -> comparison operations and logic operations, -> program jumps and program branching, -> program coordination (SINUMERIK 840D), -> macro technology.
High-speed digital inputs/outputs	The digital inputs can be used for example to start fast CNC program routines (interrupt routines). The digital CNC outputs can be used to trigger fast, program-controlled switching functions. (SINUMERIK 840D).

I

Inch system	Dimension system which defines distances in inches and fractions of inches.
Inclined surface machining	Drilling and milling operations on workpiece surfaces that do not lie in the coordinate planes of the machine can be performed easily using the function "inclined-surface machining".
Increment	Traversed distance information via the number of increments. The number of increments can be stored as -> setting data or selected using keys labeled with 10, 100, 1000, 10 000.
Incremental dimension	A destination for axis traversal is defined by a distance to be covered

	and a direction referenced to a point already reached. See also -> absolute dimension.
Initialization block	Initialization blocks are special -> program blocks. They contain value assignments which are performed before program execution. The primary purpose of initialization blocks is to initialize predefined data or global user data.
Initialization files	It is possible to create an initialization file for each -> workpiece. Various variable assignments which are intended to apply specifically to one workpiece can be stored in this file.
Interconnecting cables	Interconnecting cables are 2-wire cables with 2 connector plugs which are either pre-fabricated or need to be fabricated by the user himself. These interconnecting cables connect the -> CPU via the -> multipoint interface (MPI) with a -> programming device or other CPUs.
Intermediate blocks	Traversing movements with tool offset selected (G41/G42) can be interrupted by a limited number of intermediate blocks (block without axis movements in the compensation plane) whereby the tool offset can still be correctly calculated. The permissible number of intermediate blocks which the control reads ahead can be set in system parameters.
Interpolator	Logical unit of the -> NCK which determines intermediate values for the movements to be traversed on the individual axes on the basis of destination positions specified in the parts program.
Interpolatory compensation	Interpolatory compensation can be used to compensate for leadscrew errors and measuring system errors (LEC, MSEC) caused during production.
interrupt routine	Interrupt routines are special -> subroutines which can be started on the basis of events (external signals) in the machining process. A parts program block which is currently being worked through is interrupted and the position of the axes at the point of interruption is automatically saved.
Inverse-time feedrate	With the SINUMERIK FM-NC and 840D, it is possible to program the time required to traverse the path of a block instead of programming the feedrate for the axis movement (G93).
I/O module	I/O modules represent the link between the CPU and the process. I/O modules are: • ->Digital input/output modules • ->Analog input/output modules • ->Simulator modules

J

JOG

Control operating mode (setup mode): In JOG mode, it is possible to set up the machine. Individual axes and spindles can be moved in this mode using the direction keys. Other functions available in Jog mode are -> reference point approach, -> repositioning and -> preset (setting an actual value).

K

Keyswitch

1. **S7-300**: The keyswitch is the operating mode switch of the -> CPU. A removable key is used to operate the keyswitch.
2. **840D/FM-NC**: The keyswitch on the -> machine control panel has 4 settings, to which functions are assigned by the operating system of the control. Three differently colored keys also belong to the keyswitch. These keys can be pulled out of the keyswitch in the indicated positions.

K$_ü$

Speed ratio

K$_v$

Servo gain factor, a control variable in a control loop.

L

Languages

The operator guidance display texts and the system messages are available in five system languages (diskette):
German, English, French, Italian and **Spanish**
Two of the above languages are available and selectable in the control.

Leadscrew error compensation.

Compensation for the mechanical inaccuracies of a ball screw participating in the feed. The control uses stored deviation values for the compensation.

Limit speed

Maximum/minimum (spindle) speed: The maximum speed of a spindle may be limited by values defined in the machine data, the -> PLC or -> setting data.

Linear axis

The linear axis is an axis which, in contrast to a rotary axis, describes a straight line.

Linear interpolation

The tool travels along a straight line to the destination point while machining the workpiece.

Load memory	For the CPU 314 of the -> PLC, the load memory is equal to the ➔ working memory.

Look Ahead With the **look ahead** function, a configurable number of traversing blocks is read in advance in order to calculate the optimum machining velocity.

Look-ahead detection of contour violations The control can recognize and signal the following types of collision:
1. The path distance is shorter than the tool radius.
2. The width of the inner corner is smaller than the tool diameter.

M

Macro technology Grouping of a set of instructions under a single identifier. The identifier in the program refers to the grouped set of instructions.

Machine Control operating area

Machine axes Axes which exist physically on the machine tool.

Machine control panel An operator panel on a machine tool with operating elements such as keys, rotary switches etc. and simple indicators such as LEDs. It is used to control the machine tool directly via the PLC.

Machine coordinate system System of coordinates based on the axes of the machine tool.

Machine zero A fixed point on the machine tool, which can be referenced by all (derived) measuring systems.

Machining channel Via a channel structure, parallel sequences of movements, such as positioning a loading gantry during machining, can shorten unproductive times. Here, a CNC channel must be regarded as a separate CNC control system with decoding, block preparation and interpolation.

Main block A block prefixed by ":" containing all the parameters required to start execution of a -> parts program.

Main program Parts program identified by a number or identifier in which further main programs, subroutines or -> cycles may be called.

MDA Control operating mode: Manual Data Automatic. In MDA mode, it is possible to enter individual program blocks or sequences of blocks without reference to a main program or subroutine and to then execute them immediately via the NC start key.

Messages	All messages programmed in the parts program and -> alarms recognized by the system are output on the operator panel in plain text with the date and time and a symbol indicating the cancel criterion. The display is divided into alarms and messages.

Measuring circuit

- SINUMERIK FM-NC: The required measuring circuits for axes and spindles are normally integrated in the control module. In total. A maximum of 4 axes and spindles can be realized, with a maximum of up to 2 spindles.
- SINUMERIK 840D: The position encoder evaluation is located in the SIMODRIVE 611D-drive modules. The maximum configuration is 8 axes and spindles in total, with up to 5 spindles.

Metric measurement system

Standardized system of units: for lengths in millimeters (mm), meters (m), etc.

Mirroring

Mirroring inverts the signs of the coordinate values of a contour with respect to an axis. Mirroring can be applied to more than one axis at a time.

Mode group

At all times all of the axles/spindles are assigned to precisely one channel. Each channel is assigned to one operating mode group. The same -> mode is always assigned to the channels in a mode group.

Mounting rail

A mounting rail is used to attach the modules of an S7-300.

Multipoint interface

The multipoint interface (MPI) is a 9-pole Sub-D interface. A configurable number of devices can be connected to a multipoint interface and then communicate with each other.

- Programming devices
- Operator control and monitoring equipment
- Further automation systems

The parameter block "Multipoint Interface MPI" of the CPU contains the -> parameters which define the properties of the multipoint interface.

N

Name of identifier

The words according to DIN 66025 are supplemented by the identifiers (names) for variables (computer variable, system variable, user variable), for subroutines, for vocabulary words and words with several address letters. In terms of the block structure, these supplements have the same significance as the words. Identifiers must be unique. The same identifier must not be used for different objects.

