SIEMENS

SINUMERIK

SINUMERIK 840D sl
HMI sl Universal

Operating Manual

Valid for
SINUMERIK 840D sl/840DE sl control system

Software	Version
NCU system software for 840D sl/840DE sl	2.3
with HMI sl	2.0

08/2006
6FC5398-6AP10-0BA0

Safety Guidelines

This manual contains notices you have to observe in order to ensure your personal safety, as well as to prevent damage to property. The notices referring to your personal safety are highlighted in the manual by a safety alert symbol, notices referring only to property damage have no safety alert symbol. These notices shown below are graded according to the degree of danger.

Danger

indicates that death or severe personal injury **will** result if proper precautions are not taken.

Warning

indicates that death or severe personal injury **may** result if proper precautions are not taken.

Caution

with a safety alert symbol, indicates that minor personal injury can result if proper precautions are not taken.

Caution

without a safety alert symbol, indicates that property damage can result if proper precautions are not taken.

Notice

indicates that an unintended result or situation can occur if the corresponding information is not taken into account.

If more than one degree of danger is present, the warning notice representing the highest degree of danger will be used. A notice warning of injury to persons with a safety alert symbol may also include a warning relating to property damage.

Qualified Personnel

The device/system may only be set up and used in conjunction with this documentation. Commissioning and operation of a device/system may only be performed by **qualified personnel**. Within the context of the safety notes in this documentation qualified persons are defined as persons who are authorized to commission, ground and label devices, systems and circuits in accordance with established safety practices and standards.

Prescribed Usage

Note the following:

Warning

This device may only be used for the applications described in the catalog or the technical description and only in connection with devices or components from other manufacturers which have been approved or recommended by Siemens. Correct, reliable operation of the product requires proper transport, storage, positioning and assembly as well as careful operation and maintenance.

Trademarks

All names identified by ® are registered trademarks of the Siemens AG. The remaining trademarks in this publication may be trademarks whose use by third parties for their own purposes could violate the rights of the owner.

Disclaimer of Liability

We have reviewed the contents of this publication to ensure consistency with the hardware and software described. Since variance cannot be precluded entirely, we cannot guarantee full consistency. However, the information in this publication is reviewed regularly and any necessary corrections are included in subsequent editions.

Siemens AG
Automation and Drives
Postfach 48 48
90437 NÜRNBERG
GERMANY

Order No.: 6FC5398-6AP10-0BA0
Edition 10/2006

Foreword

Foreword

Structure of the documentation

The SINUMERIK documentation is organized in 3 parts:

- General documentation
- User documentation
- Manufacturer/service documentation

An overview of publications, which is updated monthly and also provides information about the language versions available, can be found on the Internet at:

http://www.siemens.com/motioncontrol
Select "Support" → "Technical Documentation" → "Overview of Publications".

The Internet version of DOConCD (DOConWEB) is available at:

http://www.automation.siemens.com/doconweb

Information about training courses and FAQs (Frequently Asked Questions) can be found at the following website:

http://www.siemens.com/motioncontrol under menu option "Support"

Target group

This documentation is intended for users of universal machines running the HMI sl software.

Standard scope

This documentation only describes the functionality of the standard version. Extensions or changes made by the machine tool manufacturer are documented by the machine tool manufacturer.

Other functions not described in this documentation might be executable in the control. However, no claim can be made regarding the availability of these functions when the equipment is first supplied or in the event of servicing.

For the sake of simplicity, this documentation does not contain all detailed information about all types of the product and cannot cover every conceivable case of installation, operation, or maintenance.

Technical Support

In case of questions, please contact us through the following hotline:

Europe and Africa time zone:

A&D Technical Support

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

Fax: +49 (0) 180 / 5050 - 223

Internet: http://www.siemens.com/automation/support-request

E-mail: mailto:adsupport@siemens.com

Asia and Australia time zone

A&D Technical Support

Phone.: +86 1064 719 990

Fax: +86 1064 747 474

Internet: http://www.siemens.com/automation/support-request

E-mail: mailto:adsupport@siemens.com

American time zone

A&D Technical Support

Phone.: +1 423 262 2522

Fax: +1 423 262 2289

Internet: http://www.siemens.com/automation/support-request

E-mail: mailto:adsupport@siemens.com

Note

Country-specific telephone numbers for technical support are provided under the following Internet address:

Enter http://www.siemens.com/automation/service&support

Questions about the manual

If you have any queries (suggestions, corrections) in relation to this documentation, please fax or e-mail us:

Fax: +49 (0) 9131 / 98 - 63315

E-mail: mailto:motioncontrol.docu@siemens.com

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

Internet address

http://www.siemens.com/motioncontrol

EC declaration of conformity

The EC Declaration of Conformity for the EMC Directive can be found/obtained

- the internet:

  http://www.ad.siemens.de/csinfo

  under the Product/Order No. 15257461
- at the relevant branch office of the A&D MC Business Division of

  Siemens AG

Table of contents

Introduction

1

1.1 Product overview

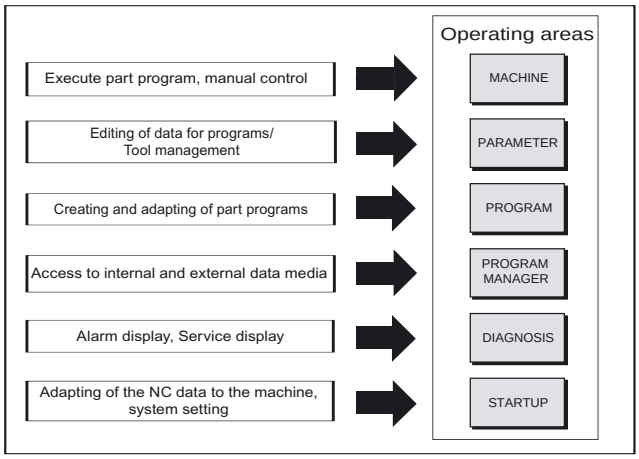
The SINUMERIK controller is a CNC (Computerized Numerical Control) control system for machine tools.

You can use the CNC control system to implement the following basic functions in conjunction with a machine tool:

- Creation and adaptation of part programs
- Execution of part programs
- Manual control
- Access to internal and external data media
- Editing of data for programs
- Displaying and troubleshooting alarms
- Editing of machine data

Operating areas

The basic functions are grouped in the following operating areas in the control:



1.2 Operator panel fronts

1.2.1 Overview

Introduction

The display (screen) and operation (e.g. hardkeys and softkeys) of the HMI sl operator interface occurs via the panel front.

In this example, the OP 010 operator panel front is used to illustrate the components that are available for operating the control and machine tool.

Operator controls and indicators

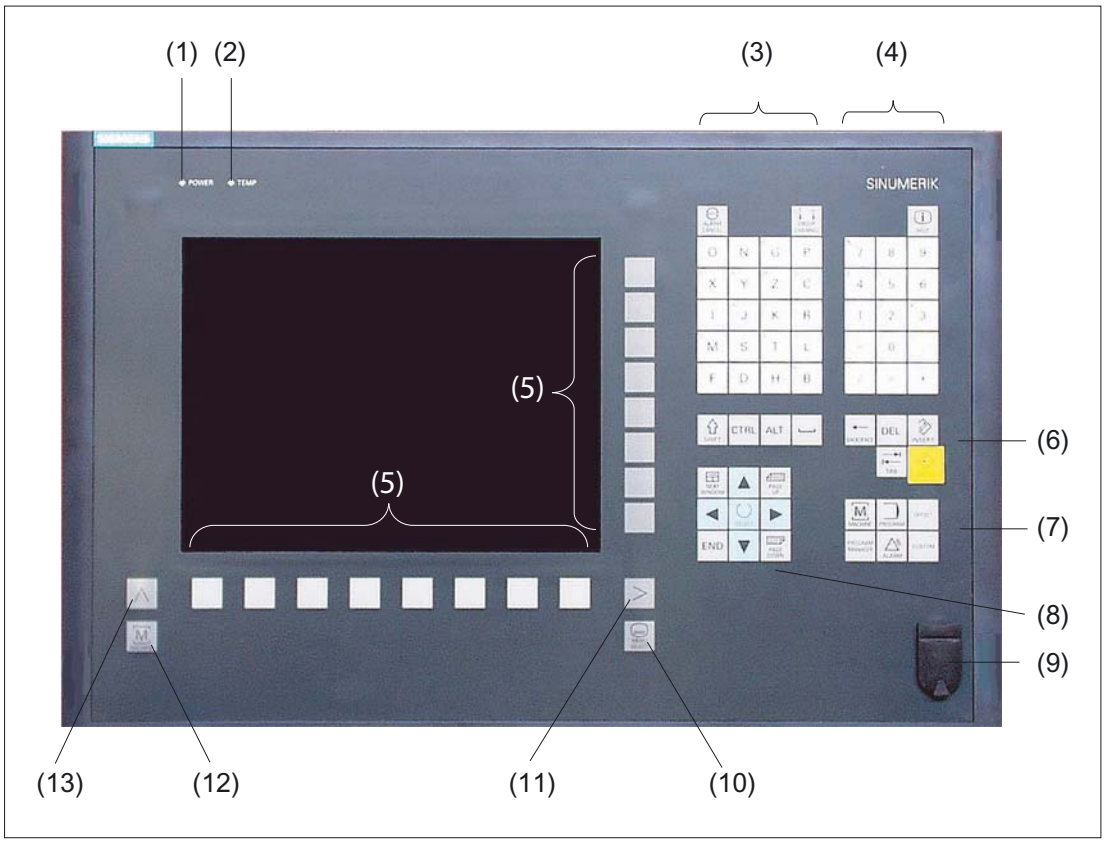


Figure 1-1 View of OP 010 operator panel front

1 Status LED: POWER
2 Status LED: TEMP
(illuminated LEDs indicate increased wear)
3 Alphabetic key group

4 Numerical key group
5 Softkeys
6 Control key group
7 Hotkey group
8 Cursor key group
9 USB interface
10 Menu Select key
11 Menu forward button
12 Machine area button
13 Menu back button

References

An exact description as well as a view of the other serviceable panel fronts may be found in /BH/, Equipment Manual Panel Components 840D sl/840 Di sl

1.2.2 Keys of the operator panel

The following buttons are available for operation of the control and the machine tool.

Keys

Table 1-1 Keys of the operator panel

Key	Function
	**Alarm Cancel** Cancel alarms and messages that are marked with this symbol.
	**Channel** Select channel or continue.
	**Help** No function.
	**Next Window** Continue to next window.
 	**Page Up or Page Down** Page up or down in a screen.

Key	Function
▲ ◀ ▶ ▼	**Cursor** Input focus/cursor between different fields. Navigate rows or characters. Use the right cursor to open a directory or program in the editor. Use the left cursor to switch to a higher level in the directory tree.
SELECT	**Select** Choose one of a number of options presented. Select an element.
END	**End** Moves the cursor to the last input field in a parameterization screen form or table.
BACKSPACE	**Backspace** Delete the value in the input field. In insertion mode, it deletes the character after the cursor.
TAB	**Tab** Indent the cursor by several characters.
SHIFT	**Shift** Depress the Shift key to enter the upper character shown on the dual input keys.
CTRL	**Ctrl** Use the following key combinations to navigate in the G code editor: • <Ctrl + Home> : Jump to the beginning. • <Ctrl + End> : Jump to the end. Selection
ALT	**Alt** Irrelevant
DEL	**Del** • Delete the value in the input field. • In insertion mode, it deletes the character after the cursor.
INSERT	**Insert** Activate insertion mode.
INPUT	**Input** • Finish the entry of a value in the input field. • Open a directory or program.
ALARM	**Alarm - only OP 010 and OP 010C** Irrelevant
PROGRAM	**Program - only OP 010 and OP 010C** Irrelevant
OFFSET	**Offset - only OP 010 and OP 010C** Irrelevant.
PROGRAM MANAGER	**Program Manager - only OP 010 and OP 010C** Irrelevant.
∧	**Return** Jump to the next highest menu level.

Key	Function
	**Machine** Open the "Machine" operating area.
	**Extension** Advance the horizontal softkey bar.
	**Menu Select** Call the main menu for operating area selection.

1.3 Machine control panels

1.3.1 Overview

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

You use the machine control panel to initiate actions on the machine tool such as traversing an axis or starting the machining of a workpiece.

1.3.2 Controls on the machine control panel

In this example, the MCP 483C IE machine control panel is used to illustrate the operator controls and displays of a Siemens machine control panel.

Overview

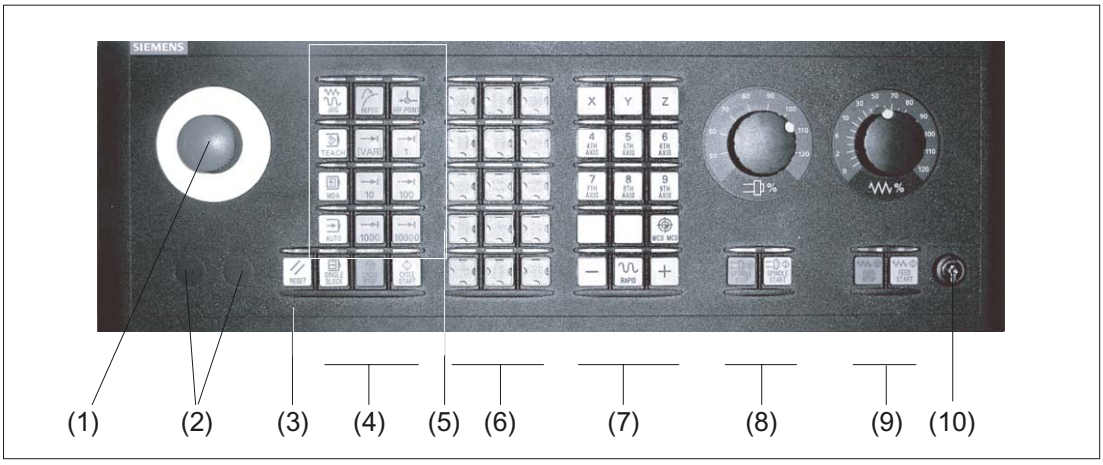
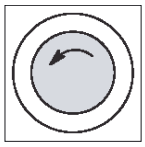


Figure 1-2 Front view of machine control panel (milling version)

(1) 

EMERGENCY STOP button

Activate the button in situations where

- life is at risk.
- there is the danger of a machine or workpiece being damaged.

All drives will be stopped with the greatest possible braking torque.

Machine manufacturer

For additional responses to pressing the Emergency Stop button, please refer to the machine manufacturer's instructions.

(2) **Installation locations for control devices (d = 16 mm)**

(3) 

Reset button

- Stop processing the current programs.
  The NCK control remains synchronized with the machine. It is in its initial state and ready for a new program run.
- Cancel alarm.

(4) **Program control**

Single Block

Single block mode on/off.

Cycle Start

Starts execution of a program.

Cycle Stop

Stops execution of a program.

(5) **Operating modes, machine functions**

JOG

Select "JOG" mode.

Teach In

Select "Teach In" submode.

MDA

Select "MDA" mode.

AUTO

Select "AUTO" mode.

REPOS

Repositions, re-approaches the contour.

REF POINT

Approach reference point.

Inc VAR (incremental feed variable)

Incremental mode with variable increment size.

Inc (Incremental feed)

Incremental mode with predefined increment size of 1, ..., 10000 increments.

...

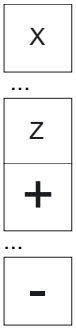
Machine manufacturer

A machine data code defines how the increment value is interpreted.

(6) **Customer keys**

T1 to T15

(7) **Traversal axes with rapid traverse superposition and coordinate exchange**



Axis keys

Selects an axis.

Arrow keys

Select the traversing direction.

Rapid

Traverse axis in rapid traverse while pressing the direction key.

WCS MCS

Switches between the workpiece coordinate system (WCS = work) and machine coordinate system (MCS = machine).

(8) **Spindle control with override switch**

Spindle Stop

Stop spindle.

Spindle Start

Spindle is enabled.

(9) **Feed control with override switch**

Feed Stop

Stops execution of the running program and shuts down axis drives.

Feed Start

Enable for program execution in the current block and enable for ramp-up to the feedrate value specified by the program.

(10) **Keyswitch (four positions)**

1.4 Operator interface

1.4.1 Operation via softkeys and buttons

Operating areas/operating modes

The operator interface consists of different windows featuring 8 horizontal and 8 vertical softkeys.

You operate the softkeys with the keys next to the softkey bars.

You can display a new window or execute functions using the softkeys.

HMI sl is divided into 6 operating areas (machine, parameter, program, program manager, diagnosis, startup) and 5 operating modes or submodes (JOG, MDA, AUTO, Teach In, Ref. Point, Repos).

Change operating area

Press the "Menu Select" key and select the desired operating area using the horizontal softkey bar.

You can call the "Machine" operating area directly using the key on the operator panel.

"Machine" operating area

Changing the Operating Mode

You can select a mode or submode directly using the keys on the machine control panel or using the vertical softkeys in the main menu.

General keys and softkeys

When the  symbol appears to the right of the dialog line on the operator interface, you can change the horizontal softkey bar within an operating area. This is done by pressing the "Expansion" key.

The  symbol indicates that you are in the expanded softkey bar.

Pressing the "ETC" key again will take you back to the original horizontal softkey bar.

Within an operating mode or operating area, you can use the "Back" softkey to return to the next highest window.

Use the "Cancel" softkey to exit a window without accepting the entered values and return to the next highest window.

When you have entered all the necessary parameters in the parameter screen form correctly, you can close the window and save the parameters using the "Accept" softkey. The values you entered are applied to a program.

Use the "OK" softkey to initiate an action immediately, e.g., to rename or delete a program.

