SIEMENS

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	2.5
with HMI sl	2.5

01/2008
6FC5398-6AP10-2BA0

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
Industry Sector
Postfach 48 48
90327 NÜRNBERG
GERMANY

Ordernumber: 6FC5398-6AP10-2BA0
℗ 07/2008

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" → "Order Documentation" → "Printed Documentation".

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.

Benefits

The Operating Manual familiarizes users with the control elements and commands. It enables the users to respond to problems and to take corrective action.

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

If you have any technical questions, please contact our hotline:

	Europe / Africa
Phone	+49 180 5050 222
Fax	+49 180 5050 223
Internet	http://www.siemens.com/automation/support-request

	Americas
Phone	+1 423 262 2522
Fax	+1 423 262 2200
E-mail	mailto:techsupport.sea@siemens.com

	Asia / Pacific
Phone	+86 1064 719 990
Fax	+86 1064 747 474
E-mail	mailto:adsupport.asia@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

Calls are subject to charge, e.g. 0.14 €/min. on the German landline network. Tariffs of other phone companies may differ.

Questions about the Operating Manual

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

Fax	+49 9131- 98 63315
E-mail	mailto:docu.motioncontrol@siemens.com

A fax form is available in the appendix of this document.

SINUMERIK Internet address

http://www.siemens.com/motioncontrol

Table of contents

Introduction 1

1.1 Product overview

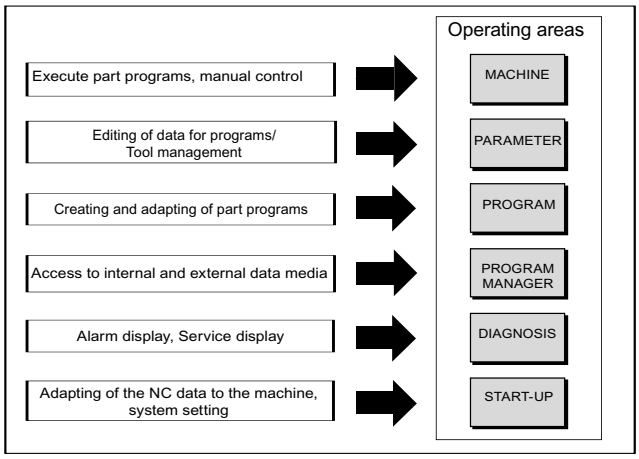
The SINUMERIK controller is a CNC (Computerized Numerical Controller) for machine tools.

You can use the CNC 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
- Management of tools, zero points and further user data required in programs
- Diagnostics of controller and machine