NC

Numerical Control: NC control incorporates all the components of the of the machine tool control system: -> NCK, -> PLC, -> MMC, -> COM.
Note: CNC (Computerized Numerical Control) is a more accurate term for the controls SINUMERIK 840D and FM-NC.

NCK

Numeric Control Kernel: Component of the NC control which executes -> parts programs and essentially coordinates the movements on the machine tool.

Network

The term "network" describes the connection of several S7-300 and other terminal devices, e.g. a programming device, via -> interconnecting cables. A data exchange takes place over the network between the connected devices.

Node number

The node number represents the "contact address" of a -> CPU or the -> programming device or any other intelligent periphery module if these are communicating via a -> network with each other. The node number is assigned to the CPU or the programming device with the S7 tool
-> "S7 configuration".

NURBS

Internal motion control and path interpolation are performed using NURBS (non-uniform rational B-splines). This provides a uniform internal method for all interpolations in the control (SINUMERIK 840D).

NRK

Numeric Robotic Kernel (operating system of the -> NCK)

O

OEM

For machine manufacturers who manufacture their own user interface or wish to integrate their own technology-specific functions in the control, free space has been left for individual solutions (OEM applications) for SINUMERIK 840D.

Offset memory	Data range in the control in which the tool offset data are stored.
Operating mode	An operating concept on a SINUMERIK control. The operating modes -> Jog, -> MDA and -> Automatic are defined.
Oriented spindle stop	Stops the workpiece spindle with a specified orientation angle, e.g. to perform an additional machining operation at a specific position.
Oriented tool retraction	RETTOOL: If machining is interrupted (because of tool breakage, for example), a program command can be used retract the tool with a defined orientation by a defined path.
Override	Manual or programmable control feature which enables the user to override programmed feedrates or speeds in order to adapt them to a specific workpiece or material.

P

Parameters	1. **S7-300**: A distinction is made between 2 types of parameters:
	- Parameters of a STEP 7 instruction
	  A parameter of a STEP 7 instruction is the address of the operand to be processed or a constant.
	- Parameters of a -> parameter block
	  A parameter of a parameter block determines the behavior of a module.
	2. **840D/FM-NC**:
	- Control operating area
	- Arithmetic parameter for which the programmer of the parts program can assign or request values as required in the program.

Parts program	A sequence of instructions to the NC control which combine to produce a specific -> workpiece. Likewise, performing a certain machining operation on a specific -> blank.
Parts program management	Part programs can be organized according to -> workpieces. The size of the user memory determines the number of programs and the amount of data that can be managed. Each file (programs and data) can be assigned a name comprising up to 24 alphanumeric characters.
Path axis	Path axes are all the machining axes in the -> channel which are controlled by the -> interpolator so that they start, accelerate, stop and reach their end positions simultaneously.

Path feed	Path feed acts on -> path axes. It represents the geometrical sum of the feeds on the participating -> geometry axes.
Path velocity	The maximum programmable path velocity depends on the input resolution. For example, with a resolution of 0.1 mm the maximum programmable path velocity is 1000 m/min.
PLC	-> Programmable Logic Controller. Component of the -> NC control: Programmable controller for processing the control logic of the machine tool.
PLC programming	The PLC is programmed using the **STEP 7** software. The STEP 7 programming software is based on the **WINDOWS** operating system, and combines the STEP 5 programming functions with additional innovative functional developments.
PLC program memory	• SINUMERIK FM-NC: PLC user memory of the CPU 314 is used to store the PLC user program and the user data together with the PLC basic program. On the S7-CPU314 version a user memory of 24 kByte is available for this purpose. • SINUMERIK 840D: PLC user memory is used to store the PLC user program and the user data together with the PLC basic program. The PLC user memory can be upgraded to 96 kByte with memory expansions.
Polar coordinates	A coordinate system, which defines the position of a point on a plane in terms of its distance from the zero point and the angle formed by the radius vector with a defined axis.
Polynomial interpolation	With polynomial interpolation, it is possible to generate many different curve characteristics, such as **straight line-, parabolic**-, and **exponential functions** (SINUMERIK 840D).
Positioning axis	Axis which performs an auxiliary movement on a machine tool (e.g. tool magazine, pallet transport). Positioning axes are axes that do not interpolate with -> path axes.
Power On	Switching the control off and back on again.
Preset	The preset function can be used to redefine the control zero in the machine coordinate system. Preset does not move the axes, but a new position value is entered for the current axis positions.
Program	1. Control operating area 2. Sequence of instructions to the control.

Program block Program blocks contain the main programs and subprograms for the -> parts programs.

Programmable frames Programmable -> frames can be used to define new coordinate system starting points dynamically while the parts program is running. A distinction is made between absolute definition using a new frame and additive definition with reference to an existing starting point.

Programmable logic controller Programmable logic controllers (PLC) are electronic controls, the function of which is stored as a program in the control unit. This means that the layout and wiring of the device do not depend on the function of the control. The programmable logic controller has the same structure as a computer; it consists of a CPU (central module) with memory, input/output modules and an internal bus system. The peripherals and the programming language are matched to the requirements of the control technology.

Programmable working area limitation Limitation of the motion space of the tool to a space defined by programmed limitations.

Programming device Programming device

Programming key Characters and character sequences, which have a defined meaning in the programming language for -> parts programs (see programming guide).

Protection zone Three-dimensional space within the -> working area which the tool tip is not permitted to enter.

Q

Quadrant error compensation Contour errors at quadrant transitions, which arise as a result of changing friction conditions on the guideways, can be virtually entirely eliminated with the quadrant error compensation. Parameterization of the quadrant error compensation is performed by means of a circuit test.

R

Rapid traverse The highest speed of an axis. It is used for example to move the tool from rest position to the -> workpiece contour or retract the tool from the contour.

Reference point	Point on the machine tool used to reference the measuring system of the -> machine axes.
Reference point approach	If the utilized distance measuring system is not an absolute value encoder then it is necessary to perform a reference point approach to ensure that the actual values returned by the measuring system match the machine coordinate values.
REPOS	1. Repositioning on the contour via the controls With the Repos function it is possible to re-approach the interruption point by means of the direction keys. 2. Repositioning on the contour via the program Several approach strategies are available through program commands: approaching the interruption point, approaching the block starting point, approaching the block end point, approaching a point on the path between the start of the block and the interruption point.
Retentive/non-volatile data	Data areas in data blocks and times, counters and markers are retentive (non-volatile) if their contents are not lost when the system is restarted or the mains supply is disconnected.
Rotation	Component of a -> frame which defines a rotation of the coordinate system through a specific angle.
Rotary axis	Rotary axes rotate a workpiece or tool to a defined angular position.
Rotary axis, endlessly turning	Depending on the application, the traversing range of a rotary axis can be selected to be limited to less than 360 degrees or to be endlessly turning in both directions. Endlessly-turning rotary axes are used for non-circular turning, grinding, and winding.
R parameters	Arithmetic parameter for which the programmer of the -> parts program can freely assign or request values in the program.
Rounding axis	Rounding axes rotate a workpiece or tool to an angular position corresponding to an indexing grid. On reaching a grid, the rounding axis is "in position".