See also

Controls on the machine control panel (Page 1-5)

Overview

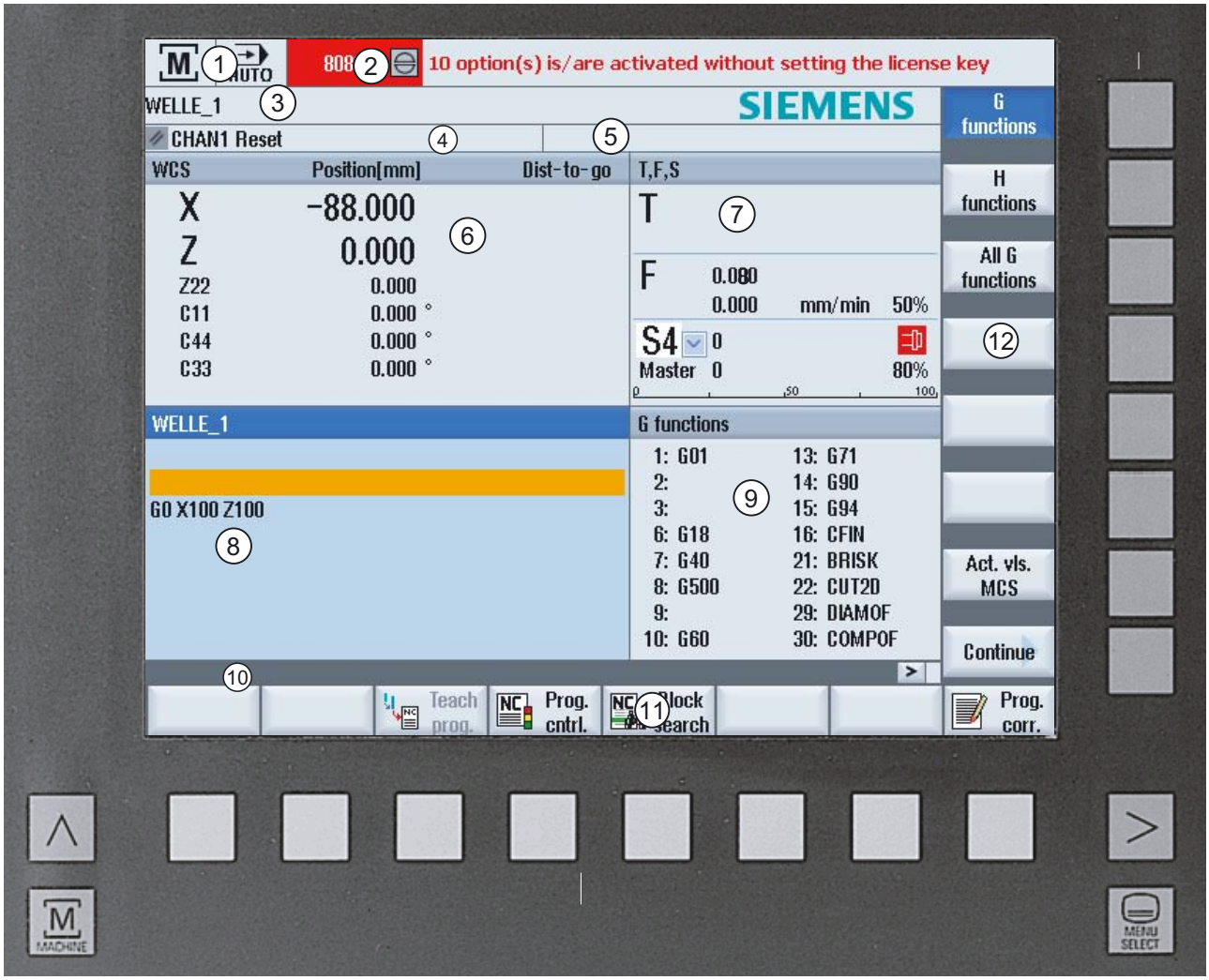


Figure 1-3 Operator interface

1 Active operating area and mode
2 Alarm/message line
3 Program name
4 Channel state and program control
5 Channel operational messages
6 Axis position display in actual value window
7 Display for
- active tool T
- current feedrate F
- current spindle condition

8 Operating window with program block display
9 Display of G functions, H functions, and input windows for different functions (e.g., program control).

10 Dialog line to provide additional user notes
11 Horizontal softkey bar
12 Vertical softkey bar

1.4.2 Status display

The status display includes the most important information about the current machine status and the status of the NCK. It also shows alarms as well as NC and PLC messages.

Depending on your operating area, the status display is made up of several lines:

- Large status display

  The status display is made up of three lines in the "Machine" operating area.

- Small status display

  In the "Parameter", "Program", "Program manager", "Diagnosis" and "Start-up" operating areas, the status display consists of the first line from the large display.

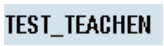
Status display of "Machine" operating area

First line

Display	Description
Active operating area	
	"Machine" operating area
	Operating area "Parameter"
	"Program" operating area
	"Program manager" operating area
	"Diagnosis" operating area
	"Start-up" operating area

Display	Description
Active mode or submode	
	"JOG" mode
	"MDA" mode
	"AUTO" mode
	"TEACH In" submode
	"REPOS" submode
	"REF POINT" submode
Alarms and messages	
	Alarm display The alarm numbers are displayed in white lettering on a red background. The associated alarm text is shown in red lettering. An arrow indicates that several alarms are active. An acknowledgment symbol indicates that the alarm can be acknowledged or canceled.
	NC or PLC message Message numbers and texts are shown in black lettering. An arrow indicates that several messages are active.
	Messages from NC programs do not have numbers and appear in green lettering.

Second line

Display	Description
	Program path and program name

Third line

Display	Description
CHAN1 RESET	Channel display. Here as an example: several channels are available; refer to the machine manufacturer's instructions. If only one channel is available, only the "Reset" channel status is displayed. With touch operation, you can change the channel here.
  	Display of channel status: The program was aborted with "Reset". The program is being executed. The program has been interrupted with "Stop".
DRYPRT	Display of active program controls: PRT: no axis motion DRY: Dry run feedrate M01: programmed stop 1 M101: programmed stop 2 (name varies) SBL1: Single block, coarse (program stops only after blocks which perform a machine function) SBL2: Data block (program stops after each block) SBL3: Single block, fine (program also only stops after blocks which perform a machine function in cycles)
 	Channel operational messages: Stop: An operator action is required. Wait: No operator action is required.

The machine manufacturer settings determine which program controls are displayed.

Machine manufacturer
Please also refer to the machine manufacturer's instructions.

See also

Channel switchover (Page 2-4)

1.4.3 Actual value window

The actual values of the axes and their positions are displayed.

WCS/MCS

The displayed coordinates are based on either the machine coordinate system or the workpiece coordinate system. The machine coordinate system (MCS), in contrast to the workpiece coordinate system (WCS), does not take any zero-point offsets into consideration.

Toggle the display between the machine coordinate system and the workpiece coordinate system via the special "MCS/WCS" key on the MCP or the "Act. value MCS" softkey.

Maximize display

You can maximize the actual value window. You can gather additional information there about feed and override.

Press the "Continue" and "Actual value zoom" softkeys.

Overview of display

Display	Description
Header columns	
WCS/MCS	Display of axes in selected coordinate system
Item	Position of displayed axes.
Display of distance-to-go	The distance-to-go for the current NC block is displayed while the program is running.
Feed/override	The feedrate acting on the axes is displayed along with the override. You only receive this information in the full screen.
Repos offset	The distances traversed in manual mode are displayed. This information is only displayed when you are in the "Repos" submode.

1.4.4 T,F,S window

The most important data concerning the current tool, the feedrate (path feed or axis feed in JOG) and the spindle are displayed in the T, F, and S windows.

Tool data

Display	Description
T	
Tool name	Name of current tool
Location	Location number of current tool
D	Cutting edge of the current tool
Ø	Diameter of current tool

Feed data

Display	Description
F	
	Feed disable/rapid traverse reduction
	Actual feed value If several axes are traversing, the largest axis feed will be displayed
Rapid traverse	G0 is active
0,000	No feed is active
Override	Display as a percentage

Spindle data

Display	Description
S	
S1	Spindle selection, identification with spindle number and main spindle
speed	Actual value (when spindle turns, display increases) Setpoint (always displayed, also during positioning)
Icon	Spindle condition (spindle disable, spindle stop, clockwise rotation, counter-clockwise rotation, positioned, axis mode
Override	Display as a percentage
Spindle utilization rate	Display between 0 and 100% The upper limit can also display 200% See machine manufacturer's specifications.

1.4.5 Program block display

The window of the current block display shows you the program blocks currently being executed.

Display of current program

The following information is displayed in the running program:

- The workpiece name or program name is entered in the title row.
- Three blocks of the current program are displayed.
- The program block which is just being processed appears colored.

See also

Correcting programs (Page 3-6)

1.4.6 Protection levels

The input and modification of vital data in the control system is protected by passwords.

Access protection via protection levels

The input or modification of data for the following functions depends on the protection level setting:

- Tool offsets
- Work offsets
- Setting data
- Program creation / program editing

For additional information, please refer to the following documentation:

HMI sl / SINUMERIK 840D sl start-up guide

Softkeys

Softkey	Protection levels
Program manager	
System data	3
Start-up	
MD Mach. data	7
General MD ... Control unit MD	4
SD Setting data	4
Set MD to active	4
NCK	2
Kennwort ändern	3
Kennwort löschen	3

1.4.7 Entering parameters

On setting up the machine and during programming, you must enter values in the white fields for various parameters. Parameters that have a gray input field are automatically calculated.

Selecting parameters

Some parameters require you to select from a number of options in the input field. Fields of this type do not allow you to type in a value.

Selecting the unit

For certain parameters, you can choose from among different units.

Proceed as follows

1. Keep pressing the "Select" key until your preferred setting or unit is selected.

   The "Select" key only works if there are several selection options available.
2. Enter a numerical value.
3. Press the "Input" key to finish the parameter input.

Changing or calculating parameters

If you only want to change individual characters in an input field rather than overwriting the entire entry, switch to insertion mode. In this mode, the pocket calculator is also active. You can use it during programming to calculate parameter values.

Press the "Insert" key.

Insertion mode and the pocket calculator are activated.


...

You can navigate within the input field using the "Left cursor" and "Right cursor" keys.

...

Use the "BACKSPACE" or "DEL" key to delete individual characters.

Accepting parameters

When you have correctly entered all necessary parameters, you can close the window and save your settings.

You cannot accept the parameters if they are incomplete or obviously erroneous. In this case, you can see from the dialog line which parameters are missing or were entered incorrectly.

Press the "OK" softkey.

- OR -

Press the "Accept" soft key.

- OR -

Press the "Left cursor" key.

1.4.8 Pocket calculator

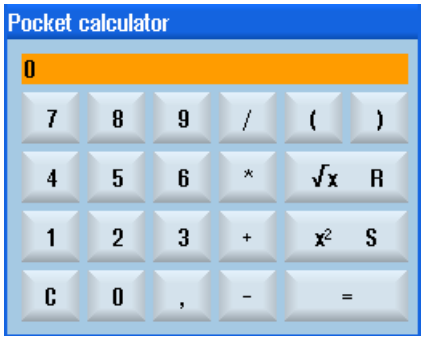
You can use the pocket calculator to quickly calculate parameter values during programming. If, for example, the diameter of a workpiece is only dimensioned indirectly in the workpiece drawing, i.e., the diameter must be derived from the sum of several other dimension specifications, you can calculate the diameter directly in the input field of this parameter.

Calculation methods

The following arithmetic operations are available:

- Addition
- Subtraction
- Multiplication
- Division
- Calculation with parentheses
- Square root of x
- x squared

You can input a maximum of 256 characters in a field.



Proceed as follows

1. Position the cursor on the desired input field.
2. Press the equals sign.
   The pocket calculator is displayed.
3. Input the arithmetic statement.
   You can use arithmetic symbols, numbers, and commas.
4. Press the "=" softkey.

- OR -

Press the "Calculate" softkey.

- OR -

Press the "Input" key.

The new value is calculated and displayed in the input field of the pocket calculator.

5. Press the "Accept" soft key.

The calculated value is accepted and displayed in the input field of the window.

Note

Input order for functions

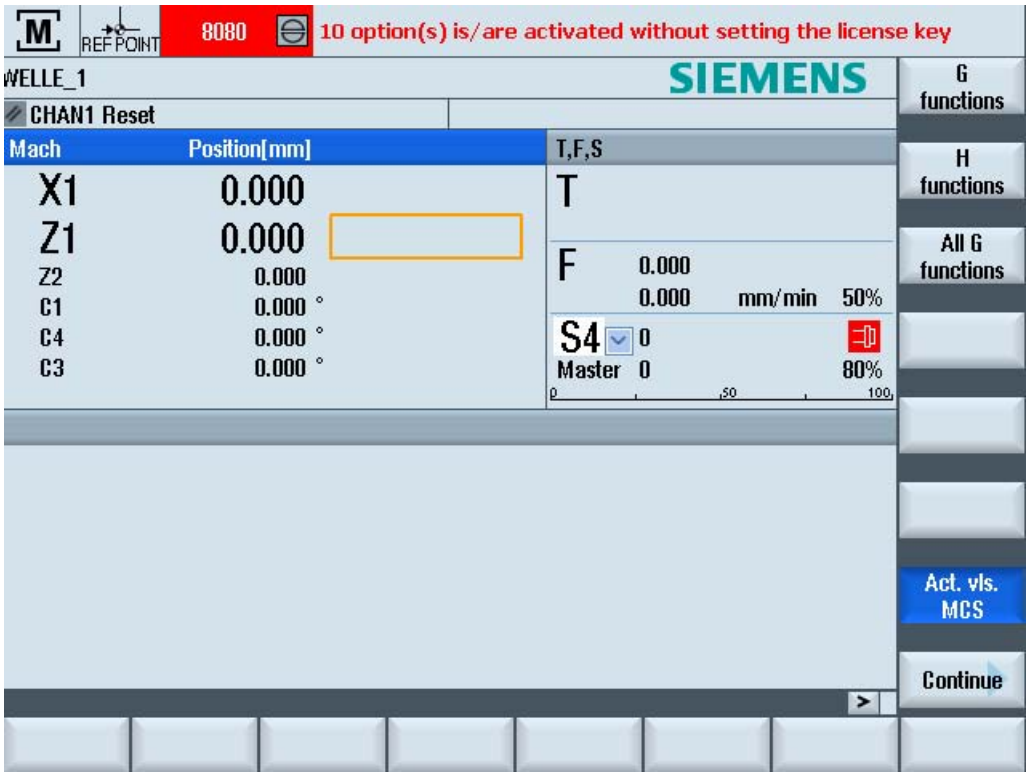
When using the square root or squaring functions, make sure to press the "R" or "S" function keys, respectively, before entering a number.

Setting up the machine

2

2.1 Switching on and switching off

Start-up



When the control starts up, the main screen opens according to the operating mode specified by the machine manufacturer. In general, this is the main screen for the "Ref Point" submode.

Machine manufacturer

Please also refer to the machine manufacturer's instructions.

2.2 Modes of operation

2.2.1 General information

You can work in three different operating modes.

"JOG" mode

"JOG" mode is used for the following preparatory actions:

- Reference point approach, i.e. calibration of the position measuring system
- Preparing a machine for executing a program in automatic mode, i.e. measuring tools, measuring the workpiece and, if necessary, defining the work offsets used in the program
- Traversing axes, e.g. during a program interruption
- Positioning axes

Select "JOG"

Press the "JOG" key.

"Ref Point" submode

The "Ref Point" submode is used to synchronize the control and the machine. For this purpose, you approach the reference point in "JOG" mode.

Select "Ref Point"

Press the "Ref Point" key.

"Repos" submode

The "Repos" submode is used for repositioning to a defined position. After a program interruption (e.g. to correct tool wear values) move the tool away from the contour in "JOG" mode.

The distances traversed in "JOG" mode are displayed in the actual value window as the "Repos" offset.

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

Select "Repos"

Press the "Repos" key.

"MDA" mode (Manual Data Automatic)

In "MDA" mode, you can enter and execute G code commands non-modally to set up the machine or to perform a single action.

Select "MDA"

Press the "MDA" key.

"AUTO" mode

In automatic mode, you can execute a program completely or only partially.

Select "AUTO"

Press the "AUTO" key.

Submode "Teach In"

The "Teach In" submode is available in the "AUTO" and "MDA" modes.

There you may create, edit and execute part programs (main programs or subroutines) for motional sequences or simple workpieces by approaching and saving positions.

Select "Teach In"

Press the "Teach In" key.

2.2.2 Channel switchover

It is possible to switch between channels when several are in use. Since individual channels may be assigned to different mode groups, a channel switchover command is also an implicit mode switchover command.

Channel switchover

Press the "CHANNEL" key.

The channel changes over to the next channel.

Channel switchover via touch operation on hand-held terminal 8

On the HT8 you can switch to the next channel via touch operation in the status display.

See also

Operation (Page 9-1)

2.3 Approaching a reference point

2.3.1 Reference axes

Your machine tool can be equipped with an absolute or incremental path measuring system. An incremental path measuring system must be calibrated after being switched on, but an absolute path measuring system does not.

For the incremental path measuring system, all the machine axes must therefore first approach a reference point, the coordinates of which are known to be relative to the machine zero-point.

Sequence

Prior to the approach, the axes must be in a position from where they can approach the reference point without a collision.

The sequence or direction is defined by the PLC program.

The axes can also all approach the reference point simultaneously, depending on the manufacturer's settings.

Machine manufacturer

Please also refer to the machine manufacturer's instructions.

Notice

If the axes are not in a collision-free position, you must first traverse them to safe positions in "JOG" or "MDA" mode.

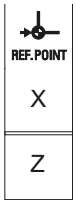
You must follow the axis motions directly on the machine!

Ignore the actual value display until the axes have been referenced!

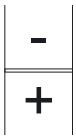
The software limit switches are not active!

Proceed as follows

1. Press the "JOG" key.



2. Press the "Ref. Point" key.
3. Select the axis to be traversed.



4. Press the "+" or "-" key.
   The selected axis moves to the reference point.

   If you have pressed the wrong direction key, the action is not accepted and the axes do not move.

A symbol is shown next to the axis if it has been referenced.

The axis is referenced as soon as the reference point is reached. The actual value display is set to the reference point value.

From now on, path limits, such as software limit switches, are active.

End the function via the machine control panel by selecting operating mode "AUTO" or "JOG".

2.3.2 User agreement

If you are using Safety Integrated (SI) on your machine, you will need to confirm that the current displayed position of an axis corresponds to its actual position on the machine when you reference an axis. Your confirmation is the precondition for the availability of other Safety Integrated functions.

You can only give your user agreement for an axis after it has approached the reference point.

The displayed axis position always refers to the machine coordinate system (MCS = machine).

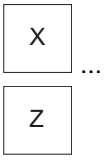
Option

User agreement with Safety Integrated is only possible with a software option.

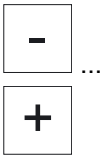
Proceed as follows

1. Select the "Machine" operating area.

2. Press the "Ref Point" softkey.



3. Select the axis to be traversed.



4. Press the "+" or "-" key.

   The selected axis moves to the reference point and stops. The coordinate of the reference point is displayed.

   The axis is marked with ⊕.

5. Press the "User agreement" softkey.

   The "User agreement" window opens.

   It shows a list of all machine axes with their current position and SI position.

5. Position the cursor in the "Agreement" field for the axis in question.

6. Activate the agreement with the "Select" button.

   The selected axis is marked with a cross meaning "safely referenced" in the "Agreement" column.

   By pressing the "Select" button, you deactivate the agreement again.

2.4 Settings for the machine

2.4.1 Coordinate system (MCS/WCS)

The coordinates in the actual value display are relative to either the machine coordinate system or the workpiece coordinate system.

By default, the workpiece coordinate system is set as a reference for the actual value display.

The machine coordinate system (MCS), in contrast to the workpiece coordinate system (WCS), does not take any zero-point offsets into consideration.

Proceed as follows

1. Select the "Machine" operating area.

...

2. Press the "JOG" or AUTO" key.

3. Press the "Act.vls. MCS" softkey.

The machine coordinate system is selected.

The title of the actual value window changes in the MCS.

2.5 Work offsets

2.5.1 Overview

Following reference point approach, the actual value display for the axis coordinates is based on the machine zero of the machine coordinate system (MCS).

The program for machining the workpiece, however, is based on the workpiece zero of the workpiece coordinate system (WCS). The machine zero and workpiece zero are not necessarily identical. The distance between the machine zero and workpiece vary in accordance with the type of tool and how it is clamped. This zero offset is taken into account during execution of the program and can be a combination of different offsets.

When the machine zero is not identical to the workpiece zero, at least one offset (base offset or work offset) exists in which the position of the workpiece zero is saved.

Base offset

The base offset is a work offset that is always active. If you have not defined a base offset, its value will be zero. The base offset is specified in the "Zero offset - Base" window.

Coarse and fine offsets

Every zero offset (G54 to G57, G505 to G599) consists of a coarse offset and a fine offset. You can call the zero offsets from any program (coarse and fine offsets are added together).

You can save the workpiece zero, for example, in the coarse offset, and then store the offset that occurs when a new workpiece is clamped between the old and the new workpiece zero in the fine offset.

2.5.2 Display active zero offset

The current active offsets and all active system offsets are displayed for all set-up axes in the "Zero offset - Active" window.

This window is generally used only for monitoring.

Display of active zero offsets

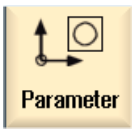
Work offsets	
Set zero offset	Display of active system offsets. You can alter the data under certain circumstances, i.e. you can correct a zero point. Access to the system offsets is protected via a keyswitch.
Entire Base ZO	Display of all active base offsets, as well as rotation, scaling and mirroring if these are set. You cannot edit these values here.
G54 ...G599	Display of all zero offsets activated with G54 - G599, as well as rotation, scaling and mirroring if these are set. You cannot edit these values here.
Prog. ZO	Display of all additional zero offsets programmed with $P_PFRAME=. as well as rotation, scaling and mirroring if these are set.
Frame cycles	Display of all additional zero offsets activated via $MC_MM_SYSTEM_FRAME_MASK.
Entire ZO	Display of the active zero offset, representing the sum of all zero offsets, as well as rotation, scaling and mirroring if these are set.

The availability of the offsets depends on the setting.

Machine manufacturer

Please also refer to the machine manufacturer's instructions.

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "Work offset" softkey.
   The "Work offset - Active" window is opened.

Note

Further details on zero offsets

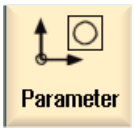
If you would like to see further details about the specified offsets or if you would like to change values for the rotation, scaling or mirroring, press the "Details" softkey.

2.5.3 Displaying and editing base zero offset

The defined channel-specific and global base offsets, divided into coarse and fine offsets, are displayed for all set-up axes in the "Work offset - Base" window.

The rotation, scaling and mirroring are displayed if they are set.

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "Work offset" softkey.

3. Press the "Base" softkey.
   The "Zero offset - Base" window is opened.
4. You can edit the values directly in the table.

Note

Activate base offsets

The offsets specified here are immediately active.

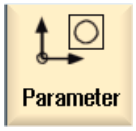
2.5.4 Displaying and editing settable zero offset

All settable offsets, divided into coarse and fine offsets, are displayed in the "Zero offset - G54...G599" window.

The current active zero offsets are displayed on a green background.

The rotation, scaling and mirroring are displayed if they are set.

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "Work offset" softkey.

3. Press the "G54…G599" soft key.
   The "Zero offset - G54...G599" window is opened.
4. You can edit the values directly in the table.

Note

Activate settable zero offsets

The settable zero offsets must first be selected in the program before they have an impact.

2.5.5 Displaying and editing details of the zero offsets

For each zero offset, you can display and edit all data for all axes. You can also delete zero offsets.

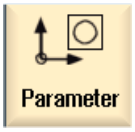
For every axis, values for the following data will be displayed:

- Coarse and fine offsets
- Rotation
- Scaling
- Mirroring

Note

Settings for rotation, scaling and mirroring are specified here and can only be changed here.

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "Work offset" softkey.

3. Press the "Active", "Base" or "G54…G599" softkey.
   The corresponding window appears.

4. Place the cursor on the desired zero offset to view its details.

5. Press the "Details" softkey.

   A window opens, depending on the selected zero offset, e.g. "Zero offset - Details: G54 to G599".

6. You can edit the values directly in the table.

   - OR -

   Press the "Clear Offset" softkey to reset all entered values.

These value changes are available in the part program immediately or after "Reset".

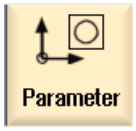
Machine manufacturer
Please also refer to the machine manufacturer's instructions.

You can return to the previous window with the "Back" softkey.

2.5.6 Deleting zero offsets

You have the option of deleting zero offsets. This resets the entered values.

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "Work offset" softkey.

...

3. Press the "Active", "Base" or "G54…G599" softkey.

4. Press the "Details" softkey.
5. Position the cursor on the zero offset you would like to delete.

6. Press the "Clear Offset" soft key.

2.6 Traversing axes

You can traverse the axes in manual mode via the Increment or Axis keys or handwheels.

During a traverse initiated from the keyboard, the selected axis moves at the programmed setup feedrate. During an incremental traverse, the selected axis traverses a specified increment.

Set the default feedrate

Specify the feedrate to be used for axis traversal in the set-up, in the "Settings for manual operation" window.

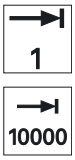
2.6.1 Traverse axes by a defined increment

You can traverse the axes in manual mode via the Increment and Axis keys or handwheels.

Proceed as follows

1. Select the "Machine" operating area.

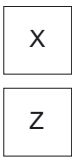
2. Press the "JOG" key.



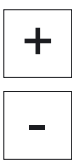
3. Press keys 1, 10, etc. up to 10000 in order to move the axis in a defined increment.

   The numbers on the keys indicate the traverse path in micrometers or micro-inches.

   Example Press the "100" button for a desired increment of 100 µm (= 0.1 mm).



4. Select the axis to be traversed.



5. Press the "+" or "-" key.

   Each time you press the key the selected axis is traversed by the defined increment.

   Feedrate and rapid traverse override switches can be operative.

Note

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

The software limit switches and the working area limitation are not yet operative!

The feed enable signal must be set.

Machine manufacturer

Please also refer to the machine manufacturer's instructions.

2.6.2 Traversing axes by a variable increment

Proceed as follows

1. Select the "Machine" operating area.

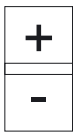
Press the "JOG" key.

2. Press the "Settings" softkey.

   The "Settings for manual operation" window is opened.
3. Enter the desired value for the "Variable increment" parameter.

   Example Enter 500 for a desired increment of 500 µm (0.5 mm).

4. Press the "Inc Var" key.
5. Select the axis to be traversed.



6. Press the "+" or "-" key.

   Each time you press the key the selected axis is traversed by the set increment.

   Feedrate and rapid traverse override switches can be operative.

2.7 Handwheel

2.7.1 Handwheel assignment

You can traverse the axes in the machine coordinate system (MCS) or in the workpiece coordinate system (WCS) via the handwheel.

All axes are provided in the following order for handwheel assignment:

- Geometry axes
- Channel machine axes

Machine manufacturer
Please also refer to the machine manufacturer's instructions.

Proceed as follows

1. Select "Machine" in the operating area.

...

Press the "JOG", "AUTO", or "MDA" key.

...

2. Select the "Continue" and "Handwheel" softkeys.
   The "Handwheel" window appears.
   A field for axis assignment will be offered for every connected handwheel.

3. Place the cursor in the field next to the handwheel with which you wish to assign the axis (e.g. no. 1).

4. Press the corresponding softkey to select the desired axis (e.g. "X").

- OR

...


To open the "Axis" selection box using the "Insert" key, navigate to the desired axis, and press the "Input" key.

Handwheel no. 1 is immediately activated for the X-axis.

Deactivate handwheel

1. Place the cursor on the handwheel whose assignment you wish to cancel (e.g., no. 1).
2. Press the softkey for the assigned axis again (e.g. "X").

- OR -

...


Open the "Axis" selection box using the "Insert" key, navigate to the empty field, and press the "Input" key.

The X-axis is deselected and deactivated for handwheel no. 1.

2.7.2 Default settings for manual mode

Specify the configurations for manual mode in the "Settings for manual operation" window.

Presettings

Settings	Description
Type of feedrate	Here, you select the type of feedrate.
	• G94: Axis feedrate/linear feedrate • G95: Rev. feedrate
Default feedrate G94	Enter the desired feedrate in mm/min.
Default feedrate G95	Enter the desired feedrate in mm/r.
Variable increment	Enter the desired increment for axis traversal by variable increments.
Spindle speed	Enter the desired spindle speed in rpm.

Proceed as follows

1. Select "Machine" in the operating area.

2. Press the "JOG" key.

3. Press the "Settings" softkey on the expanded softkey bar.

   The "Settings for manual operation" window is opened.

2.8 MDA

In "MDA" mode (Manual Data Automatic mode), you can enter G-code commands block-by-block and immediately execute them for setting up the machine.

You can load an MDA program straight from the Program Manager into the MDA buffer. You may also store programs which were rendered or changed in the MDA operating window into any directory of the Program Manager.

2.8.1 Loading an MDA program from the Program Manager

Proceed as follows

1. Select the "Machine" operating area.

2. Press the "MDA" key.

   The MDI editor opens.

3. Press the "Load save" softkey.

   A changeover is made into the Program Manager.

4. Select the program that you would like to edit or execute in the MDA window.

5. Press the "Copy into MDA" softkey.

   A changeover occurs back to the MDA window, and the program is ready for editing.

2.8.2 Saving an MDA program

Proceed as follows

1. Select the "Machine" operating area.

2. Press the "MDA" key.

   The MDA editor opens.

3. Create the MDA program by entering the G-code commands using the operator's keyboard.

4. Press the "Load save" softkey.

   A changeover is made into the Program Manager.

5. Select the drive on which you want to save the MDA program you created, and select the directory in which the program is to be saved.

6. Press the "Save from MDA" softkey.

   When you place the cursor on a folder, a window opens which prompts you to assign a name.

   - OR -

   When you place the cursor on a program, you are asked whether the file should be overwritten.

7. Enter the name for the rendered program and press the "OK" softkey.

   The program will be saved under the specified name in the selected directory.

2.8.3 Executing an MDA program

Proceed as follows

1. Select the "Machine" operating area.

2. Press the "MDA" key.

   The MDA editor opens.
3. Input the desired G-code commands using the operator's keyboard.

4. Press the "Cycle Start" key.

   The control executes the input blocks.

When executing the G-code commands, you can control the sequence as follows:

- Executing the program block-by-block
- ProgramTesting

  Settings under program control
- Setting the test-run feedrate

  Settings under program control

See also

Program control (Page 3-13)

2.8.4 Deleting an MDA program

You can delete the program created in MDA mode.

Proceed as follows

1. Select the "Machine" operating area.

2. Press the "MDA" key.

   The MDI editor opens.
3. Input the desired G-code commands using the operator's keyboard.

4. Press the "Delete MDI buffer" soft key.

   The program displayed in the program window is deleted.

Machine the workpiece

3

3.1 Starting program execution

3.1.1 Starting and stopping machining

During execution of a program, the workpiece is machined in accordance with the programming on the machine. After the program is started in automatic mode, workpiece machining is performed automatically.

Prerequisites

The following prerequisites must be met before executing a program:

- The measuring system of the control is referenced with the machine.
- The necessary tool offsets and zero offsets have been entered.
- The necessary safety interlocks implemented by the machine manufacturer are activated.

General sequence

1. Use the Program Manager to select the desired program.

Select the desired program under "NC". If necessary, it must first be copied to "NC".

3. Press the "Execute" softkey.
   The program is selected for execution and automatically switched to the "Machine" operating area.

4. Press the "Cycle Start" key.
   The program is started and executed.

Note

Start program in any operating area

If the control is in "AUTO" mode, you can also start the selected program when you are in any operating area.

Stopping the program

Press the "Cycle Stop" key.
The machining stops immediately. Individual program blocks are not executed to the end. When it is started again, it will continue execution from the point where it left off

Cancel execution

Press the "Reset" key.
Execution of the program is interrupted. When it is started again, it will execute from the beginning.

Machine manufacturer

Please also refer to the machine manufacturer's instructions.

3.1.2 Selecting a program

Proceed as follows

1. Select the "Program manager" operating area.

   The directory overview is opened.
2. Place the cursor on the directory containing the program that you want to select.

3. Press the "Input" key

   - OR -

   Press the "Cursor Right" key.

   The directory contents are displayed.
4. Place the cursor on the desired program.

5. Press the "Execute" softkey.

   The program is selected.

   When the program has been successfully selected, an automatic changeover to the "Machine" operating area occurs.

3.2 Program running-in

3.2.1 Executing single blocks

When running-in a program, the system can interrupt the machining of the workpiece after each program block, which triggers a movement or auxiliary function on the machine. In this way, you can control the machining result block-by-block during the initial run-through of a program on the machine.

Move by single block

In "Program control" you may select from among several types of block processing:

SBL mode	Scope
SBL1 Single block, coarse	The machining stops after every machine block (except for cycles)
SBL2 Data block	The machining stops after every block, i.e. also for data blocks (except for cycles)
SBL3 Single block, fine	The machining stops after every machine block (also in cycles)

Requirement

A program must be selected for execution in "AUTO" or "MDA" mode.

Proceed as follows

1. Press the "Prog. cntrl." softkey and select the desired variant in the "SBL" field.

2. Press the "Single Block" key.

3. Press the "Cycle Start" key.

   Depending on the execution variant, the first block will be executed. Then the machining stops.

   In the channel status line, the text “Stop: Block in single block ended" appears.

4. Press the "Cycle Start" key.

   Depending on the mode, the program will continue executing until the next stop.

5. Press the "Single Block" key again, if the machining is not supposed to run block-by-block.

   The key is deselected again.

   If you now press the "Cycle Start" key again, the program is executed to the end without interruption.

See also

Selecting a program (Page 3-3)

3.3 Display current program block

3.3.1 Current block display

The window of the current block display shows you the program blocks currently being executed.

Display of current program

The following information is displayed in the running program:

- The workpiece name or program name is entered in the title row.
- Three blocks of the current program are displayed.
- The program block which is just being processed appears colored.

3.3.2 Display program level

You can display the current program level during the execution of a large program with several subroutines.

Display program level

The following information will be displayed:

- Level number
- Program name
- Block number, or line number

Requirement

A program must be selected for execution in "AUTO" or "MDA" mode.

Proceed as follows

1. Select the expanded softkey area.

2. Press the "Program levels" soft key.
   The "Program levels" window appears.

3.4 Correcting programs

As soon as a syntax error in the part program is detected by the control, program execution is interrupted and the syntax error is displayed in the alarm line.

Correction possibilities

Depending on the state of the control system, you can make the following corrections using the Program editing function.

- Stop state

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

  All program lines can be edited.

Requirement

A program must be selected for execution in "AUTO" or "MDA" mode.

Proceed as follows

1. The program to be corrected is in the Stop or Reset state.
2. Press the "Prog. corr." softkey
3. Navigate to the block containing errors.


...

4. Correct the program block.

   - OR -

   Press the "Editor" softkey if you would like to make more extensive changes.

   The program is opened in the editor.

5. Press the "Cycle Start" key to resume program execution.

3.5 Repositioning axes

After a program interruption in automatic mode (e.g. after a tool breaks) you can move the tool away from the contour in manual mode.

The coordinates of the interrupt position will be saved. The distances traversed in manual mode are displayed in the actual value window. This path difference is called "Repos-offset".

Continue executing the program

Using the "Repos" function, you can return the tool to the contour in order to continue executing the program.

You cannot traverse the interrupt position, because it is blocked by the control system.

The feedrate/rapid traverse override is in effect

Warning

When repositioning, the axes move with the programmed feedrate and linear interpolation, i.e. in a straight line from the current position to the interrupt point. Therefore, you must first move the axes to a safe position in order to avoid collisions.

If you do not use the "Repos" function and subsequently move the axes in manual mode after a program interrupt, the control automatically moves the axes during the switch to automatic mode and the subsequent start of the machining process in a straight line back to the point of interruption.