S

S7-300 bus
The S7-300 bus is a serial databus by means of which the modules communicate with each other and are supplied with the required voltage. The connections between the modules are made with the -> bus connectors.

S7 configuration
S7 configuration is a tool with the aid of which modules can be parameterized. With S7 configuration, various -> parameter blocks of the -> CPU and the I/O modules are set on the -> programming unit. These parameters are transmitted to the CPU.

Safety functions
The controls contain watchdog monitors which are always active. These monitors detect problems in the CNC, PLC or machine in time to prevent damage to workpiece, tool or machine as far as possible. In the event of a malfunction the machining sequence is interrupted and the drives are stopped, the cause of the malfunction is saved and displayed as an alarm. At the same time, the PLC is informed that a CNC alarm is pending.

Scaling
Component of a -> frame, which causes axis-specific scale modifications.

Services
Control operating area

Serial RS-232-C interface
For data input/output there is
- a serial RS-232-C interface on the MMC module MMC100, and on
- the MMC modules MMC101 and MMC102 two RS-232-C interfaces

are available. Machining programs and manufacturer and user data can be loaded and saved via these interfaces.

Setting data
Data which communicate properties of the machine tool to the NC control in a way defined by the system software.

Simulator module
A simulator module is a module
- on which digital input variables can be simulated via control elements and
- digital output variables can be displayed.

Soft Key
A key whose name appears on an area of the screen. The selection of keys displayed is adapted dynamically to the operating situation. The freely assignable function keys (softkeys) are assigned defined functions in the software.

Software limit switches	Software limit switches limit the traversing range of an axis and prevent the slide from striking the hardware limit switch. 2 pairs of values can be assigned on each axis, which can then be activated separately via the -> PLC.
Spindles	The spindle functionality is divided into two power ranges.

1. Spindles: Speed or position-controlled analog spindle drives
   +10V   (SINUMERIK FM-NC)
   digital   (SINUMERIK 840D)
2. Auxiliary spindles: speed-controlled spindle drives, "auxiliary spindle" function package e.g. for driven tools.

Spline interpolation   Using spline interpolation, the control can obtain a very smooth curve from just a few defined interpolation points along a set contour.

Standard cycles   Standard cycles are available for frequently recurring machining tasks.

- for drilling/milling technology
- for turning technology (SINUMERIK FM-NC)

In the "Program" operating area, the available cycles are listed under the menu "Cycle Support". After selecting the desired machining cycle the required parameters for the value assignment are displayed in clear text.

Structure

- The SINUMERIK FM-NC is integrated into the CPU row of the SIMATIC S7-300. The module is 200 mm wide and fully encapsulated, with the same external structure as the SIMATIC S7-300 modules.
- The SINUMERIK 840D is integrated as a compact module into the SIMODRIVE 611D converter system. The dimensions correspond to a 50 mm wide SIMODRIVE 611D module. The SINUMERIK 840D module comprises the NCU module and the NCU box.

Subblock   Block prefixed by "N" containing information for a machining step such as position data.

Subprogram   A sequence of instructions in a -> parts program, which can be called repeatedly with various defining parameters. The subroutine is called from a main program. It is not possible to block every subroutine against unauthorized reading and displaying. -> Cycles are a form of subroutine.

Synchronization   Instructions in -> parts programs for coordination of sequences in different -> channels at specific machining points.

Synchronized axes	Synchronized axes take the same time to traverse as the geometry axes take for their path.
Synchronized actions	1. Auxiliary function output During the workpiece machining, technology functions (-> auxiliary functions) can be issued from the CNC program to the PLC. These auxiliary functions are used for example to control additional equipment for the machine tool, such as quills, grabbers, clamping chucks etc. 2. Fast auxiliary function output For switching functions which are time-critical, the confirmation times for the -> auxiliary functions are minimized, and unnecessary stopping points in the machining process can be avoided.
System memory	The system memory is a memory in the CPU in which the following data are stored: • Data required by the operating system • The operands times, counters, markers
System variables	A variable, which exists although it has not been programmed by the -> parts program programmer. It is defined by a data type and the variable name preceded by the character $. See also -> user-defined variable.

T

Tapping without a compensating chuck	This function allows threads to be tapped without a compensating chuck. By using the method whereby the spindle, as a rotary axis, and the drilling axis interpolate, threads can be cut to a precise final drilling depth (e.g. for blind hole threads) (requirement: spindles in axis operation).
Teach In	**Teach In** can be used to create or correct parts programs. The individual program blocks can be input via the keyboard and traversed immediately. Positions approached via the direction keys or the handwheel can also be saved. Additional details such as G functions, feeds or M functions can be input in the same block.
Text editor	-> Editor

Tool	A tool is a software tool for inputting and changing the -> parameters of a parameter block. Tools include: • -> S7 configuration • S7 TOP • S7 Info
Tool	A part used on the machine tool for machining. Examples of tools include lathe tools, milling cutters, drills, laser beams, etc.
Tool offset	By programming a **T function** (5 integer decades) in the block, you can select the tool. Up to 9 cutting edges (D addresses) can be assigned to every T number. The number of tools to be managed in the controller is specified in the configuring data.
Tool radius compensation	In order to be able to program a desired -> workpiece contour directly, the control must traverse a path equidistant to the programmed contour, taking into account the radius of the tool used. (G41/G42).
Transformation	Programming in a Cartesian coordinate system, execution in a non-Cartesian coordinate system (e.g. with machine axes as rotary axes).
Traversing range	The maximum permissible traversing range on linear axes is $\pm$ 9 decades. The absolute value depends on the selected input sensitivity and positioning resolution and the system of units used (inches or metric).

U

User-defined variable	The user can declare user-defined variables for any use in the -> parts program or data block (global user data). A definition contains a data type specification and the variable name. See also -> system variable.
User interface	The user interface (UI) is the display medium for a CNC control in the form of a screen. It is laid out with eight horizontal and eight vertical softkeys.
User memory	All program and data, such as part programs, subroutines, comments, tool compensations, and work offsets/frames, as well as channel- and program user data can be stored in the shared CNC user memory.

User program

User programs for the S7-300 automation systems are created using the programming language STEP 7. The user program has a modular layout and consists of individual blocks.
The basic block types are:
Code blocks: these blocks contain the STEP 7 commands.
Data blocks: these blocks contain the constants and variables for the STEP 7 program.

V

Variable definition

A variable definition includes the specification of a data type and a variable name. Variable names can be used to access the value of the variables.

Velocity control

In order to be able to achieve an acceptable traversing velocity on very short traverse movements within a single block, predictive velocity control can be set over several blocks (-> look ahead).

Vocabulary words

Words with a specific notation, which have a defined meaning in the programming language for -> parts programs.