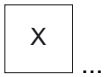
Requirement

The following prerequisites must be met when repositioning the axes:

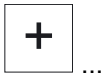
- The program execution was interrupted using "Cycle Stop".
- The axes were moved from the interrupt point to another position in manual mode.

Proceed as follows

1. Press the "Repos" key.



2. Select the axes to be traversed one after the other.



3. Press the "+" or "-" key for the relevant direction.
   The axes are moved to the interrupt position.

3.6 Starting execution at a specific point

3.6.1 Use block search

If you would only like to perform a certain section of a program on the machine, then you need not start the program from the beginning. You can also start the program from a specified program block.

Applications

- Stopping or interrupting a program
- Specify a target position, e.g. during remachining

Prerequisites

1. You have selected the desired program.
2. The control system is in the RESET condition.
3. The desired search mode is selected.

Notice

Collision-free start position

Pay attention to a collision-free start position and appropriate active tools and other technological values.

If necessary, manually approach a collision-free start position. Select the target block considering the selected block search type.

3.6.2 Approaching an interruption point again

Requirement

A program was selected in "AUTO" mode and interrupted during execution with "Reset".

Proceed as follows

1. Press the "Interrupt point" softkey.

   The interruption point is loaded.

2. Press the "Start search" softkey.

   The search is started. Your specified search mode will be taken into account.

   The search screen closes. The current block will be displayed in the program window as soon as the target is found.

3. Press the "Cycle Start" key twice.

   Processing will continue from the interruption point.

3.6.3 Entering a search target

Enter the program point which you would like to proceed to in the "Search indicator" window.

Requirement

The program is selected and the control is in Reset mode.

Screen form

Each line represents one program level. The actual number of levels in the program depends on the nesting depth of the program.

Level 1 always corresponds to the main program and all other levels correspond to subroutines.

You must enter the target in the line of the window corresponding to the program level in which the target is located.

For example, if the target is located in the subroutine called directly from the main program, you must enter the target in program level 2.

The specified target must always be unambiguous. This means, for example, that if the subroutine is called in the main program in two different places, you must also specify a target in program level 1 (main program).

Proceed as follows

1. Enter the complete target path in the input fields
2. 

Press the "Start search" softkey.

   The search is started. Your specified search mode will be taken into account.

   The search window closes. The current block will be displayed in the program window as soon as the target is found.
3. 

Press the "Cycle Start" key twice in order to continue processing from the required position.

3.6.4 Parameters for block search

Parameters in "Search indicator" window

parameters	Description
Number of program level	
Program:	The name of the main program is automatically entered.
Ext:	File extension
P:	Pass counter. If a program part is performed several times, you can enter the number of the pass here at which processing is to be continued.
line:	Is automatically filled for an interruption point
Type	" " search target is ignored on this level N no. block number Label jump label Text string Sub-r. Subroutine call Row row number
Search target	Point in the program at which machining is to start

3.6.5 Block search mode

Set the desired search variant in the "Search mode" window.

The set mode is retained when the the control is shut down. When you activate the "Search" function after restarting the control, the current search mode is displayed in the title row.

Search variants

Block search mode	Description
With calculation - with approach	It is used to be able to approach the contour in any circumstance. On "Cycle Start", the start position of the target block or the end position of the block before the target block is approached. The program runs in the same way as in normal program processing.
With calculation - without approach	It is used in order to be able to approach a target position in any circumstance (e.g. tool change position). The end position of the target block or the next programmed position is approached using the type of interpolation valid in the target block. Only the axes programmed in the target block are moved.
Without calculation	For a quick search in the main program. Calculations will not be performed during the block search, i.e. the calculation is skipped up to the target block. All settings required for execution have to be programmed from the target block (e.g. feedrate, spindle speed, etc.).

Proceed as follows

1. Select the "Machine" operating area.

2. Press the "AUTO" key.

3. Press the "Block search" and "Blk sear. mode" softkeys.
   The "Search mode" window will open.

...

3.7 Controlling the program run

3.7.1 Program control

You can change the program sequence in the "AUTO" and "MDA" modes.

Abbreviation/program control	Scope
PRT no axis motion	The program is started and executed with auxiliary function outputs and dwell times. In this mode, the axes are not traversed. The programmed axis positions and the auxiliary function outputs are controlled this way. Note: Program processing without axis motion can also be activated with the function "dry run feedrate".
DRY Dry run feedrate	The traversing velocities programmed in conjunction with G1, G2, G3, CIP and CT are replaced by a defined dry run feedrate. The dry run feedrate also applies instead of the programmed revolutional feedrate. Caution: Workpieces must not be machined when "Dry run feedrate" is active because the altered feedrates might cause the permissible tool cutting rates to be exceeded and the workpiece or machine tool could be damaged.
M01 Programmed stop 1	The processing of the program stops at every block in which supplementary function M01 is programmed. In this way you can check the already obtained result during the processing of a workpiece. Note: In order to continue executing the program, press the "Cycle Start" key again.
Programmed stop 2 (e.g. M101)	The processing of the program stops at every block in which the "Cycle end" is programmed (e.g. with M101). Note: In order to continue executing the program, press the "Cycle Start" key again. The display can be changed. Please also refer to the machine manufacturer's instructions.
DRF Handwheel offset	Enables an additional incremental zero offset while processing in automatic operation mode with an electronic handwheel. This function can be used to compensate for tool wear within a programmed block.
SBL	Individual blocks are configured as follows. Single block, coarse: The program stops only after blocks which perform a machine function. Data block: The program stops after each block. Single block, fine: The program also only stops after blocks which perform a machine function in cycles. Select the desired setting using the "Select" key.
SKP	Skip blocks are skipped during machining.

Activate program control

You can control the program sequence however you wish by selecting and clearing the relevant check boxes.

Display / response of active program controls:

If a program control is activated, the abbreviation of the corresponding function appears in the status display as response.

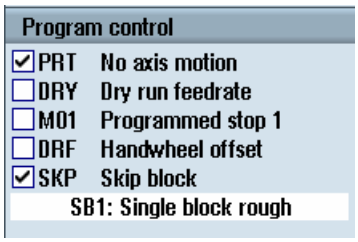
Proceed as follows

1. Select the "Machine" operating area.

...

2. Press the "AUTO" or "MDA" key.

3. Press the "Prog. cntrl." softkey.
   The "Program control" window appears.



3.7.2 Skip blocks

It is possible to skip program blocks, which are not to be executed every time the program runs.

The skip blocks are identified by placing a "/" (forward slash) or "/x (x = number of skip level) character in front of the block number. Several consecutive blocks can also be skipped.

The statements in the skipped blocks are not executed, i.e. the program continues with the next block, which is not skipped.

The number of skip levels that can be used depends on a machine datum.

Machine manufacturer
Please also refer to the machine manufacturer's instructions.

Skip levels, activate

Check the corresponding checkbox in order to skip the required block level.

Note

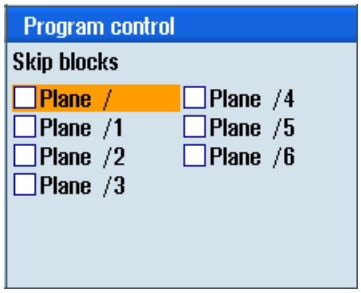
The "Program control - skip blocks" window is only available when more than one skip level is set up.

Proceed as follows

1. Select the "Machine" operating area.

2. Press the "AUTO" or "MDA" key.

3. Press the "Prog. cntrl." and "Skip blocks" softkeys.
   The "Program control" window appears and shows a list of block levels.



3.8 Editing programs

3.8.1 Overview

With the editor, you are able to render, supplement, or change part programs.

Note

The maximum block length is 512 characters.

Calling the editor

- The editor is started via the "Program correction" function in the "Machine" operating area.
- The editor is started via the "Open" softkey or the "Input" key in the "Program manager" operating area.
- The editor opens with the most recently edited part program in the "Program" operating area.

Note

Please note that the changes to programs stored in the NC memory take immediate effect. You can exit the editor only after you have saved the changes. This applies for all local drives that allow editing.

3.8.2 Searching in programs

You can use the search function to quickly arrive at points where you would like to make changes, e.g. in very large programs.

Requirement

The desired program is opened in the editor.

Proceed as follows

1. Press the "Search" softkey.
   A new vertical softkey bar appears.
   The "Search" window opens at the same time.
2. Enter the desired search term in the "Text" field.

3. Place the cursor in the "Direction" field and choose the search direction (forward, backward) with "Select".

4. Press the "OK" softkey to start the search.
   If the text you are searching for is found, the corresponding line is highlighted.

5. Press the "Continue search" softkey if the text located during the search does not correspond to the point you are looking for.
   - OR -

   Press the "Cancel" softkey when you want to cancel the search.

Further search options

Softkey	Function
	The cursor is set to the first character in the program.
	The cursor is set to the last character in the program.

3.8.3 Replacing program text

You can find and replace text in one step.

Requirement

The desired program is opened in the editor.

Proceed as follows

1. Press the "Search" soft key
   A new vertical softkey bar appears.

2. Press the "Find + replace" softkey.
   The "Find and replace" window appears.
3. In the "Text" field, enter the term you are looking for and in the "Replace with" field, enter the text you would like to insert automatically during the search.

4. Place the cursor in the "Direction" field and choose the search direction (forward, backward) with "Select".

5. Press the "OK" softkey to start the search.
   If the text you are searching for is found, the corresponding line is highlighted.

6. Press the "Replace" softkey to replace the text.

- OR -

Press the "Replace all" softkey to replace all text in the file that corresponds to the search term.

- OR -

Press the "Continue search" softkey if the text located during the search should not be replaced.

- OR -

Press the "Cancel" softkey when you want to cancel the search.

3.8.4 Copying/inserting/deleting program blocks

Requirement

The program is opened in the editor.

Proceed as follows

1. Press the "Mark" softkey.
2. Select the desired program blocks with the cursor or mouse.

3. Press the "Copy" softkey in order to copy the selection to the buffer memory.

4. Place the cursor on the desired insertion point in the program and press the "Paste" softkey.
   The content of the buffer memory is inserted.

Deleting program blocks

Use the "Cut" softkey to delete selected program blocks.

Note

The buffer memory contents are retained even after the editor is closed, enabling you to insert the contents in another program, as well.

3.8.5 Renumbering programs

You can modify the block numbering of programs opened in the editor at a later point in time.

Requirement

The program is opened in the editor.

Proceed as follows

1. Press the "Continue" softkey.
   A new vertical softkey bar appears.

2. Press the "Renumbering" softkey.
   The "Renumbering" window appears.
3. Enter the values for the first block number and the increment to be used for numbering.

4. Press the "OK" softkey.
   The program is renumbered.

Note

If you only want to renumber a section, select the program blocks whose block numbering you want to edit.

3.8.6 Editor settings

In the "Settings" window enter the default settings for newly rendered programs in the editor.

Presettings

Settings	Description
Number automatically	Yes: A new block number will automatically be assigned after every line change. In this case, the specifications provided under "First block number" and "Increment" are applicable. No: No automatic numbering
First block number	Specifies the starting block number of a newly created program. The field is only editable when "Yes" is available under "Number automatically".
Increment	Defines the increment used for the block numbers. The field is only editable when "Yes" is available under "Number automatically".
Display hidden lines	Hidden lines marked with "*HD" (hidden) will be displayed.
Display block end as an icon	The "LF" (line feed) symbol will be displayed at the block end.
Scroll horizontally	A horizontal scrollbar is displayed. In this way, you can scroll horizontally to the end of long lines that would otherwise wrap.

All entries that you make here are effective immediately.

Proceed as follows

1. Select the "Program" operating area

You have activated the editor.

...

2. Select the "Continue" and "Settings" softkeys.
   The "Settings" window appears.

3. Make the desired changes here and press the "OK" softkey to confirm your settings.

3.9 Display G and auxiliary functions

3.9.1 Selected G functions

16 selected G groups are displayed in the "G function" window.

Within a G group, the G function currently active in the control is displayed.

Some G codes (e.g. G17, G18, G19) are immediately active after switching the machine control on.

Which G codes are always active depends on the settings.

Machine manufacturer
Please also refer to the machine manufacturer's instructions.

Displayable G groups

Group	Description
G group 1	Modally active motion commands (e.g. G0, G1, G2, G3)
G group 2	Non-modally active motion commands, dwell time (e.g. G4, G74, G75)
G group 3	Programmable offsets, working area limitations and pole programming (e.g. TRANS, ROT, G25, G110)
G group 6	Level selection (e.g. G17, G18)
G group 7	Tool radius compensation (e.g. G40, G42)
G group 8	Settable zero offsets (e.g. G54, G57, G500)
G group 9	Offset suppression (e.g. SUPA, G53)
G group 10	Exact stop - continuous-path mode (e.g. G60, G641)
G group 13	Workpiece dimensioning inches/metric (e.g. G70, G700)
G group 14	Workpiece dimensioning absolute/incremental (G90)
G group 15	Feedrate type (e.g. G93, G961, G972)
G group 16	Feedrate override on inside and outside curvature (e.g. CFC)
G group 21	Acceleration profile (e.g. SOFT, DRIVE)
G group 22	Tool offset types (e.g. CUT2D, CUT2DF)
G group 29	Radius/diameter programming (e.g. DIAMOF, DIAMCYCOF)
G group 30	Compressor ON/OFF (e.g. COMPOF)

Active transformations are displayed in the header.

Display	Description
TRANSMIT	Polar transformation active
TRACYL	Cylinder transformation active
TRAORI	Orientation transformation active
TRAANG	Inclined axis transformation active
TRACON	Cascaded transformation active

Proceed as follows

1. Select the "Machine" operating area.

...

2. Press the "JOG", "AUTO", or "MDA" key.

3. Press the "G functions" softkey.
   The "G functions" window is opened.

4. Press the "G functions" softkey again to hide the window again.

The G groups selection displayed in the "G functions" window may differ.

Machine manufacturer
Please also refer to the machine manufacturer's instructions.

Note

Displaying all G functions

You can display all G codes. This displays active transformations in addition to the G functions.

3.9.2 All G functions

All G groups and their group numbers are listed in the "G functions" window.

Within a G group, only the G function currently active in the control is displayed.

Active transformations are also displayed.

Information in the header

Active transformations are displayed in the header.

Display	Description
TRANSMIT	Polar transformation active
TRACYL	Cylinder transformation active
TRAORI	Orientation transformation active
TRAANG	Inclined axis transformation active
TRACON	Cascaded transformation active For TRACON, two transformations (TRAANG and TRACYL or TRAANG and TRANSMIT) are activated in succession.

Proceed as follows

1. Select the "Machine" operating area.

2. Press the "JOG", "AUTO", or "MDA" key.

3. Press the "All G functions" softkey.
   The "G functions" window appears.

4. Press the "All G functions" softkey again to hide the window again.

3.9.3 Auxiliary functions

Auxiliary functions include M and H functions preprogrammed by the machine manufacturer, which pass parameters to the PLC to trigger reactions defined by the manufacturer.

Displayed auxiliary functions

Up to 5 current M functions and 3 H functions are displayed in the "Auxiliary functions" window.

Proceed as follows

1. Select the "Machine" operating area.

2. Press the "JOG", "AUTO", or "MDA" key.

3. Press the "H functions" softkey.
   The "Auxiliary functions" window opens.

4. Press the "H functions" softkey again to hide the window again.

3.10 Displaying status of synchronized actions

3.10.1 Status of synchronized actions

You can display status information for diagnosing synchronized actions in the "Synchronized actions" window.

You get a list with all currently active synchronized actions.

You can see the status of the synchronized actions in the "Status" column.

- Waiting
- active
- Blocked

Non-modal synchronized actions can only be identified by their status display. They are only displayed during execution.

Synchronization types

Synchronization types	Description
ID=n	Modal synchronized actions in automatic mode, local to program; n = 1 to 254
IDS=n	Static synchronized actions in all modes, n = 1 to 254
Without ID/IDS	Non-modal synchronized actions in automatic mode

Note

Numbers ranging from 1 to 254 can only be assigned one time, irrespective of the identification number.

Proceed as follows

1. Select the "Machine" operating area.

2. Press the "AUTO" key.

3. In the expansion area, press the "Synchron." softkey.
   The "Synchronized actions" window appears.

3.11 Settings for automatic mode

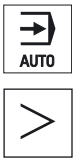
3.11.1 Defining the dry run feedrate

Before machining a workpiece, test the program without moving the machine axes. This allows for early detection of programming errors. For this test, you can use a dry run feedrate that you have defined.

This defined feedrate replaces the programmed feedrate during execution if you have selected "Dry run feedrate" under Program control.

Proceed as follows

1. Select the "Machine" operating area.



2. Press the "AUTO" key and change over to the expanded horizontal softkey bar.

3. Press the "Settings" softkey.
   The "Settings for automatic operation" window is opened.
4. In "DRY run feedrate," enter the desired dry run speed.

Note

The feedrate can be changed while the operation is running.

Teaching in programs

4

4.1 Overview

The "Teach in" function can be used to edit programs in the "AUTO" and "MDA" modes. You can create and modify simple traversing blocks.

You traverse the axes manually to specific positions in order to implement simple machining sequences and make them reproducible. The positions you approach are applied.

In "AUTO" teach-in mode, the selected program is "taught".

In "MDA" teach-in mode, you teach to the MDA buffer.

External programs, which may have been rendered offline, can therefore be adjusted and modified according to need.

4.2 General sequence

General sequence

Select the desired program block, press the relevant softkey "Teach position", "Rap. tra. G01", "Straight line G1" or "Circ. interp. pos. CIP", and "Circ. end pos. CIP" and traverse the axes to change the program block.

You can only overwrite a block with a block of the same type.

- OR -

Position the cursor at the desired point in the program, press the relevant softkey "Teach position", "Rap. tra. G01", "Straight line G1" or "Circ. interp. pos. CIP", and "Circ. end pos. CIP" and traverse the axes to insert a new program block.

In order for the block to be inserted, the cursor must be positioned in an empty line via the "Cursor" and "Input" keys.

Press the "Accept" softkey to teach-in the modified or new program block.

Switching modes or operating areas

If you switch to another operating mode or operating area in teach-in mode, the position changes will be canceled.

4.3 inserting blocks

You have the option of traversing the axes and writing the current actual values directly to a new position block.

Requirement

"AUTO" mode: The program to be edited is selected.

Proceed as follows

1. Select the "Machine" operating area.

2. Press the "AUTO" or "MDA" key.

3. Press the "Teach In" key.

4. Press the "Teach prog." softkey.

5. Traverse the axes to the relevant position.

6. Press the "Teach position" softkey.

   A new program block with the current actual position values will be created.

4.3.1 General information

The windows for inserting program blocks contain input fields for the actual values in the WCS. Depending on the default setting, selection fields with parameters for motion behavior and motion transition are available.

The first time the input fields are selected, they are not prepopulated.

The cursor must be positioned on an empty line.

Requirement

"AUTO" mode: The program to be edited is selected.

Proceed as follows

1 Select the "Machine" operating area.

...


2. Press the "AUTO" or "MDA" key.

3. Press the "Teach In" key.

4. Press the "Teach prog." softkey.

5. Use the cursor and input keys to position the cursor at the desired point in the program.

   If an empty row is not available, insert one.

...


6. Press the softkeys "Rap. tra. G0", "Straight line G1", or Circ. interm. pos. CIP" and "Circ. end pos. CIP".

   The relevant windows with the input fields are displayed.

7. Traverse the axes to the relevant position.

8. Press the "Accept" soft key.

   A new program block will be inserted at the cursor position.

   - OR -

Press the "Cancel" softkey to cancel your input.

4.3.2 Teach in rapid traverse G0

You traverse the axes and teach-in a rapid traverse block with the approached positions.

Note

Selection of axes and parameters for teach-in

You can select the axes to be included in the teach-in block in the "Settings" window.

You also specify here whether motion and transition parameters are offered for teach-in.

4.3.3 Teach in straight G1

You traverse the axes and teach-in a machining block (G1) with the approached positions.

Note

Selection of axes and parameters for teach-in

You can select the axes to be included in the teach-in block in the "Settings" window.

You also specify here whether motion and transition parameters are offered for teach-in.

4.3.4 Teach-in circle intermediate position CIP

Enter the intermediate and end positions for the circle interpolation CIP. You teach-in each of these separately in a separate block. The order in which you program these two points is not specified.

Note

Make sure that the cursor position does not change during teach-in of the two positions.

You teach-in the intermediate position in the "Circle intermediate position CIP" window.

You teach-in the end position in the "Circle end position CIP" window.

The intermediate or interpolation point is only taught-in with geometry axes. For this reason, at least 2 geometry axes must be set up for the transfer.

4.3.5 Teaching in circle end point CIP

Enter the intermediate and end positions for the circle interpolation CIP. You teach-in each of these separately in a separate block. The order in which you program these two points is not specified.

Note

Make sure that the cursor position does not change during teach-in of the two positions.

You teach-in the intermediate position in the "Circle intermediate position CIP" window.

You teach-in the end position in the "Circle end position CIP" window.

The intermediate or interpolation point is only taught-in with geometry axes. For this reason, at least 2 geometry axes must be set up for the transfer.

Note

Selection of axes for teach in

You can select the axes to be included in the teach-in block in the "Settings" window.

4.3.6 Input parameters for teach-in blocks

Parameters for teach-in of position and teach-in of G0, G1, and circle end position CIP

parameters	Description
X	Approach position in X direction
Y	Approach position in Y direction
Z	Approach position in Z direction
F	Feedrate (mm/r; mm/min) - only for teach-in of G1 and circle end position CIP

Parameters for teach-in of circle intermediate position CIP

parameters	Description
I	Coordinate of the circle center point in the X direction
J	Coordinate of the circle center point in the Y direction
K	Coordinate of the circle center point in the Z direction

Motion type for teach-in of position and teach-in of G0 and G1

The following parameters are offered for the motion behavior:

parameters	Description
G60	Exact stop
G64	Corner rounding
G641	Programmable corner rounding
G642	Axis-specific corner rounding
G643	Block-internal corner rounding
G644	Axis dynamics corner rounding

Continuous-path mode for teach-in of position and teach-in of G0 and G1

The following parameters are offered for the transition:

parameters	Description
CP	Path-synchronous
PTP	Point-to-point
PTPG0	Only G0 point-to-point

4.4 Changing blocks

You can only overwrite a program block with a teach-in block of the same type.

The axis values displayed in the relevant window are actual values, not the values to be overwritten in the block.

Note

If you wish to change any variable in a block in the program block window other than the position and its parameters, then we recommend alphanumerical input.

Requirement

The program to be edited is selected.

Proceed as follows

1. Select the "Machine" operating area.

...

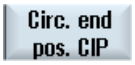
2. Press the "AUTO" or "MDA" key.

3. Press the "Teach In" key.

Press the "Teach prog." softkey.

4. Click the program block to be edited.

...



5. Press the relevant softkey "Teach position, "Rap. tra. G0", "Straight line G1", or "Circ. interm. pos. CIP", and "Circ. end pos. CIP".

   The relevant windows with the input fields are displayed.

6. Traverse the axes to the desired position and press the "Accept" softkey.

   The program block is taught with the modified values.

   - OR -

Press the "Cancel" softkey to cancel the changes.

4.5 Block selection

You have the option of setting the interrupt pointer to the current cursor position. The next time the program is started, processing will resume from this point.

Selecting a block allows you to reteach-in program areas that have already been executed.

Note

The "Select block" function is only available in "AUTO" mode.

Requirement

The program to be edited is selected.

Proceed as follows

1. Select the "Machine" operating area.

2. Press the "AUTO" key.

3. Press the "Teach In" key.

4. Press the "Teach prog." softkey.

5. Press the "Block selection" softkey.

4.6 Deleting blocks

You have the option of deleting a program block entirely.

Requirement

"AUTO" mode: The program to be edited is selected.

Proceed as follows

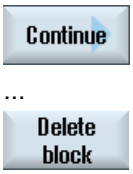
1. Select the "Machine" operating area.

2. Press the "AUTO" or "MDA" key.

3. Press the "Teach In" key.

4. Press the "Teach prog." softkey.

5. Click the program block to be deleted.



6. Press the "Continue" and "Delete block" softkeys.
   The program block on which the cursor is positioned is deleted.

4.7 Parameter settings

4.7.1 Settings for teach-in

In the "Settings" window, you define which axes are to be included in the teach-in block and whether motion-type and continuous-path mode parameters are to be provided.

Proceed as follows

1. Select the "Machine" operating area.

...

2. Press the "AUTO" or "MDA" key.

3. Press the "Teach In" key.
4. Press the "Teach prog." softkey.
5. Select the "Continue" and "Settings" softkeys.

   The "Settings" window appears.

6. Under Axes to be taught and Selection box display, select the relevant check boxes and press the "Accept" softkey to confirm the settings.

Machine and setting data

5

5.1 Displaying and editing machine data

In the "Start-up" operating area, you can display lists for machine data. You can also modify machine data here.

- General machine data
- Channelspecific machine data
- Axis-specific machine data
- Control unit machine data (drive parameters)

Note

Access to the machine data operating area can be controlled by keylock switch or password.

Read access to machine data is available with protection level 4 or higher, and machine data can generally be modified with the manufacturer's password.

Danger

Changes in the machine data have a considerable influence on the machine. Incorrect configuration of the parameters can endanger human life and cause damage to the machine.

Information about machine data

The following information is displayed from left to right:

- Machine data number
- Machine data name, with field index if applicable.
- Value of the machine data
- Unit of the machine data
- Effective

Note

If the machine data does not use units, the corresponding column is empty. If the data are not available, the "#" symbol is displayed instead of the value. If the value ends in an "H", it is a hexadecimal value.

The physical units of machine data are displayed on the right-hand side of the input field.

For each machine data element, an activation type can be read in the column on the right.

so	Immediately active: no action required
cf	Configuration: "Set MD to active" softkey
re	Reset: "Reset" key on the machine control panel
po	POWER ON: "NCK reset" softkey

Proceed as follows

1. Select the "Start-up" operating area.

2. Press the "Mach. data" softkey.

...

3. Press the "General MD", "Channel MD", "Axis MD", or "Control unit MD" softkey.

   The "General MD", "Channel-specific MD", "Axis-specific MD", or "Control unit MD" window will open.

4. Position the cursor on the entry to be modified and enter the desired value.

...

5. Press the "Set MD to active" or "NCK" softkey, depending on the activation type specified.

5.2 Setting data

In the "Start-up" operating area, you can display and modify lists for all setting data:

- General setting data
- Channelspecific setting data
- Axis-specific setting data

Proceed as follows

1. Select the "Start-up" operating area.

2. Press the "Mach.data" softkey.

3. Press the ETC key to show the "General SD", "Channel SD", and "Axis SD" softkeys.

4. Press the "General SD", "Channel SD", or "Axis SD" softkey.

...

5.3 Specify working area limitations

The "Working area limitation" function can be used to limit the range within which a tool can traverse in all channel axes. These commands allow you to set up protection zones in the working area which are out of bounds for tool movements.

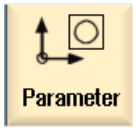
In this way, you are able to restrict the traversing range of the axes in addition to the limit switches.

Prerequisites

You can only make changes in "AUTO" mode when in the RESET condition. These changes are then immediate.

You can make changes in "JOG" mode at any time. These changes, however, only become active at the start of a new motion.

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "Setting data" softkey.

The "Working area limit." window appears.

3. Place the cursor in the required field and enter the new values via the numeric keyboard.

   The upper or lower limit of the protection zone changes according to your inputs.

4. Click the checkbox to "active" to activate the protection zone.

Note

A list of all setting data is available in the "Start-up" operating area.

5.4 Searching for machine and setting data

You can search for specific machine and setting data.

Proceed as follows

1. Select the "Start-up" operating area.

2. Press the "Mach.data" softkey.

...

3. Press the "General MD", "Channel MD", "Axis MD", or "Control unit MD" softkey.

   The corresponding window will open to display the list of all machine and setting data.

Press the "Search..." softkey and enter the desired alphanumeric text in the search form.

The cursor is positioned on the first entry that corresponds to the search term.

Press the "Continue search" softkey if this machine or setting data element is not the one you are looking for.

User data

6

6.1 Introduction

The defined user data may be displayed in lists.

The following variables can be defined:

- Data parameters (R parameters)
- Global user data (GUD) is valid in all programs
- Local user data (LUD) is valid in one program
- Program-global user data (PUD) is valid in one program and the called subroutines.

Channel-specific user data can be defined with a different value for each channel.

Entering and displaying parameter values

Up to 15 positions (including decimal places) are evaluated. If you enter a number with more than 15 places, it will be written in exponential notation (15 places + EXXX).

LUD or PUD

Only local or program-global user data can be displayed at one time.

Whether the user data are available as LUD or PUD depends on the current control configuration.

Machine manufacturer

Please also refer to the machine manufacturer's instructions.

Note

Reading and writing variables protected

Reading and writing of user data are protected via a keyswitch and protection levels.

Searching for user data

You may search for user data within the lists using any character string.

Refer to the "Defining and activating user data" section to learn how to edit displayed user data.

6.2 R parameters

R parameters (arithmetic parameters) are channel-specific variables that you can use within a G code program. G code programs can read and write R parameters.

These values are retained after the control is switched off.

Number of channel-specific R parameters

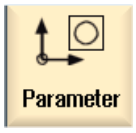
The number of channel-specific R parameters is defined in a machine data element.

Range: R0-R999 (dependent on machine data).

There are no gaps in the numbering within the range.

Machine manufacturer
Please also refer to the machine manufacturer's instructions.

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "User variable" softkey.

3. Press the "R variables" softkey.
   The "R parameters" window appears.

Delete R variables

You arrive at the "Delete area" softkey via the "Continue" softkey.

- You can delete the channel-specific values for individual R parameters or R parameters of a specific area. The value 0 is entered in their place.
- Press the "Delete all" softkey to delete all values defined for the channel and set all values to 0.

6.3 Displaying global user data (GUD)

Global user data

Global GUDs are NC global user data (**G**lobal **U**ser **D**ata) that remains available after switching the machine off.

GUDs apply in all programs.

Definition

A GUD variable is defined with the following:

- Keyword DEF
- Range of validity NCK
- Data type (INT, REAL, ….)
- Variable names
- Value assignment (optional)

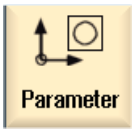
Example

DEF NCK INT ZAEHLER1 = 10

GUDs are defined in files with the ending DEF. The following file names are reserved for this purpose:

File name	Description
MGUD.DEF	Definitions for global machine manufacturer data
UGUD.DEF	Definitions for global user data
GUD4.DEF	User-definable data
GUD8.DEF, GUD9.DEF	User-definable data

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "User variable" softkey.

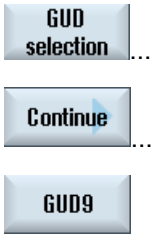
3. Press the "Global GUD" softkeys.

   The "Global user data" window is displayed. A list of the defined UGUD variables will be displayed.

- OR -

Press the "GUD selection" softkey and the "SGUD" to "GUD6" softkeys if you want to display GUD 1 to GUD 6 of the global user data.

- OR -



Press the "GUD selection" and "Continue" softkeys as well as the "GUD7" to "GUD9" softkeys if you want to display GUD 7 to GUD 9 of the global user data.

Note

After each start-up, a list with the defined UGUD variables is displayed in the "Global user data" window.

6.4 Displaying channel GUDs

Channel-specific user data

Like the GUDs, channel-specific user data are applicable in all programs for each channel. However, unlike GUDs, they have specific values.

Definition

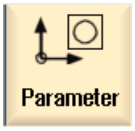
A channel-specific GUD variable is defined with the following:

- Keyword DEF
- Range of validity CHAN
- Data type
- Variable names
- Value assignment (optional)

Example

DEF CHAN REAL X_POS = 100.5

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "User variable" softkey.

3. Press the "Channel GUD" and "GUD selection" softkeys.

A new vertical softkey bar appears.

4. Press the "SGUD" to "GUD6" softkeys if you want to display GUD 1 to GUD 6 of the channel-specific user data.

- OR -

Press the "Continue" softkey and the "GUD7" to "GUD9" softkeys if you want to display GUD 7 to GUD 9 of the channel-specific user data.

6.5 Displaying local user data (LUD)

Local user data

LUDs are only valid in the program or subroutine in which they were defined.

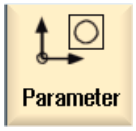
The control displays the LUDs after the start of program processing. The display is available until the end of program processing.

Definition

A local user variable is defined with the following:

- Keyword DEF
- Data type
- Variable names
- Value assignment (optional)

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "User variable" softkey.

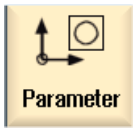
3. Press the "Local LUD" softkey.

6.6 Displaying program user data (PUD)

Program-global user data

PUDs are global part program variables (**P**rogram **U**ser **D**ata). PUDs are valid in all main programs and subroutines, where they can also be written and read.

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "User variable" softkey.

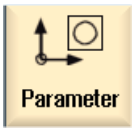
3. Press the "Program PUD" softkey.

6.7 Searching for user data

You can search for R parameters and user data.

Enter the R parameter number you wish to find via the numeric keypad.

Proceed as follows



1. Select the "Parameter" operating area.

...


2. Press the "R variables", "Global GUD", "Channel GUD", "Local GUD" or "Program PUD" softkeys to select the list in which you would like to search for user data.

3. Press the "Search" softkey.

   The "Search for R parameters" or "Search for user data" window opens.

4. Enter the desired search term and press "OK".

   The cursor is automatically positioned on the the R parameter or user data you are searching for, if they exist.

6.8 Defining and activating user data

By editing a DEF/MAC file, you can alter or delete existing definition/macro files or add new ones.

Proceed as follows

1. Select the "Program manager" operating area.

2. Press the "System data" softkey in the expansion area.
3. In the directory structure, select the "NC data" folder and then open the "Definitions" folder.
4. Select the file you want to edit.
5. Double-click the file.

   - OR -

   Press the "Open" softkey.

   - OR -

   Press the "Input" key.

   - OR -

   Press the "Right cursor" key.

   The selected file is opened in the editor and can be edited there.
6. Define the desired user data.
7. Press the "Exit" softkey to close the editor.

Activating user data

1. Press the "Activate" softkey.

   A prompt is displayed.
2. Select whether the current values in the definition files should be retained

   - OR -

   Select whether the current values in the definition files should be deleted.

   This will overwrite the definition files with the initial values.

3. Press the "OK" softkey to continue the process.

Tool management

7

7.1 Overview

Various tools are used for machining workpieces.

Lists for tool management

HMI sl provides the "Tool list", "Tool wear list", "OEM tools" and "Magazine" windows for managing your tools.

The various lists may have been changed by the machine manufacturer, in comparison to the following description.

Machine manufacturer

Please also refer to the machine manufacturer's instructions.

For additional information, please refer to the following documentation:

HMI sl / SINUMERIK 840D sl start-up guide

7.2 Tool list

You must enter all the tools that you want to use on the machine in the tools list.

Each tool is clearly identified and stored with all geometry values.

Tool parameters

Column heading	Description
Location  	Location number • Spindle location as an icon • The locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons. • The magazine location numbers The magazine number is specified first, followed by the location number. Tools that are not assigned to a magazine in the tool list are stored behind the magazine without a location number.

Column heading	Description
Type	Tool type Depending on the tool type (represented by a symbol), certain tool offset data are enabled.
Tool name	A tool is identified by its name and duplo number of the tool. You may enter the names as text or numbers.
DP	Duplo number of twin tool (replacement tool).
D	Tool offset data for a selected cutting edge of a tool.
Length 1	Tool length Geometry data length 1
Radius	Tool radius
N	Number of teeth

Icons in the tool list

Icon / Identification	Description
Magazine location	
Red "X"	The magazine location is disabled.
Type	
Red "X"	The tool is disabled.
Yellow triangle	The prewarning limit has been reached.