W

Working area

Three-dimensional zone into which the tool tip can be moved on account of the physical design of the machine tool.
See also -> protection zone.

Working area limitation

With the aid of the working area limitation, the traversing range of the axes can be further restricted in addition to the limit switches. One value pair per axis may be used to describe the protected working area.

Working memory

The working area is a RAM area in the -> CPU which is accessed by the processor to access the user program during program execution .

Workpiece

Part to be created/machined by the machine tool.

Workpiece contour

Setpoint contour of the -> workpiece to be created/machined.

Workpiece coordinate system

The starting position of the workpiece coordinate system is the -> workpiece zero. In machining operations programmed in the workpiece coordinate system, the dimensions and directions refer to this system.

Workpiece zero	The workpiece zero is the starting point for the -> workpiece coordinate system. It is defined in terms of the distance from the machine zero.
X	
Y	
Z	
Zero offset	Specification of a new reference point for a coordinate system through reference to an existing zero point and a -> frame.

1. adjustable
   SINUMERIK FM-NC: Four independent zero offsets can be selected for each CNC axis.
   SINUMERIK 840D: A configurable number of adjustable zero offsets is available for each CNC axis. The offsets which can be selected via G functions are effective on an alternating basis.
2. external
   In addition to all displacements which define the position of the workpiece zero point, an external zero offset can be superimposed via
   - a handwheel (DRF offset) or
   - the PLC.
3. programmable
   Zero offsets are programmable for all path and positioning axes with the TRANS command.

C   References

General Documentation

/BU/ SINUMERIK & SIMODRIVE, Automation Systems for Machine Tools
Catalog NC 60
Order number: E86060-K4460-A101-A9
Order number: E86060-K4460-A101-A9-7600 (English)

/IKPI/ Industrial Communications and Field Devices
Catalog IK PI
Order number: E86060-K6710-A101-B2
Order number: E86060-K6710-A101-B2-7600 (English)

/ST7/ SIMATIC
Products for Totally Integrated Automation and Micro Automation
Catalog ST 70
Order number: E86060-K4670-A111-A8
Order number: E86060-K4670-A111-A8-7600 (English)

/Z/ MOTION-CONNECT
Connections & System Components for SIMATIC, SINUMERIK, MASTERDRIVES, and SIMOTION
Catalog NC Z
Order number: E86060-K4490-A001-B1
Order number: E86060-K4490-A001-B1-7600 (English)

Safety Integrated
Application Manual
The safety program for the world's industries
Order number: 6ZB5000-0AA01-0BA0
Order number: 6ZB5000-0AA02-0BA0 (English)

Electronic Documentation

/CD1/ The SINUMERIK System   (03. 04 edition)
DOC ON CD
(includes all SINUMERIK 840D/840Di/810D/802- and SIMODRIVE publications)
Order number: 6FC5298-7CA00-0BG0

User Documentation

/AUK/	SINUMERIK 840D/810D **AutoTurn** Short Operating Guide Order number: 6FC5298-4AA30-0BP2	(09.99 edition)
/AUP/	SINUMERIK 840D/810D **AutoTurn Graphic Programming System** Programming/Setup Order number: 6FC5298-4AA40-0BP3	(02.02 edition)
/BA/	SINUMERIK 840D/810D **MMC** Operator's Guide Order number: 6FC5298-6AA00-0BP0	(10.00 edition)
/BAD/	SINUMERIK 840D/840Di/810D Operator's Guide: **HMI Advanced** Order number: 6FC5298-6AF00-0BP3	(03.04 edition)
/BAH/	SINUMERIK 840D/840Di/810D Operator's Guide **HT 6** Order number: 6FC5298-0AD60-0BP3	(03.04 edition)
/BAK/	SINUMERIK 840D/840Di/810D **Short Operating Guide** Order number: 6FC5298-6AA10-0BP0	(02.01 edition)
/BAM/	SINUMERIK 810D/840D **ManualTurn** Operator's Guide Order number: 6FC5298-6AD00-0BP0	(08.02 edition)
/BAS/	SINUMERIK 840D/840Di/810D Operator's Guide **ShopMill** Order number: 6FC5298-6AD10-0BP2	(11.03 edition)
/BAT/	SINUMERIK 840D/810D Operator's Guide **ShopTurn** Order number: 6FC5298-6AD50-0BP2	(06.03 edition)
/BEM/	SINUMERIK 840D/810D Operator's Guide: **HMI Embedded** Order number: 6FC5298-6AC00-0BP3	(03.04 edition)
/BNM/	SINUMERIK 840D840Di//810D User's Guide **Measuring Cycles** Order number: 6FC5298-6AA70-0BP3	(03.04 edition)
/BTDI/	SINUMERIK 840D840Di//810D Motion Control Information System (MCIS) User's Guide **Tool Data Information** Order number: 6FC5297-6AE01-0BP0	(04.03 edition)
/CAD/	SINUMERIK 840D/840Di/810D Operator's Guide **CAD Reader** Order number: (is part of the Online Help)	(03.02 edition)

/DA/	SINUMERIK 840D/840Di/810D	(03.04 edition)
	Diagnostics Guide	
	Order number: 6FC5298-7AA20-0BP0	

/KAM/	SINUMERIK 840D/810D	(04.01 edition)
	Short Guide **ManualTurn**	
	Order number: 6FC5298-5AD40-0BP0	

/KAS/	SINUMERIK 840D/810D	(04.01 edition)
	Short Guide **ShopMill**	
	Order number: 6FC5298-5AD30-0BP0	

/KAT/	SINUMERIK 840D/810D	(07.01 edition)
	Short Guide **ShopTurn**	
	Order number: 6FC5298-6AF20-0BP0	

/PG/	SINUMERIK 840D/840Di/810D	(03.04 edition)
	Programming Guide **Fundamentals**	
	Order number: 6FC5298-7AB00-0BP0	

/PGA/	SINUMERIK 840D/840Di/810D	(03.04 edition)
	Programming Guide **Advanced**	
	Order number: 6FC5298-7AB10-0BP0	

/PGK/	SINUMERIK 840D/840Di/810D	(03.04 edition)
	Short Guide **Programming**	
	Order number: 6FC5298-7AB30-0BP0	

/PGM/	SINUMERIK 840D/840Di/810D	(11.02 edition)
	Programming Guide **ISO Milling**	
	Order number: 6FC5298-6AC20-0BP2	

/PGT/	SINUMERIK 840D/840Di/810D	(11.02 edition)
	Programming Guide **ISO Turning**	
	Order number: 6FC5298-6AC10-0BP2	

/PGC/	SINUMERIK 840D840Di//810D	(03.04 edition)
	Programming Guide **Cycles**	
	Order number: 6FC5298-7AB40-0BP0	

/PI /	PCIN 4.4	
	Software for Data Transfer to/from **MMC Modules**	
	Order number: 6FX2060-4AA00-4XB0 (English, German, French)	
	Order from: WK Fürth	

/SYI/	SINUMERIK 840Di	(02.01 edition)
	System Overview	
	Order number: 6FC5298-6AE40-0BP0	