7.2.1 Entering new tools in the tool list

A number of tool types are available when you create a new tool. The tool type determines which geometry data are required and how they will be computed.

Possible tool types

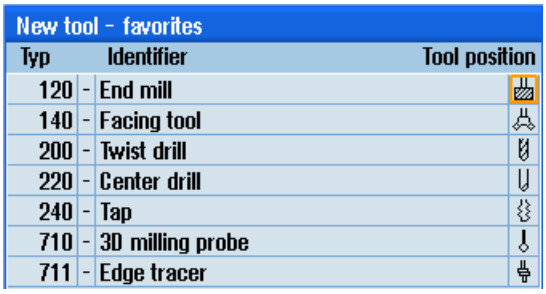


Figure 7-1 Example of Favorites list

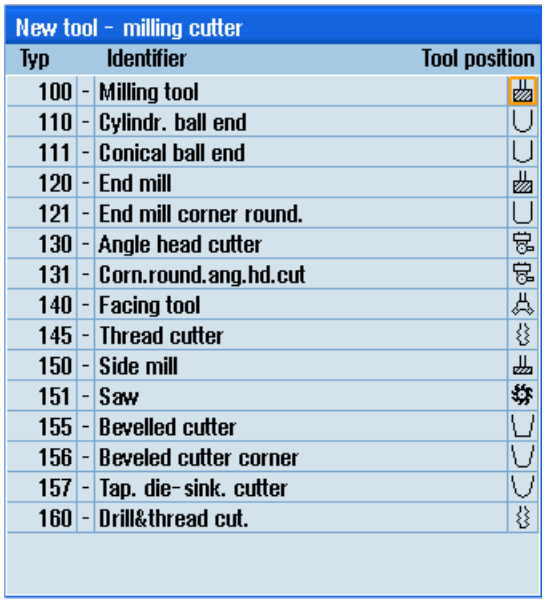


Figure 7-2 Available tools in the "New tool - milling cutter" window

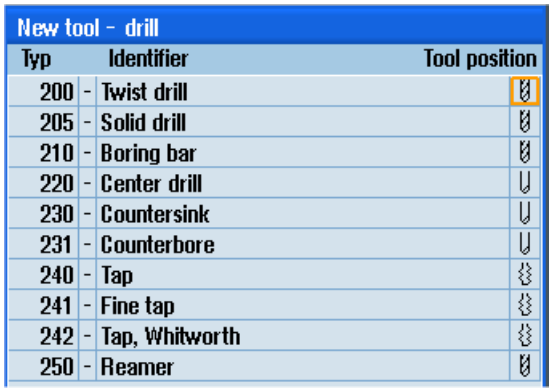


Figure 7-3 Available tools in the "New tool - drill" window

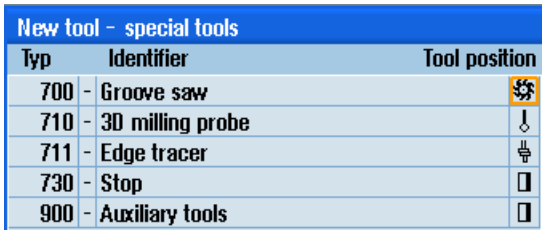
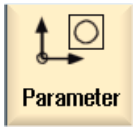


Figure 7-4 Available tools in the "New tool - special tools" window

When creating a new tool, the "New tool - favorites" window offers you a number of select tool types, known as "favorites".

If you do not find the desired tool type in the favorites list, then select the milling, drilling or special tool via the corresponding softkeys.

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "Tool list" softkey.

3. Place the cursor in the tool list at the position where the new tool should be stored.

   For this, you can select an empty magazine location or the NC tool memory outside of the magagine.

   You may also place the cursor on an existing tool in the NC tool memory region. Data from the displayed tool will not be overwritten.

4. Press the "New tool" softkey.

The "New tool - favorites" window appears on the screen.

- OR -

...

Press the softkey "Cutters 100-199", "Drills 200-299" or "Sp. tools 700-799".

The "New tool - milling cutter", "New tool - drill", or "New tool - special tools" window opens.

5. Select the tool by placing the cursor on the corresponding icon.

6. Press the "OK" softkey.

   The "New tool" window appears.

7. Enter the desired data.

8. Press the "OK" softkey.

   The tool is inserted into the tool list. If the cursor is located on an empty magazine location in the tool list, then the tool is loaded to this magazine location.

Multiple load points

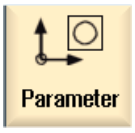
If you have configured several loading points for a magazine, then the "Load point selection" window appears after pressing the "Load" softkey.

Select the required load point and confirm with the "OK" softkey.

7.2.2 Managing several cutting edges

In the case of tools with more than one cutting edge, a separate set of offset data is assigned to each cutting edge. The number of possible cutting edges depends on the control configuration.

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "Tool list" softkey.
3. Position the cursor on the tool for which you would like to store more cutting edges.

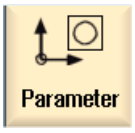
4. Press the "Edges" softkey in the "Tool list".

5. Press the "New cuttgEdge" softkey.
   A new data set is stored in the list.
   The edge number is incremented by 1 and the offset data are assigned the values of the edge on which the cursor is positioned.
6. Enter the offset data for the second cutting edge.
7. Repeat this process if you wish to create more tool edge offset data.

7.2.3 Deleting cutting edges

You can delete unnecessary edges of a tool from the tool list.

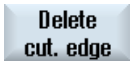
Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "Tool list" softkey.
3. Place the cursor on the edge of a tool that you would like to delete.

4. Press the "Edges" softkey in the "Tool list".

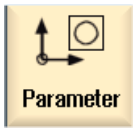


5. Press the "Delete cut. edge" softkey.
   The data set is deleted from the list.

7.2.4 Deleting tools

Tools that are no longer in use can be deleted from the tool list for a clearer overview.

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "Tool list" softkey.
3. Place the cursor on the tool that you would like to delete.

4. Press the "Delete tool" softkey.

5. Press the "OK" softkey.

   The tool is deleted and the magazine location is enabled again.

Multiple load points - tool in magazine location

If you have configured several loading points for a magazine, then the "Loading point selection" window displays after pressing the "Delete tool" softkey.

Select the required load point and press the "OK" softkey to unload and delete the tool.

7.2.5 Loading and unloading tools

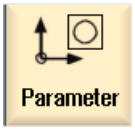
You can load and unload tools to and from a magazine via the tool list. When a tool is loaded, it is taken to a magazine location. When it is unloaded, it is removed from the magazine and stored in the tool list.

When you are loading a tool, the application automatically suggests an empty location. You may also directly specify an empty magazine location.

You can unload tools from the magazine that you are not using at present. HMI sl then automatically saves the tool data in the tool list outside the magazine.

Should you want to use the tool again later, simply load the tool with the tool data into the corresponding magazine location again. Then the same tool data does not have to be entered more than once.

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "Tool list" softkey.

3. Place the cursor on the tool that you want to load into the magazine (if the tools are sorted according to magazine location number you will find it at the end of the tool list).

4. Press the "Load" softkey.

   The "Load to… " window opens.

   The "... empty location" field is initialized with the number of the first empty magazine location.

5. Press the "OK" softkey to load the tool into the suggested location.

   - OR -

   Enter the location number you require and press the "OK" softkey.

   The tool is loaded into the specified magazine location.

Several magazines

If you have configured several magazines, the "Load to ..." window appears after pressing the "Load" softkey.

If you do not want to use the suggested empty location, then enter your desired magazine and magazine location. Confirm your selection with "OK".

Multiple load points

If you have configured several loading points for a magazine, then the "Load point selection" window appears after pressing the "Load" softkey.

Select the required loading point and confirm with "OK".

Unloading tools

Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" softkey.

7.3 Wear list

Tools that are in use for long periods are subject to wear. You can measure this wear and enter it in the tool wear list. The control then takes this information into account when calculating the tool length or radius compensation. This ensures a consistent accuracy in workpiece machining.

Tool parameters

Column heading	Description
Location	Display of magazine location number.
Type	Display of tool type.
Tool name	Display of tool name.
DP	Duplo number of twin tool (replacement tool).
D	The displayed wear data is relative to the specified cutting edge of the tool.
Δ Length 1	Wear in the Z direction
Δ Radius	Radius wear
T C	Selection of tool monitoring - by tool life (T) - by count (C) - by wear (W)
Setpoint	Setpoint for tool life, workpiece count, or wear
Pre-warning limit	Specification of the tool life, workpiece count or wear at which a warning is displayed.
Tool life	Tool life.
Number of workpieces	Number of completed workpieces.
Wear	Maximum permissible tool wear.
G	The tool is disabled when the check box is selected.

Symbols in the wear list

Icon / Identification	Description
Magazine location	
Red "X"	The magazine location is disabled.
Type	
Red "X"	The tool is disabled.
Yellow triangle	The prewarning limit has been reached.

Activating tool monitoring

You can automatically monitor the tools' working times via the workpiece count, tool life or wear.

In addition, you can disable tools when you no longer wish to use them.

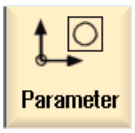
7.3.1 Reactivating tools

You can replace disabled tools or make them ready for use again.

Prerequisites

In order to reactivate a tool, the monitoring function (supervision) must be activated and a setpoint must be stored.

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "Tool wear" softkey.
3. Position the cursor on the disabled tool which you would like to reuse.

4. Press the "Reactivate" softkey.

   The value entered as the setpoint is entered as the new tool life or workpiece count.

   The disabling of the tool is cancelled.

Reactivate with positioning

When the "Reactivate with positioning" function is activated, the selected tool's magazine location will be positioned at a loading point. You can exchange the tool.

Multiple load points

If you have configured several loading points for a magazine, then the "Load point selection" window appears after pressing the "Load" softkey.

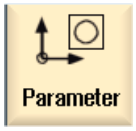
Select the required load point and confirm with the "OK" softkey.

7.4 OEM tool list

In the OEM tool list, you can customize a list according to your specific needs.

For more information, see the following references:

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "OEM Tool" soft key.

7.5 Magazine list

All tools that are assigned to one or several magazines are contained in the magazine list. Individual magazine locations can be reserved or disabled for existing tools.

Tool parameters

Column heading	Description
Location	Display of magazine location numbers.
Type	Display of tool type with icons.
Tool name	Display of tool name.
DP	Display of duplo number.
D	Display of edge number.
G	Disabling of the magazine location.
Mag.loctype	Display of magazine location type
Tool.loctype	Display of magazine location type of tool.
L R	Assignment of neighboring magazine locations. Enables a tool to labeled as oversized. Up to 7 neighboring half locations may be reserved for oversized tools.
P	Fixed location coding. The tool is assigned to a magazine location.

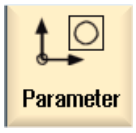
Magazine list icons

Icon / Identification	Description
Magazine location	
Red "X"	The magazine location is disabled.
Type	
Red "X"	The tool is disabled.
Yellow triangle	The prewarning limit has been reached.

7.5.1 Position magazine

You can position magazine locations directly on the loading point.

Proceed as follows



1. Select the "Parameter" operating area.

2. Press the "Magazine" softkey.

3. Place the cursor on the magazine location that you want to position onto the load point.

4. Press the "Position magazine" softkey.

   The magazine location is positioned on the loading point.

Multiple load points

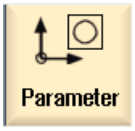
If you have configured several loading points for a magazine, then the "Position magazine" window displays after pressing the "Position magazine" softkey.

Select the desired loading point in the "Target" field and confirm your selection with "OK" to position the magazine location at the loading station.

7.6 Sorting tool management lists

When you are working with many tools, with large magazines or several magazines, it is useful to display the tools sorted according to different criteria. Then you will be able to find a specific tool more easily in the lists.

Proceed as follows



1. Select the "Parameter" operating area.

...

2. Select the "Tool list", "Tool wear" or "Magazine" softkey in the "Parameter" operating area.

...

3. Select the "Continue" and "Sort" softkeys.

The lists are displayed sorted numerically according to magazine location.

Tool types are used to sort tools with the same magazine location. Identical types (e.g. milling cutters), in turn, are sorted according to their radius value.

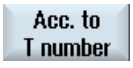
4. Press the "Surname" softkey to display the tool names in alphabetical order.

Duplo numbers are used to sort tools with the same names.

- OR -

Press the "Acc. to type" softkey to display the tools arranged by location type.

- OR -



Press the "Acc. to T number" softkey to display the tools numerically sorted by tool number.

The list is sorted according to the specified criteria.

Program management

8

8.1 Overview

Place all programs that you would like to execute in the NC.

You can access these programs at any time via the Program Manager for execution, editing, copying, or renaming. Programs that you no longer require can be deleted to release their storage space.

Storage for programs

Possible storage locations are:

- NC
- Local drives
- Network drives
- USB drives

Note

Options

You need the "256 MB user memory" option in order to display the "Local drive" softkey.

To set up network drives, you need the "Network drives" option.

Communication with other workstations

You have the following options for exchanging programs and data with other workstations:

- USB drives (e.g. USB FlashDrive)
- Network drives

Choosing storage locations

In the horizontal softkey bar, you can select the storage location that contains the directories and programs that you want to display. In addition to the "NC" softkey, via which the passive file system data can be displayed, additional softkeys can be displayed.

The "USB" softkey is only operational when an external storage medium is connected (e.g., USB FlashDrive on the USB port of the operator panel).

Structure of the directories

In the overview, the symbols in the left-hand column have the following meaning:

Folder

Program

The directories and programs are always listed complete with the following information:

- Name

  The name can contain up to 24 characters + dot + 3-character extension (e.g., MPF).

  Permissible characters include all upper-case letters (no umlauts), numbers, and underscores.

- Type

  Directory: WPD

  Program: MPF

  Subroutine: SPF

  Zero point/tool data: INI

- Size (in bytes)
- Date/time (of creation or last change)

8.1.1 NC memory

The complete NC working memory is displayed along with all tools and the main programs and subroutines.

You can create further subdirectories here.

Proceed as follows

1. Select the "Program manager" operating area.

2. Press the "NC" softkey.

8.1.2 Local drives

The workpieces, main programs and subroutines stored in HMI sl are displayed.

You can create any number of subdirectories here, in which to store any files (e.g., text files with notes).

Proceed as follows

1. Select the "Program manager" operating area.

2. Press the "Local drive" softkey.

8.1.3 USB drives

USB drives enable you to exchange data. For example, you can copy to the NC and execute programs that were created externally.

Proceed as follows

1. Select the "Program manager" operating area.

2. Press the "USB" softkey.

Note

The "USB" softkey can only be operated when a USB FlashDrive is inserted in the front interface of the operator panel.

8.2 Opening and closing a program

To view a program in more detail or modify it, open the program in the editor.

Proceed as follows

1. Select the "Program manager" operating area.
2. Select the desired storage location and position the cursor on the program that you would like to edit.

3. Press the "Open" softkey.

- OR -

Press the "Input" key.

- OR -

Press the "Right cursor" key.

- OR -

Double-click the required program.

The selected program is opened in the "Editor" operating area.

4. Press the "Continue" and "Exit" softkeys to close the program and editor again.

...

Note

A program does not have to be closed in order for it to be executed.

8.3 Executing a program

When you select a program for execution, the control automatically switches to the "Machine" operating area.

Program selection

Select the workpieces (WPD), main programs (MPF) or subroutines (SPF) by placing the cursor on the desired program or workpiece.

For workpieces, the workpiece directory must contain a program with the same name. This program is automatically selected for execution (e.g., when you select the workpiece SHAFT.WPD, the main program SHAFT.MPF is automatically selected).

If an INI file of the same name exists (e.g., SHAFT.INI), it will be executed once at the first part program start after selection of the part program. Any additional INI files are executed in accordance with machine data 11280 $MN_WPD_INI_MODE.

$MN_WPD_INI_MODE=0:

The INI file with the same name as the selected workpiece is executed. For example, when you select SHAFT1.MPF, the SHAFT1.INI file is executed upon "Cycle Start".

$MN_WPD_INI_MODE=1:

All files of type INI, SEA, GUD, RPA, UFR, PRO, TOA, TMA and CEC which have the same name as the selected main program are executed in the specified sequence. The main programs stored in a workpiece directory can be selected and processed by several channels.

Machine manufacturer

Please also refer to the machine manufacturer's instructions.

Proceed as follows

1. Select the "Program manager" operating area.

2. Select the desired storage location and position the cursor on the workpiece/program that you would like to execute.

3. Press the "Execute" softkey.

   The control switches automatically into the "Machine" operating area.

4. Then press the "Cycle Start" key.

   Execution of the workpiece is started.

Note

Only workpieces/programs located in the NCK memory can be selected for execution.

8.4 Creating directories/programs

8.4.1 Creating a new directory

Directory structures help you to manage your program and data transparently. You can create subdirectories in a directory on local drives and USB network drives.

In a subdirectory, in turn, you can create programs and then create program blocks for them.

Note

Directories names must end in .DIR or .WPD. The maximum name length is 28 characters including the extension.

All letters (except umlauts), numbers, and underscores are permitted for name assignment. These names are automatically converted to upper-case letters.

This limitation does not apply for work on USB/network drives.

Proceed as follows

1. Select the "Program manager" operating area.

2. Select the desired storage medium and position the cursor on the folder, in which you would like to create a directory.

3. Press the "New" and "Directory" softkeys.

...

The "New directory" window appears.

4. Enter the desired directory name and press the "OK" softkey.
   The new directory is displayed.

8.4.2 Creating a new workpiece

You can set up various types of files such as main programs, initialization files, tool offsets, etc. in a workpiece.

Note

Additional workpiece directories cannot be created within a workpiece directory (WPD).

Proceed as follows

1. Select the "Program manager" operating area.

2. Select the desired storage location and position the cursor on the folder, in which you would like to create a workpiece.

3. Press the "New" and "Workpiece" softkeys.

...

The "New workpiece" window appears.

4. Enter the desired workpiece name and press the "OK" softkey.

The name can contain up to 28 characters (name + dot. + 3-character extension).

You can use any letters (except umlauts), digits or the underscore symbol (_).

The directory type (WPD) is set by default.

A new folder with the workpiece name will be created.

The "New G code program" window will open.

5. Press the "OK" softkey again if you want to create the program.

The program will open in the editor.

8.4.3 Creating a new G code program

You can create G code programs and then render G code blocks for them in a directory/workpiece.

Proceed as follows

1. Select the "Program manager" operating area.

2. Select the desired storage location and position the cursor on the folder, in which you would like to store the program.

3. Press the "New" and "G code" softkeys.

The "New G code program" window appears.

...

4. Select a template if any are available.
5. Select the file type (MPF or SPF).

6. Enter the desired program name and press the "OK" softkey.

   The program name can contain up to 28 characters (name + dot. + 3-character extension).

   You can use any letters (except umlauts), digits or the underscore symbol (_).

8.4.4 Storing any new file

You can store any type of file in any directory or subdirectory.

Proceed as follows

1. Select the "Program manager" operating area.

2. Select the desired storage location and position the cursor on the folder, in which you would like to create the file.

3. Press the "New" and "Any" softkeys.

   The "Any new program" window appears

...

4. In the "Type" selection box, select the desired file type (e.g., "GUD").

   The file created automatically has the selected file format.

   - OR -

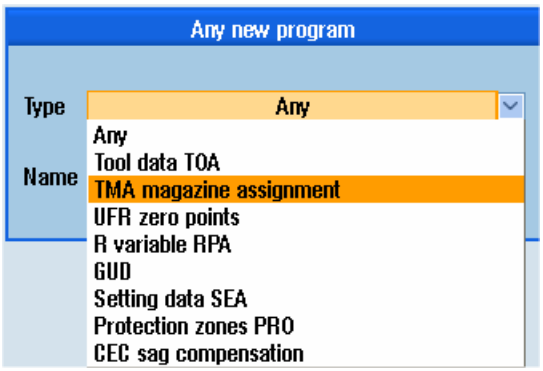
   Select "Any" under type.

   You can now specify any file format (e.g., My_Text.txt).

5. Enter the desired file name and press the "OK" softkey.

   The name can contain up to 28 characters (name + dot. + 3-character extension).

   You can use any letters (except umlauts), digits or the underscore symbol (_).



8.5 Selecting several directories/programs

You can select several files and directories for further processing. When you select a directory, all directories and files located beneath it are also selected.

Note

If you have selected several directories and one of them closes, then selection of the directory and all of the files contained therein is cancelled.

Proceed as follows

1. Select the "Program manager" operating area.
2. Choose the desired storage location and position the cursor on the file or directory from which you would like your selection to start.

3. Press the "Mark" softkey.

The softkey is active.

4. Select the required directories/programs with the cursor or mouse.

5. Press the "Mark" softkey again to deactivate the cursor key and cancel the selection.

Canceling a selection

By reselecting an element, the existing selection is cancelled.

Selecting via keys

Key combination	Description
SELECT	Renders or expands a selection. You can only select individual elements.
SHIFT + ▼ ... ▲	Renders a consecutive selection.
INSERT	A previously existing selection is cancelled.

Selecting with the mouse

Key combination	Description
Left mouse	Click on element: the element is selected. A previously existing selection is cancelled.
Left mouse + SHIFT pressed	Expand selection consecutively up to the next click.
Left mouse + CTRL pressed	Expand selection to individual elements by clicking. An existing selection will expand to include the element you clicked.

8.6 Copying and inserting directories/programs

To create a new directory or program that is similar to an existing program, you can save time by copying the old directory or program and only changing selected programs or program blocks.

The capability of copying and pasting directories and programs can also be used to exchange data with other systems via USB/network drives (e.g., USB FlashDrive).

Copied files or directories can be reinserted in a different location.

Note

You can only insert directories on local drives and on USB or network drives.

Note

Write protection

If the current directory is write-protected for the user, then the function is not offered.

Note

When you copy directories, any missing endings are added automatically.

All letters (except umlauts), numbers, and underscores are permitted for name assignment. The names are automatically converted to upper-case letters, and extra dots are converted to underscores.

Example

If the name is not changed during the copy procedure, a copy is created automatically:

MYPROGRAM.MPF is copied to MYPROGRAM__1.MPF. The next time it is copied, it is changed to MYPROGRAM__2.MPF, etc.

If the files MYPROGRAM.MPF, MYPROGRAM__1.MPF, and MYPROGRAM__3.MPF already exist in a directory, MYPROGRAM__2.MPF is created as the next copy of MYPROGRAM.MPF.

Proceed as follows

1. Select the "Program manager" operating area.

2. Choose the desired storage location and position the cursor on the file or directory which you would like to copy.

3. Press the "Copy" soft key.

4. Select the directory in which you want to insert your copied directory/program.

5. Press the "Insert" softkey.

If a directory/program of the same name already exists in this directory, a prompt asks whether you want to overwrite the existing directory or insert it under a different name.

If the name contains illegal characters or is too long, a prompt will appear for you to enter a permissible name.

6. Press the "OK" or "Overwrite all" softkey if you want to overwrite existing directories/programs.

...

- OR -

Press the "No overwriting" softkey if you do not want to overwrite already existing directories/programs.

- OR -

Press the "Skip" softkey if the copy operation is to be continued with the next file.

- OR -

Enter another name if you want to insert the directory/program under another name and press the "OK" softkey.

Note

Copying files in the same directory

You cannot copy files to the same directory. You must copy the file under a new name.

8.7 Deleting directories/programs

Delete programs or directories from time to time that you are no longer using to maintain a clearer overview of your data management. Back up the data beforehand, if necessary, on an external data medium (e.g., USB FlashDrive) or on a network drive.

Please note that when you delete a directory, all programs, tool data and zero point data and subdirectories that this directory contains are deleted.

Proceed as follows

1. Select the "Program manager" operating area.

2. Choose the desired storage location and position the cursor on the file or directory that you would like to delete.

3. Press the "Continue" and "Delete" softkeys.

   A prompt appears asking whether you really want to delete the file or directory.

...

4. Press the "OK" softkey to delete the program/directory.

   - OR -.

   Press the "Cancel" softkey to cancel the process.

8.8 Renaming file and directory properties

Information on directories and files can be displayed in the "Properties for ..." window.

Information on the creation date is displayed near the file's path and name.

You can change names.

Proceed as follows

1. Select the Program Manager.

2. Choose the desired storage location and position the cursor on the file or directory whose properties you want to display or change.

3. Select the "Continue" and "Properties" softkeys.
   The "Properties from ..." window appears.

...

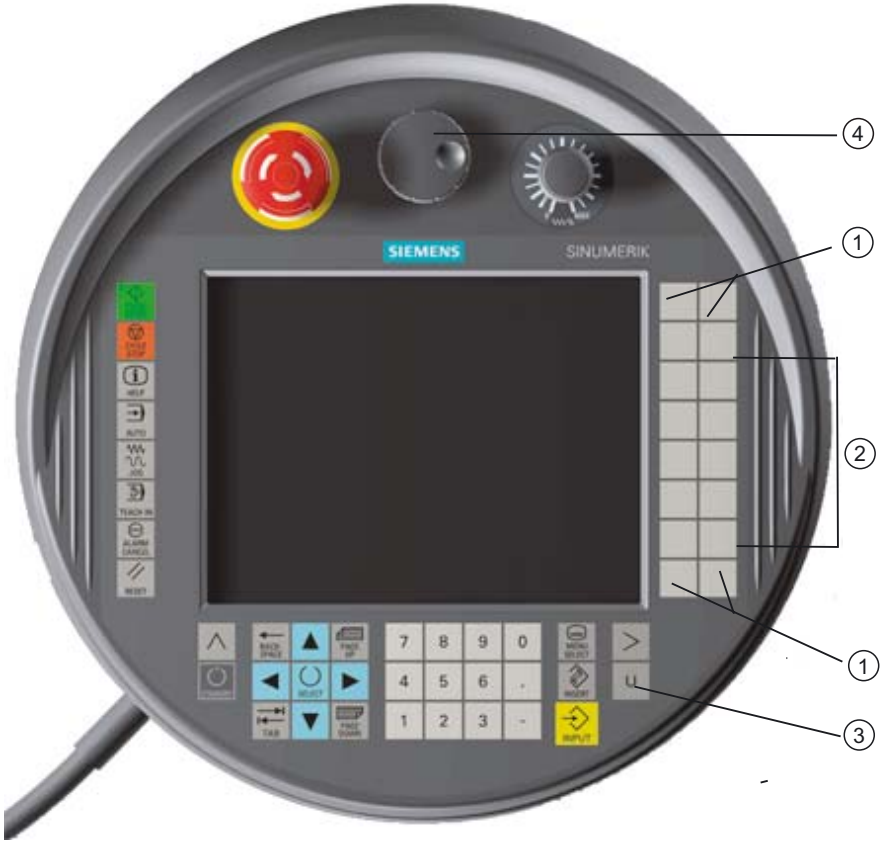
4. Enter any necessary changes.

5. Press the "OK" softkey to save the changes.

HT8

9

The mobile SINUMERIK HT8 handheld terminal combines the functions of an operator panel and a machine control panel. It is therefore suitable for visualization, operation, teach in, and programming at the machine.



1	Customer keys (user-defined)
2	Traversing keys
3	User menu key
4	Handwheel (optional)

Operation

The 7.5 TFT color display provides touch operation.

It also has membrane keys for traversing the axes, for numeric input, for cursor control, and for machine control panel functions like start and stop.

It is equipped with an emergency stop button and two 3-position enabling buttons. You can also connect an external keyboard.

For more information about connection and start-up of the HT8, see the following references:

BH sl Operator Components Manual

IM5 / TCU and HT8 startup

Customer keys

The four customer keys are freely assignable and can be set up customer-specifically by the machine manufacturer.

Machine manufacturer
Please also refer to the machine manufacturer's instructions.

Integrated machine control panel

The HT8 has an integrated machine control panel consisting of keys (e.g., start, stop, traversing keys, etc.), and keys reproduced as softkeys (see machine control panel menu).

See chapter "Controls on the machine control panel" for a description of the individual keys.

Enabling button

The HT8 has two enabling buttons. Thus, you can initiate enabling functions for operations that require enabling (e.g., displaying and operating of traversing keys) with either your right hand or your left hand.

Enabling buttons are available for the following key positions:

- Released (no activation)
- Enabling (center position) - enabling for channel 1 and 2 is on the same switch.
- Panic (completely pushed through)

Traversing keys

In order to traverse the axes of your machine using the traversing keys of the HT8, you must select "JOG" mode or either the "Teach In" or "Ref.Point" submode. Depending on the setting, the enabling button must be activated.

Machine manufacturer

Please also refer to the machine manufacturer's instructions.

Virtual keyboard

A virtual keyboard is available for the easy entry of values.

Channel switchover

You are able to switch the channel by touch in the status display:

- In the Machine operating area (large status display), by touch operation of the channel display in the status display.
- In the other operating areas (no status display), by touch operation of the channel display in the screen headers (yellow field).

Operating-area switchover

You can display the operating area menu by touching the display symbol for the active operating area.

Handwheel

The HT8 is available with a hand wheel.

For information about connecting the hand wheel, refer to:

/IM5/ Commissioning Manual, Thin Client Configuration

See also

Channel switchover (Page 2-4)

9.1 Traversing keys

The traversing keys are not labeled. However, you can display a label for the keys in place of the vertical softkey bar.

Labeling of the traversing keys is displayed for up to 6 axes on the touch panel by default.

Machine manufacturer
Please also refer to the machine manufacturer's instructions.

Showing and hiding

You can link the showing and hiding of the label to activation of the enabling button, for example. In this case, the traversing keys are displayed when you press the enabling button.

If you release the enabling button, the traversing keys are hidden again.

Machine manufacturer
Please also refer to the machine manufacturer's instructions.

All existing vertical and horizontal softkeys will be covered up or hidden, i.e., other softkeys cannot be used.

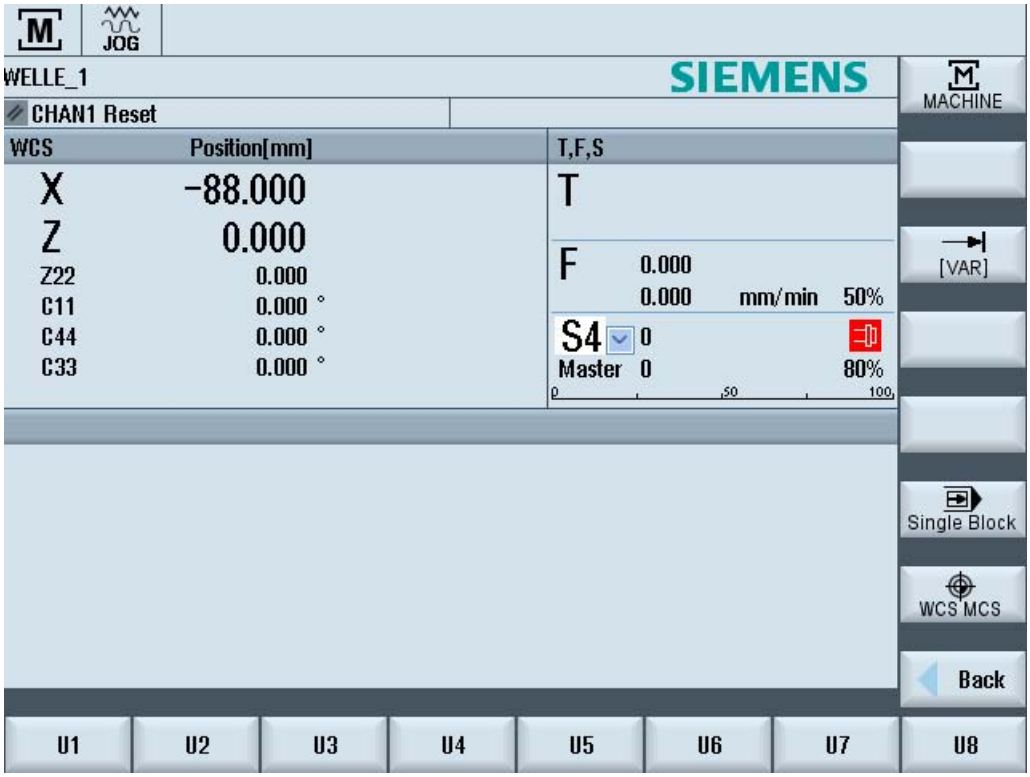
9.2 Machine control panel menu

Here you select keys from the machine control panel which are reproduced by the software by touch operation of the relevant softkeys.

See chapter "Controls on the machine control panel" for a description of the individual keys.

Showing and hiding

The user menu key "U" displays the CPF softkey bar (vertical softkey bar) and the user softkey bar (horizontal softkey bar).



You use the "Back" softkey to hide the menu bar again.

Softkeys on the machine control panel menu

Available softkeys:

"Machine" softkey	Select the "Machine" operating area
"[VAR]" softkey	Select the axis feedrate in the variable increment
"Single Block" softkey	Switch single block execution on/off
"WCS MCS" softkey	Switch between WCS and MCS
"Back" softkey	Close the window.

Note

The window will automatically disappear when changing regions areas with the "Menu Select" key.

9.3 Virtual keyboard

The virtual keyboard is used as the input device for touch operator panels.

It opens when you double-click an operator element with input capability (editor, edit field). The virtual keyboard can be positioned anywhere on the operator interface. In addition, you can toggle between a full keyboard and a reduced keyboard that only includes the number block. Moreover, with the full keyboard, you can toggle between English key assignments and the keyboard assignment for the current language setting.

Proceed as follows

1. Click in the required input field in order to place the cursor there.
2. Click the input field.
   The virtual keyboard is displayed.
3 Enter your values via the virtual keyboard.
4. Press the "Input" key.

- OR -

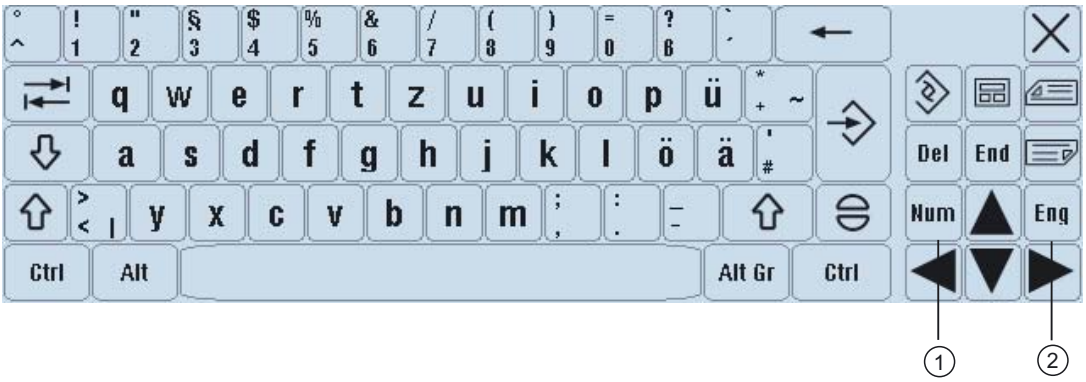
Position the cursor on an another operator element.

The value is accepted and the virtual keyboard is closed.

Positioning of the virtual keyboard

You can position the virtual keyboard anywhere in the window by depressing the empty bar next to the "Close window" symbol with your finger or a stylus and moving it back and forth.

Special keys on the virtual keyboard



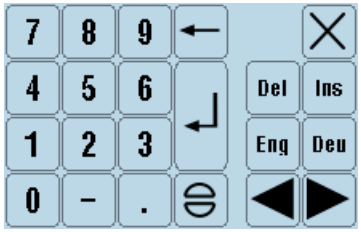
1 Num:

Reduces the virtual keyboard to the number block.

2 Eng:

Toggles the keyboard assignment between the English keyboard assignment and the keyboard assignment for the current language setting.