Manufacturer/Service Documentation

a) Lists
/LIS/   SINUMERIK 840D/840Di/810D   (03.04 edition)
                  SIMODRIVE 611D
                  Lists
                  Order number: 6FC5297-7AB70-0BP0

b) Hardware
/ASAL/   SIMODRIVE 611, MASTERDRIVES VC/MC   (10.03 edition)
                  Planning Guide General Part for **Asynchronous Servo Motors**
                  Order number: 6SN1197-0AC62-0BP0

/APH2/   SIMODRIVE 611   (10.03 edition)
                  Planning Guide **Asynchronous Servo Motors 1PH2**
                  Order number: 6SN1197-0AC63-0BP0

/APH4/   SIMODRIVE 611   (10.03 edition)
                  Planning Guide **Asynchronous Servo Motors 1PH4**
                  Order number: 6SN1197-0AC64-0BP0

/APH7S/   SIMODRIVE 611   (03.04 edition)
                  Planning Guide **Asynchronous Servo Motors 1PH7**
                  Order number: 6SN1197-0AC66-0BP0

/APH7M/   MASTERDRIVES MC   (04.04 edition)
                  Planning Guide **Asynchronous Servo Motors 1PH7**
                  Order number: 6SN1197-0AC65-0BP0

/APL6/   MASTERDRIVES VC/MC   (03.04 edition)
                  Planning Guide **Asynchronous Servo Motors1PL6**
                  Order number: 6SN1197-0AC63-0BP0

/BH/   SINUMERIK 840D840Di//810D   (11.03 edition)
                  Operator Components Manual
                  Order number: 6FC5297-6AA50-0BP3

/BHA/   SIMODRIVE Sensor   (03.03 edition)
                  User Guide (HW) **Absolute Valuators with Profibus DP**
                  Order number: 6SN1197-0AB10-0YP2

/EMC/   SINUMERIK, SIROTEC, SIMODRIVE, SIMOTION   (06.99 edition)
                  Planning Guide **EMC Installation Guide**
                  Order number: 6FC5297-0AD30-0BP1

                  The up-to-date declaration of conformity can be viewed on the Internet at
                  http://www4.ad.siemens.de

                  Please enter the ID no. 15257461 in the "Find"/"Search" field (upper right) and click
                  "go".

/GHA/   SINUMERIK/SIMOTION   (02.03 edition)
                  ADI4 - Analog Drive Interface for 4 Axes
                  Manual
                  Order number: 6FC5297-0BA01-0BP1

/PFK6/	SIMODRIVE 611, MASTERDRIVES MC Planning Guide **Synchronous Servo Motors 1FK6** Order number: 6SN1197-0AD05-0BP0	(05.03 edition)
/PFK7/	SIMODRIVE 611, MASTERDRIVES MC Planning Guide **Synchronous Servo Motors 1FK7** Order number: 6SN1197-0AD06-0BP0	(01.03 edition)
/PFS6/	MASTERDRIVES MC Planning Guide **Synchronous Servo Motors 1FS6** Order number: 6SN1197-0AD08-0BP0	(07.03 edition)
/PFT5/	SIMODRIVE 611 Planning Guide **Synchronous Servo Motors 1FT5** Order number: 6SN1197-0AD01-0BP0	(05.03 edition)
/PFT6/	SIMODRIVE 611, MASTERDRIVES MC Planning Guide **Synchronous Servo Motors 1FT6** Order number: 6SN1197-0AD02-0BP0	(01.04 edition)
/PFU/	SINAMICS, MASTERDRIVES MICROMASTER **SIEMOSYN Motors 1FU8** Order number: 6SN1197-0AC80-0BP0	(09.03 edition)
/PHC/	SINUMERIK 810D **Configuring** Manual **(HW)** Order number: 6FC5297-6AD10-0BP1	(11.02 edition)
/PHD/	SINUMERIK 840D **Configuring** Manual **(HW)** Order number: 6FC5297-6AC10-0BP3	(11.03 edition)
/PJAL/	SIMODRIVE 611, MASTERDRIVES MC Planning Guide **Synchronous Servo Motors** **General Part for 1FT/1FK Motors** Order number: 6SN1197-0AD07-0AP1	(01.04 edition)
/PJAS/	SIMODRIVE 611, MASTERDRIVES VC/MC Planning Guide for Asynchronous Motors Contents: **General Part, 1PH2, 1PH4, 1PH7, 1PL6** Order number: 6SN1197-0AC61-0BP0	(06.04 Edition)
/PJFE/	SIMODRIVE Planning Guide **Built-In Synchronous Motors 1FE1** Three-Phase AC Motors for Main Spindle Drives Order number: 6SN1197-0AC00-0BP5	(03.04 edition)
/PJF1/	SIMODRIVE Installation Guide **Built-In Synchronous Motors 1FE1 051.-1FE1 147**. AC Motors for Main Spindle Drives Order number: 610.43000.02	(12.02 edition)

/PJLM/	SIMODRIVE	(06.02 edition)
	Planning Guide **Linear Motors 1FN1, 1FN3**	
	ALL General Information on the Linear Motor	
	1FN1 Three-Phase AC Linear Motor 1FN1	
	1FN3 Three-Phase AC Linear Motor 1FN3	
	CON Connections Order No.: 6SN1197-0AB70-0AP4	

/PJM2/	SIMODRIVE 611, MASTERDRIVES MC	(03.04 edition)
	Planning Guide for Synchronous Servo Motors	
	Contents: General Part, 1FT5, 1FT6, 1FK6, 1FK7, 1FS6	
	Order number: 6SN1197-0AC20-0BP0	

/PJTM/	SIMODRIVE	(05.03 edition)
	Planning Guide **Built-In Torque Motors 1FW6**	
	Order number: 6SN1197-0AD00-0BP1	

/PJU/	SIMODRIVE 611	(02.03 edition)
	Planning Guide **Converters**	
	Order number: 6SN1197-0AA00-0BP6	

/PKTM/	MASTERDRIVES	(03.04 edition)
	Planning Guide **Complete Torque Motors 1FW3**	
	Order number: 6SN1197-0AC70-0BP0	

/PMH/	SIMODRIVE Sensor	(07.02 edition)
	Planning/Installation Guide	
	Hollow-Shaft Measuring System SIMAG H	
	Order number: 6SN1197-0AB30-0BP1	

/PMH2/	SIMODRIVE Sensor	(03.04 edition)
	Planning/Installation Guide	
	Hollow-Shaft Measuring System SIMAG H2	
	Order number: 6SN1197-0AB31-0AP0	

/PMHS/	SIMODRIVE	(12.00 edition)
	Installation Guide **Measuring System for Main Spindle Drives**	
	Gear Wheel Encoder SIZAG2	
	Order number: 6SN1197-0AB00-0YP3	

/PMS/	SIMODRIVE	(03.04 edition)
	Planning Guide	
	ECO Motor Spindle for Main Spindle Drives 2SP1	
	Order number: 6SN1197-0AD04-0AP1	

/PPH/	SIMODRIVE	(12.01 edition)
	Planning Guide **1PH2/1PH4/1PH7 Motors**	
	Three-Phase Asynchronous Motors for Main Spindle Drives	
	Order number: 6SN1197-0AC60-0BP0	

/PPM/	SIMODRIVE	(11.01 edition)
	Planning Guide	
	Hollow-Shaft Motors for Main Spindle Drives **1PM4 and 1PM6**	
	Order number: 6SN1197-0AD03-0BP0	

c) Software

/FB1/	SINUMERIK 840D/840Di/810D/FM-NC	(03.04 edition)

Description of Function **Basic Machine (Part 1)**
(the various manuals are listed below)
Order number: 6FC5297-6AC20-0BP3

A2 Miscellaneous Interface Signals
A3 Axis Monitoring, Protection Areas
B1 Path Control Mode, Exact Positioning and Look Ahead
B2 Acceleration
D1 Diagnostic Tools
D2 Interactive Programming
F1 Traverse to Fixed Stop
G2 Velocities, Setpoint/Actual Value Systems, Closed-Loop Control
H2 Auxiliary Function Output to PLC
K1 Mode Group, Channels, Program Operation Mode
K2 Axis Types, Coordinate Systems, Frames,
   Actual-Value System for Workpiece, External Zero Offset
K4 Communication
N2 EMERGENCY STOP
P1 Traverse Axes
P3 Basic PLC Program
R1 Reference Point Approach
S1 Spindles
V1 Feeds
W1 Tool offset

/FB2/	SINUMERIK 840D/840Di/810D	(03.04 edition)

Description of Functions **Extended Functions (Part 2)**
including FM-NC: Turning, Stepping Motor
(the various manuals are listed below)
Order number: 6FC5297-7AC30-0BP0

A4 Digital and Analog NCK Devices
B3 Multiple Operator Panels and NCUs
B4 Operation via PG/PC
F3 Remote Diagnosis
H1 Manual Traverse and Handwheel Traverse
K3 Corrections
K5 Mode Groups, Channels, Axis Replacement
M1 Kinematic Transformation
M5 Measuring
N3 Software Cams, Clear Signals
N4 Punching and Nibbling
P2 Positioning Axes
P5 Oscillating
R2 Rotary Axes
S3 Synchronous Spindle
S5 Synchronized Actions (SW 3 and lower, higher /FBSY/)
S6 Stepper Motor Control
S7 Memory Configuration
T1 Indexing Axes
W3 Tool Change
W4 Grinding

/FB3/	SINUMERIK 840D/840Di/810D	(03.04 edition)
	Description of Functions **Special Functions (Part 3)**	
	(the various manuals are listed below)	
	Order number: 6FC5297-6AC80-0BP2	

F2	3-Axis to 5-Axis Transformation	
G1	Gantry Axes	
G3	Clock Times	
K6	Contour Tunnel Monitoring	
M3	Coupled Axes and ESR	
S8	Constant workpiece speed for centerless grinding	
S9	Setpoint Circuit (S9)	
T3	Tangential Control	
TE0	Installation and Activation of Compile Cycles	
TE1	Distance Control	
TE2	Analog Axes	
TE3	Speed/Torque Coupling Master-Slave	
TE4	Handling Transformation Package	
TE5	Setpoint Exchange	
TE6	MKS Coupling	
TE7	Resynchronization - Retrace Support	
TE8	Pulse-Independent Path-Synchronized Switching Signal Output	
V2	Preprocessing	
W5	3D Tool Radius Compensation	

/FBA/	SIMODRIVE 611D/SINUMERIK 840D/810D	(03.04 edition)
	Description of Functions **Drive Functions**	
	(the various sections are listed below)	
	Order number: 6SN1197-0AA80-1BP0	
DB1	Operating Messages/Alarm Responses	
DD1	Diagnostic Functions	
DD2	Speed Control Loop	
DE1	Extended Drive Functions	
DF1	Enable Commands	
DG1	Encoder Parameter Settings	
DL1	Linear Motor MD	
DM1	Calculating Motor/Power Section Parameters and Controller Data	
DS1	Current Control Loop	
DÜ1	Monitors/Limitations	

/FBAN/	SINUMERIK 840D/SIMODRIVE 611 DIGITAL	(02.00 edition)
	Description of Functions **ANA MODULE**	
	Order number: 6SN1197-0AB80-0BP0	

/FBD/	SINUMERIK 840D	(07.99 edition)
	Description of Functions **Digitizing**	
	Order number: 6FC5297-4AC50-0BP0	
DI1	Start-Up	
DI2	Scan with Tactile Sensor (scancad scan)	
DI3	Scan with Laser (scancad laser)	
DI4	Milling Program Generation (scancad mill)	

/FBDM/	SINUMERIK 840D/840Di/810D	(09.03 edition)
	Description of Functions **NC Program Management**	
	DNC Machine	
	Order number: 6FC5297-1AE81-0AP0	

/FBDN/	SINUMERIK 840D/840Di/810D	(03.03 edition)
	Motion Control Information System (MCIS)	
	Description of Functions **NC Program Management DNC**	
	Order number: 6FC5297-1AE80-0BP0	
	DN1 DNC Plant/DNC Cell	
	DN2 DNC IFC SINUMERIK	
/FBFA/	SINUMERIK 840D/840Di/810D	(11.02 edition)
	Description of Functions **ISO Dialects for SINUMERIK**	
	Order number: 6FC5297-6AE10-0BP3	
/FBFE/	SINUMERIK 840D/810D	(03.04 edition)
	Motion Control Information System (MCIS)	
	Description of Functions **Remote Diagnosis**	
	Order number: 6FC5297-0AF00-0BP3	
	FE1 Remote Diagnosis (ReachOut)	
	FE3 RCS Host/RCS Viewer (pcAnywhere)	
/FBH/	SINUMERIK 840D/840Di/810D	
	HMI configuring package	(11.02 edition)
	Order no.: (included with the software)	
	Part 1 User's Guide	
	Part 2 Description of Functions	
/FBH1/	SINUMERIK 840D/840Di/810D	
	HMI configuring package	(03.03 edition)
	ProTool/Pro Option SINUMERIK	
	Order number: (included with the software)	
/FBHL/	SINUMERIK 840D/SIMODRIVE 611 digital	(10.03 edition)
	Description of Functions **HLA Module**	
	Order number: 6SN1197-0AB60-0BP3	
/FBIC/	SINUMERIK 840D/840Di/810D	(06.03 edition)
	Motion Control Information System (MCIS)	
	Description of Functions **TDI Ident Connection**	
	Order number: 6FC5297-1AE60-0BP0	
/FBMA/	SINUMERIK 840D/810D	(08.02 edition)
	Description of Functions **ManualTurn**	
	Order number: 6FC5297-6AD50-0BP0	
/FBO/	SINUMERIK 840D/810D	(09.01 edition)
	Description of Functions Configuring of **OP 030 Operator Interface**	
	(the various sections are listed below)	
	Order number: 6FC5297-6AC40-0BP0	
	BA Operating Instructions	
	EU Development Environment (Project Planning Package)	
	PSE Introduction to Configuring the Operator Interface	
	(IK Screen Kit: Software Update and Configuration)	