Number block of the virtual keyboard



Use the "Deu" or "Eng" keys to return to the full keyboard with the English keyboard assignment or the keyboard assignment of the current language setting.

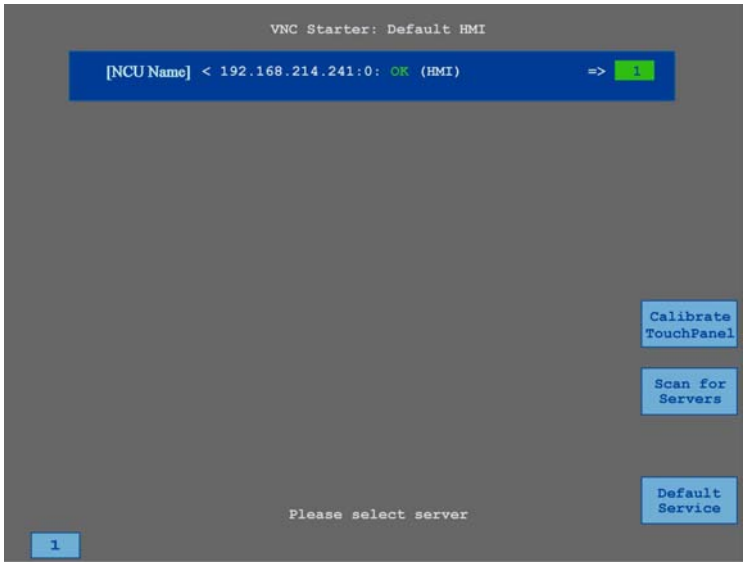
9.4 Calibrate touch panel

It is necessary to calibrate the touch panel upon first connection to the control.

Note

Recalibration

If the operation is not exact, then redo the calibration.



Proceed as follows


...

1. Press the "Back" key and the "Menu Select" key at the same time to start the TCU service screen.
2. Touch the "Calibrate TouchPanel" button.
   The calibration process will be started.
3. Follow the instructions on the screen and touch the three calibration points one after the other.
   The calibration process has terminated.
4. Touch the horizontal softkey "1" or the key with the number "1" to close the TCU service screen.

Alarm, error, and system messages 10

10.1 Displaying alarms

If faulty conditions are recognized in the operation of the machine, then an alarm will be generated and, if necessary, the machining will be interrupted.

The error text that is displayed together with the alarm number gives you more detailed information on the error cause.

Warning

Please check the situation in the plant on the basis of the description of the active alarm(s). Eliminate the cause/s of the alarm/s and acknowledge it/them as instructed.

Failure to observe this warning will place your machine, workpiece, stored settings and possibly even your own safety at risk.

Alarm overview

You can display all upcoming alarms and acknowledge them.

The alarm overview contains the following information:

- Date and time
- Clearing criterion

  specifies the key or softkey used to acknowledge the alarm
- Alarm number
- Alarm text

Proceed as follows

1. Select the "Diagnosis" operating area.

2. Press the "Alarm list" softkey.
   The "Alarms" window appears.

3. Position the cursor on an alarm.

4. Press the "Delete HMI alarm" softkey to cancel the alarm.

   - OR -

   Press the "Acknowl. alarm" softkey for alarms that require acknowledgement.

Acknowledgement symbols

Icon	Description
	Turn the unit off and back on (main switch), or press NCK POWER ON.
	Press the "Reset" key.
	Press the "ALARM CANCEL" key - OR - Press the "Acknowl. HMI alarm" key.
	Press the key provided by the manufacturer.

Machine manufacturer

Please also refer to the machine manufacturer's instructions.

10.2 Displaying Alarms

PLC and part program messages may be issued during machining.

These message will not interrupt the program execution. Messages provide information with regard to a certain behavior of the cycles and with regard to the progress of machining and are usually kept beyond a machining step or until the end of the cycle.

Overview of messages

You can display all issued messages.

The message overview contains the following information:

- Date
- Message number
  is only displayed for PLC messages
- Message text

Proceed as follows

1. Select the "Diagnosis" operating area.

2. Press the "Messages" softkey.
   The "Messages" window appears.

10.3 Displaying and modifying PLC variables

You can monitor and modify PLC memory cells in the "PLC status" window.

You receive information in the list on operands, with their format and status value.

PLC operands	
Inputs	Input bit (Ex), input byte (EBx), input word (EWx), input double word (EDx)
Outputs	Output bit (Ax), output byte (ABx), output word (AWx), output double word (ADx)
Bit memory	Memory bit (Mx), memory byte (MBx), memory word (MWx), memory double word (MDx)
Times	Time (Tx)
Meters	Counter (Cx)
data	Data block (DBx), Data bit (DBXx), data byte (DBBx), data word (DBWx), data double word (DBDx)

Formats	
B	Binary
H	hex
D	dec
G	Floating comma (for double words)
C	Character (ASCII character)

Examples

EB2

MW20

DB2.DBB180

Changing PLC operands

Changes can only be made to the PLC operands with the appropriate password.

Danger

Changes in the states of PLC memory locations have a major impact on the machine. Incorrect configuration of the parameters can endanger human life and cause damage to the machine.

Changing and deleting values

1. Select the "Diagnosis" operating area.

2. Press the "Variab. view" softkey.

   The "Variables" window opens.

3. Position the cursor in the "Operand" column and enter the required variable.

4. Press the "Input" key.
   The operand is displayed with the value.

5. Place the cursor in the "Format" field and choose the desired format with "Select".

4. Press the "Change" softkey if you would like to edit the value.

   - OR -

   Press the "Delete" softkey if you would like to delete the entries for the operands.

5. Press the "Accept" softkey to confirm the changes or the deletion.

   - OR -

   Press the "Cancel" softkey to cancel the changes.

Changing the operand address

You can increase or decrease the address by 1 place at a time with the "Operand +" and "Operand -" softkeys.

...

You can increase or decrease the address by 1 place at a time with the "Operand +" and "Operand -" softkeys.

These operand settings are retained after the control is switched on and off.

10.4 Axis diagnosis

The information in the "Service axis" window is used to

- check the setpoint branch (e.g., position setpoint, speed setpoint, spindle speed setpoint prog.)
- check the actual-value branch (e.g., actual position value, measuring system ½, actual speed value), optimize the position control loop of the axis (e.g., following error, control difference, servo gain factor)
- check the entire control loop of the axis (e.g., through position setpoint/actual-value comparison and speed setpoint/actual-value comparison)
- check hardware faults (e.g. encoder check: if the axis is moved mechanically, the actual position value must change)
- set and check axis monitoring functions.

For additional information, please refer to the following documentation:

CNC Commissioning Part 1 (NCK, PLC, Drive) / SINUMERIK 840D sl/840D/840Di/810D

Proceed as follows

1. Select the "Diagnosis" operating area.

2. Press the "Axis diag." softkey.
   The "Axis diagnosis" window is opened.

3. Press the "Service axis" softkey.

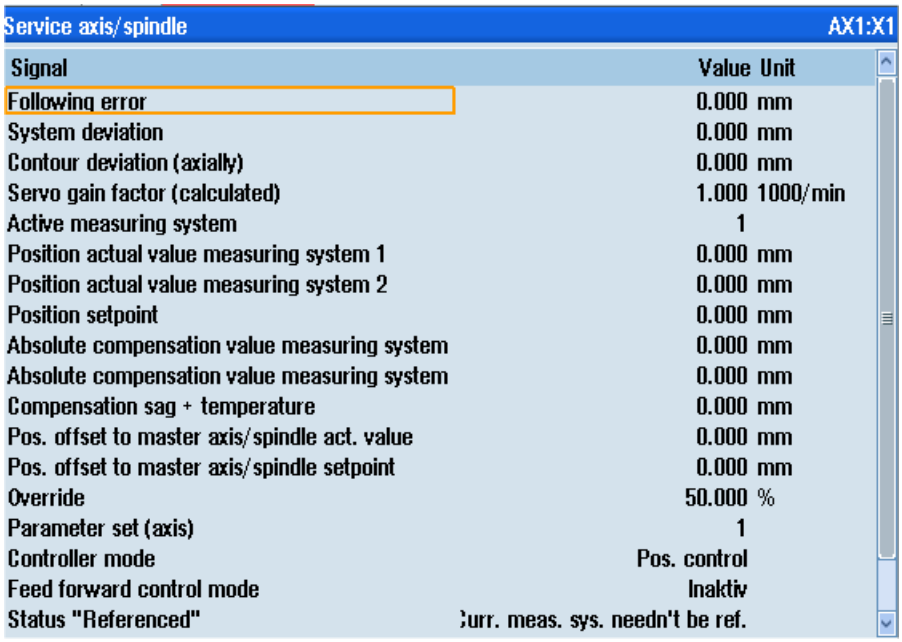


Figure 10-1 "Axis diagnosis" window with information about the machine axes

10.5 System utilization

For the NC areas you can display the system resources (utilization display) currently being used: This shows you the net and gross runtimes for the position controller, interpolator, and forward motion.

Proceed as follows

1. Select the "Diagnosis" operating area.

2. Press the ETC key.
   New horizontal softkeys are displayed.

3. Press the "Syst. utiliz." softkey.
   The "System utilization" window appears.
   You can track the dynamic utilization display.

4. Press the "Stop" softkey to stop the display update.

5. Press the "Start" softkey to refresh the values.

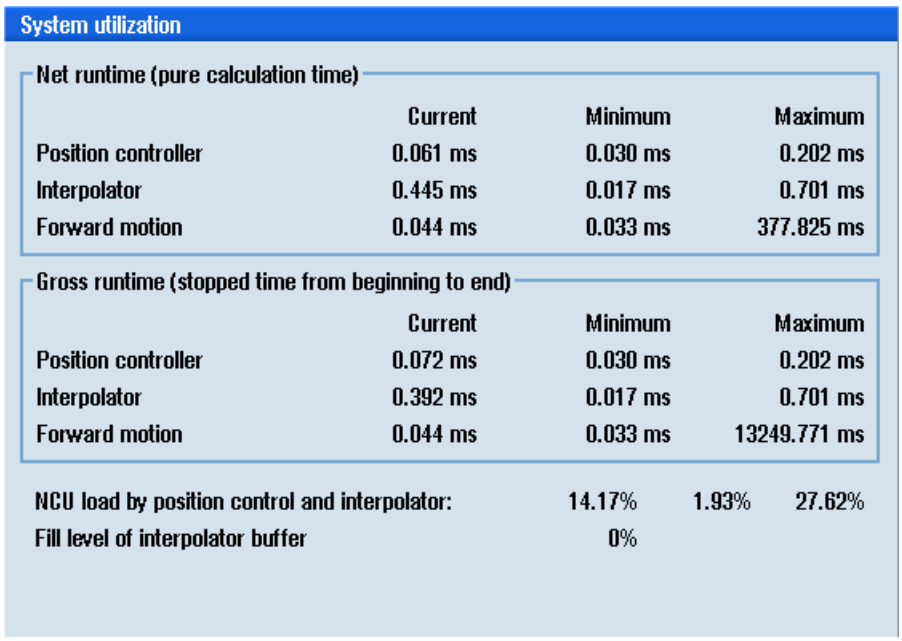


Figure 10-2 "System utilization" window with system resources display

10.6 Displaying version data

All system software components are provided with their corresponding version data in the "Version data" window.

You may save the version data. Version displays saved as text files can be further processed as required or sent to the hotline in the event of an error.

Proceed as follows

1. Select the "Diagnostics" operating area.

2. Press the "Version" softkey.
   The "Version data" window appears.
   Data from the available components are displayed.

3. Press the "Details" softkey, in order to receive more exact information on the components displayed.

Appendix

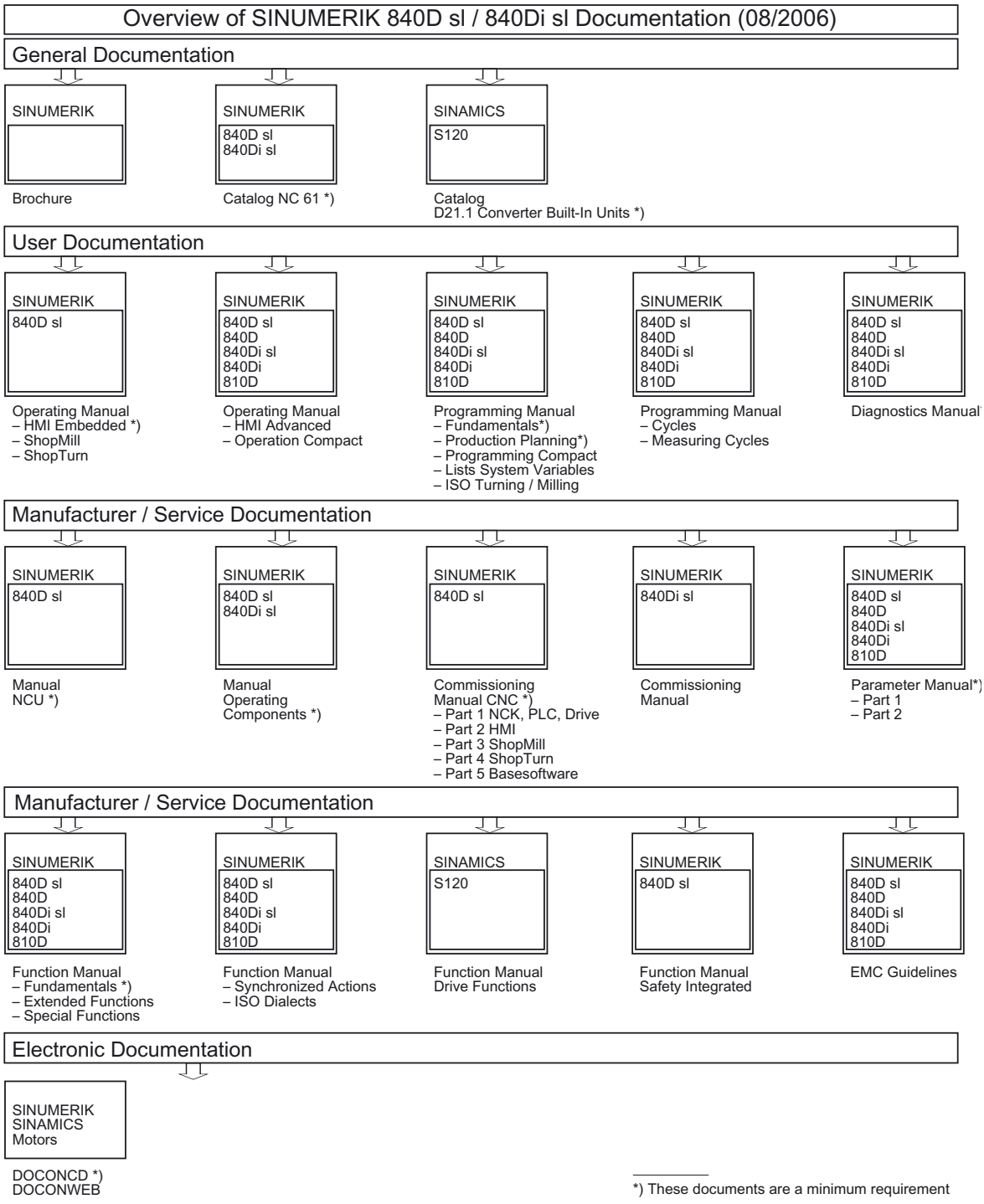
A

A.1 Correction sheet - fax template

To SIEMENS AG A&D MC MS1 P.O. Box 3180 91050 Erlangen / Germany Phone: +49-(0)180-5050-222 (Service Support) Fax.: +49-(0)9131-98-63315 (Documentation) mailto: motioncontrol.docu@siemens.com	From Name:
	Address of your company / department
	Address:
	Postal code: City:
	Phone: /
	Fax: /

Suggestions and / or corrections

A.2 Overview


Overview of SINUMERIK 840D sl / 840Di sl Documentation (08/2006)
General Documentation
SINUMERIK
Brochure
SINUMERIK
840D sl
840Di sl
Catalog NC 61 *)
SINAMICS
S120
Catalog
D21.1 Converter Built-In Units *)
User Documentation
SINUMERIK
840D sl
Operating Manual
– HMI Embedded *)
– ShopMill
– ShopTurn
SINUMERIK
840D sl
840D
840Di sl
840Di
810D
Operating Manual
– HMI Advanced
– Operation Compact
SINUMERIK
840D sl
840D
840Di sl
840Di
810D
Programming Manual
– Fundamentals*)
– Production Planning*)
– Programming Compact
– Lists System Variables
– ISO Turning / Milling
SINUMERIK
840D sl
840D
840Di sl
840Di
810D
Programming Manual
– Cycles
– Measuring Cycles
SINUMERIK
840D sl
840D
840Di sl
840Di
810D
Diagnostics Manual
Manufacturer / Service Documentation
SINUMERIK
840D sl
Manual
NCU *)
SINUMERIK
840D sl
840Di sl
Manual
Operating
Components *)
SINUMERIK
840D sl
Commissioning
Manual CNC *)
– Part 1 NCK, PLC, Drive
– Part 2 HMI
– Part 3 ShopMill
– Part 4 ShopTurn
– Part 5 Basesoftware
SINUMERIK
840Di sl
Commissioning
Manual
SINUMERIK
840D sl
840D
840Di sl
840Di
810D
Parameter Manual*)
– Part 1
– Part 2
Manufacturer / Service Documentation
SINUMERIK
840D sl
840D
840Di sl
840Di
810D
Function Manual
– Fundamentals *)
– Extended Functions
– Special Functions
SINUMERIK
840D sl
840D
840Di sl
840Di
810D
Function Manual
– Synchronized Actions
– ISO Dialects
SINAMICS
S120
Function Manual
Drive Functions
SINUMERIK
840D sl
Function Manual
Safety Integrated
SINUMERIK
840D sl
840D
840Di sl
840Di
810D
EMC Guidelines
Electronic Documentation
SINUMERIK
SINAMICS
Motors
DOCONCD *)
DOCONWEB
*) These documents are a minimum requirement

Index

S

T

U

V

W