/FBP/	SINUMERIK 840D Description of Functions **C-PLC Programming** Order number: 6FC5297-3AB60-0BP0	(03.96 edition)
/FBR/	SINUMERIK 840D/840Di/810D Description of Functions **Computer Linking RPC SINUMERIK** Order number: 6FC5297-6AD61-0BP0 NFL Interface to Central Production Computer NPL Interface to PLC/NCK	(01.04 edition)
/FBSI/	SINUMERIK 840D/SIMODRIVE Description of Functions SINUMERIK **Safety Integrated** Order number: 6FC5297-6AB80-0BP2	(11.03 edition)
/FBSP	SINUMERIK 840D/840Di/810D Description of Functions **ShopMill** Order number: 6FC5297-6AD80-0BP2	(11.03 edition)
/FBST/	SIMATIC Description of Functions **FM STEPDRIVE/SIMOSTEP** Order number: 6SN1197-0AA70-0YP4	(01.01 edition)
/FBSY/	SINUMERIK 840D/810D Description of Functions **Synchronized Actions** Order number: 6FC5297-7AD40-0BP2	(03.04 edition)
/FBT/	SINUMERIK 840D/810D Description of Functions **ShopTurn** Order number: 6FC5297-6AD70-0BP2	(03.04 edition)
/FBTC/	SINUMERIK 840D/810D IT Solutions Description of Functions **Tool Data Communication SinTDC** Order number: 6FC5297-5AF30-0BP0	(01.02 edition)
/FBTD/	SINUMERIK 840D/810D IT Solutions Description of Functions **Tool Information Systems (SinTDI)** with Online Help Order number: 6FC5297-6AE00-0BP0	(02.01 edition)
/FBTP/	SINUMERIK 840D/840Di/810D Motion Control Information System (MCIS) Description of Functions **Preventive Maintenance TPM** Order number: Document is included with the software	(01.03 edition)
/FBU/	SIMODRIVE 611 universal/universal E Description of Functions **Closed-Loop Control Component** **for Speed Control and Positioning** Order number: 6SN1197-0AB20-0AP9	(03.04 edition)
/FBU2/	SIMODRIVE 611 **universal** Installation Guide (included with every SIMODRIVE 611)	(04.02 edition)

/FBW/	SINUMERIK 840D/810D	(11.02 edition)
	Description of Functions **Tool Management**	
	Order number: 6FC5297-6AC60-0BP1	
/HBA/	SINUMERIK 840D/840Di/810D	(03.02 edition)
	Manual **@Event**	
	Order number: 6AU1900-0CL20-0AA0	
/HBI/	SINUMERIK 840Di	(03.04 edition)
	Manual **SINUMERIK 840Di**	
	Order number: 6FC5297-7AE60-0AP0	
/INC/	SINUMERIK 840D840Di//810D	(06.03 edition)
	System Overview **Commissioning Tool SINUMERIK SinuCOM NC**	
	Order number: (part of the Online Help for the Startup Tool)	
/PGA1/	SINUMERIK 840D/840Di/810D	(03.04 edition)
	List Guide **System Variables**	
	Order number: 6FC5297-7AE10-0AP0	
/PJE/	SINUMERIK 840D/810D	(08.01 edition)
	Description of Functions **HMI Embedded Configuring Package**	
	Software Update, Configuration, Installation	
	Order number: 6FC5297-6EA10-0BP0	
/PS/	SINUMERIK 840D/810D	(09.03 edition)
	Planning Guide - Planning Syntax	
	This document is part of the SW delivery and is available as a PDF file.	
/POS1/	SIMODRIVE **POSMO A**	(08.03 edition)
	User Guide **Decentralized Positioning Motor on PROFIBUS DP**	
	Order number: 6SN2197-0AA00-0BP5	
/POS2/	SIMODRIVE **POSMO A**	(08.03 edition)
	Installation Guide (included with every POSMO A)	
/POS3/	SIMODRIVE POSMO SI/CD/CA	(03.04 edition)
	User Guide **Distributed Servo Drive Systems**	
	Order number: 6SN2197-0AA20-0AP6	
/POS4/	SIMODRIVE **POSMO SI**	(08.03 edition)
	Installation Guide (included with every POSMO SI)	
/POS5/	SIMODRIVE **POSMO CD/CA**	(08.03 edition)
	Installation Guide (included with every POSMO CD/CA)	
/S7H/	SIMATIC S7-300	(2002 edition)
	Installation Manual **Technological Functions**	
	Order number: 6ES7398-8AA03-8BA0	
	- Reference Manual: CPU Data (HW Description)	
	- Reference Manual: Module Data	

/S7HT/	SIMATIC S7-300 Manual **STEP 7, Fundamentals, V. 3.1** Order number: 6ES7810-4CA02-8BA0	(03.97 edition)
/S7HR/	SIMATIC S7-300 Manual **STEP 7, Reference Manuals, V. 3.1** Order number: 6ES7810-4CA02-8BR0	(03.97 edition)
/S7S/	SIMATIC S7-300 **FM 353 for Stepper Drive** Positioning Module Order together with configuring package	(04.02 edition)
/S7L/	SIMATIC S7-300 **FM 354 for Servo Drive** Positioning Module Order together with configuring package	(04.02 edition)
/S7M/	SIMATIC S7-300 **FM 357.2** Multimodule for Servo and Stepper Drives Order together with configuring package	(01.03 edition)
/SP/	SIMODRIVE 611-A/611-D **SimoPro 3.1** Program for Configuring Machine-Tool Drives Order number: 6SC6111-6PC00-0BA⊤ Order from: WK Fürth	

**d) Comm-
issioning**

/BS/ SIMODRIVE 611 analog (10.00 edition)
Description **Start-Up Software for
Main Spindle and Asynchronous Motor Modules Version 3.20**
Order number: 6SN1197-0AA30-0BP1

/IAA/ SIMODRIVE 611A (10.00 edition)
Installation and Start-Up Guide
Order number: 6SN1197-0AA60-0BP6

/IAC/ SINUMERIK 810D (11.02 edition)
Installation and Start-Up Guide
(including description of SIMODRIVE 611D start-up
software)
Order number: 6FC5297-6AD20-0BP1

/IAD/ SINUMERIK 840D/SIMODRIVE 611D (03.04 edition)
Installation and Start-Up Guide
(including description of SIMODRIVE 611D start-up
software)
Order number: 6FC5297-7AB10-0BP0

/IAM/ SINUMERIK 840D/840Di/810D (03.04 edition)
HMI Installation and Start-Up Guide
Order number: 6FC5297-6AE20-0BP3
AE1 Updates/Options
BE1 Expanding the Operator Interface
HE1 Online Help
IM2 Start-Up HMI Embedded
IM4 Start-Up HMI Advanced
TX1 Creating Foreign Language Texts

∎

SINUMERIK 840D/840Di/810D Operator's Guide Handheld Terminal 6 (BAH) - 03.04 Edition A-179

D   Index

<table>
<tr>
<td>
To
Siemens AG

A&D MC BMS
P.O. Box 3180

D-91050 Erlangen

Phone +49 (0) 180 / 5050-222 [hotline]
Fax +49 (0) 9131 / 98-2176 [documentation]
Email: motioncontrol.docu@erlf.siemens.de
</td>
<td>
Suggestions

Corrections
</td>
</tr>
</table>

From Name: Company/Dept. Address: _____ Zip code: _____ City: _____ Phone: _____ / _____ Fax: _____ / _____	For Publication/Manual: SINUMERIK 840D/840Di/810D Handheld Terminal 6 User Documentation

	Operator's Guide Order No.: 6FC5298-0AD60-0BP3 03.04 Edition
	Should you come across any printing errors when reading this publication, please notify us using this form. Suggestions for improvements are also welcome.

Suggestions and/or corrections

Documentation Overview SINUMERIK 840D/840Di/810D (04.2004)

General Documentation

SINUMERIK
840D/810D

Brochure

SINUMERIK
840D/840Di/
810D/
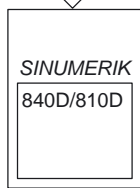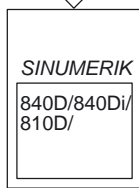
Catalog
Ordering info.
NC 60 ***)**

Safety Integrated
Application Manual

Safety Interated
Application Manual

User Documentation

SINUMERIK
840D/810D/
FM–NC
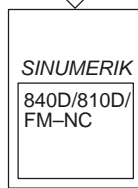
AutoTurn
– Short Guide
– Programming/
  Setup

SINUMERIK
840D/840Di
810D

Operator's Guide
– **HT 6**

SINUMERIK
840D/840Di/
810D

Diagnostics–guide *)

SINUMERIK
840D/840Di/
810D
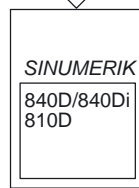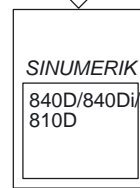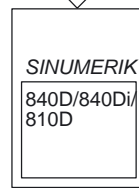
Operators Guide *)
– Short Guide
– HMI Embedded
– HMI Advanced

User Documentation

SINUMERIK
840D/840Di/
810D
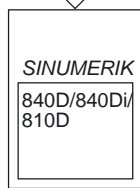
Program. Guide
– Short Guide
– Fundamentals ***)**
– Advanced. ***)**
– Cycles
– Measuring Cycles
– ISO Turning/Milling

SINUMERIK
840D/810D
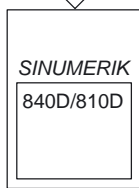
Operator's Guide
– **ManualTurn**
– Short Guide ManualTurn
– **ShopMill**
– Short Guide ShopMill
– **ShopTurn**
– Short Guide ShopTurn

SINUMERIK
840Di

Systemoverview

Manufacturer/Service Documentation

SINUMERIK
840D/840Di/
810D
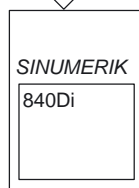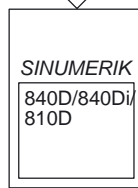
Lists
System Variables

SINUMERIK

Configuring (HW) ***)**
– 810D
– 840D

SINUMERIK
840D/840Di
810D

Operator Components (HW) ***)**

SINUMERIK
840D/840Di/
810D
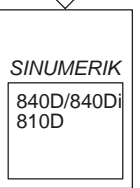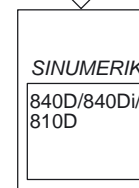
Description of Functions
– **ManualTurn**
– **ShopMill**
– **ShopTurn**

Manufacturer/Service Documentation
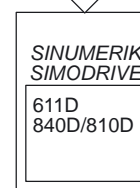
SINUMERIK SIMODRIVE
611D
840D/810D

Description of Functions
Synchronized Actions

SINUMERIK
840D/840Di/
810D

Desription of Functions
Drive Functions*)

SINUMERIK
840D/840Di/
810D
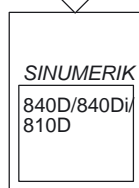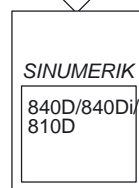
Desciption of Functions
– Basic Machine ***)**
– Extended Functions
– Special Functions

SINUMERIK
840D/810D

Desription of Functions
Tool Manage–ment

SINUMERIK
840D/810D
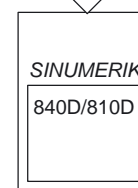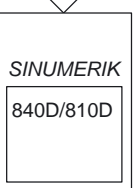
Configuring Kit
HMI Embedded

SINUMERIK
840D/810D

Description of Functions
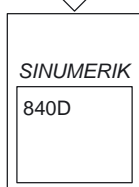Operator Interface
OP 030

SINUMERIK
840D/810D

MCIS
–Computer Link
–Tool Data Information System
–NC Data Management
–NC Data Transfer
–Tool Data Communication

Manufacturer/Service Documentation
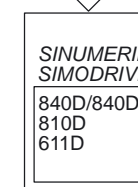
SINUMERIK SIMODRIVE

Description of Functions
SINUMERIK
Safety Integrated

SINUMERIK
840D

Description of Functions
Digitizing

SINUMERIK SIMODRIVE
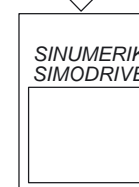
Installation & Start–up Guide. *)
– 810D
– 840D/611D
– HMI

SINUMERIK SIMODRIVE
840D/840Di
810D
611D

– **Lists*)**
– **Lists Syst. Var.**

SINUMERIK SIMODRIVE
840D
611D
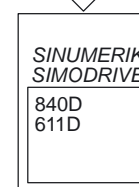
Description of Functions
Linear motor

SINUMERIK SIMODRIVE
840D
611D
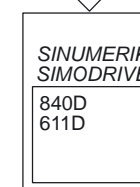
Description of Functions
– **Hydraulics Modules**
– **Analog Modules**
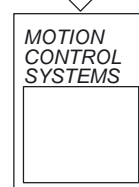
MOTION CONTROL SYSTEMS

EMV Guidelines

Electronic Documentation

SINUMERIK SIMODRIVE
840D/840Di/
810D
611, Motoren

DOC ON CD *)
The SINUMERIK System

Manufacturer/Service Documentation

SINUMERIK
840D/840Di/
810D
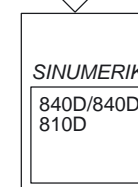
Description of Functions
ISO–Dialects for SINUMERIK
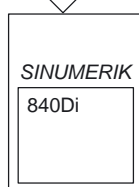
SINUMERIK
840Di

Manual
(HW + Installation and Start–up)

SINUMERIK
840D/810D

Desription of Functions
Remote Diagnosis

SINUMERIK
840D/840Di/
810D

**Manual
@ Event**

*) These documents are a minimum requirement