Operating areas

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



1.2 Operator panel fronts

1.2.1 Overview

Introduction

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

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

Operator controls and indicators

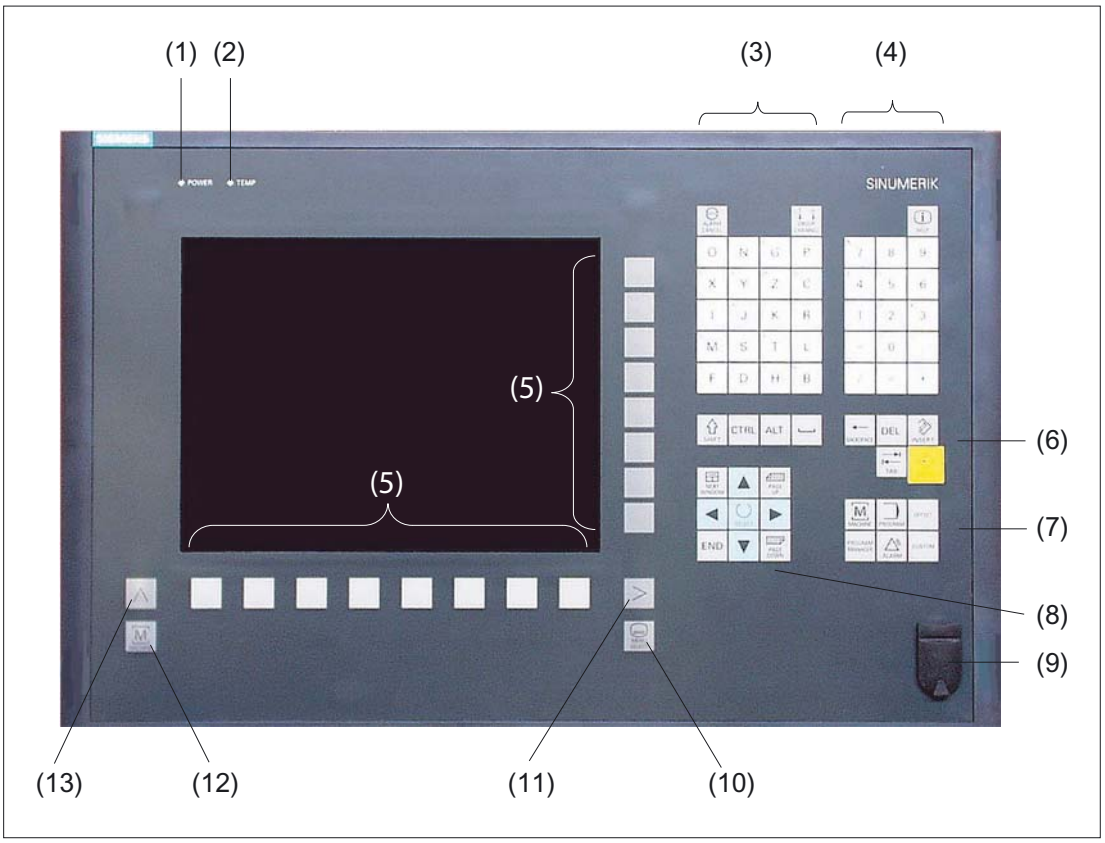


Figure 1-1 View of operator OP 010 panel front

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 key

References

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

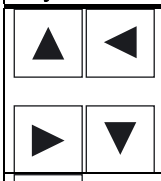
1.2.2 Keys of the operator panel

The following buttons are available for operation of the controller 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** Calls the context-sensitive online help for the selected window.
	**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** 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** Delete the value in the input field. In insertion mode, it deletes the character after the cursor.
	**TAB** Indent the cursor by several characters.
	**SHIFT** Press 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** Activate insertion mode.
	**INPUT** • Finish the entry of a value in the input field. • Open a directory or program.
	**ALARM - only OP 010 and OP 010C** Irrelevant
	**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
	**Menu back key** Jump to the next highest menu level.

Key	Function
	**MACHINE** Open the "Machine" operating area.
	**Menu forward key** 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

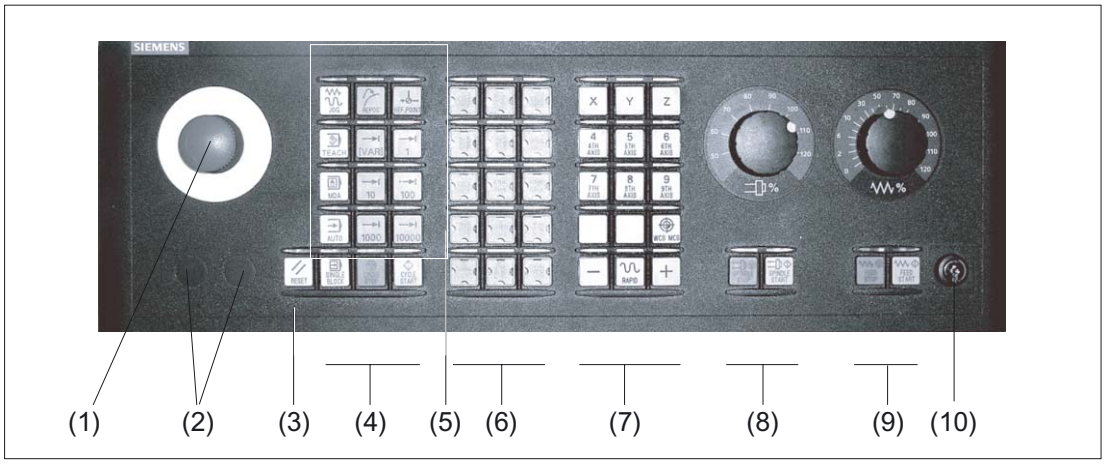
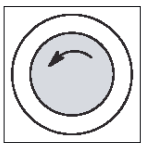


Figure 1-2 Front view of machine control panel (milling version)

(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

- 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

The key is also referred to as NC Start.

Execution of a program is started.

CYCLE STOP

The key is also referred to as NC Stop.

Execution of a program is stopped.

(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.

Direction 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 Screen layout

Overview

Figure 1-3 User 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
- active spindle with current status (S)

8 Operating window with program block display

9 Display of active G functions, all G functions, H functions and input window for different functions (for example, skip blocks, 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.

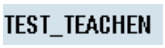
Status display of "Machine" operating area

First line

Display	Description
Active operating area	
	"Machine" operating area With touch operation, you can change the operating area here.
	"Parameter" operating area
	"Program" operating area
	"Program manager" operating area
	"Diagnosis" operating area
	"Start-up" operating area
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

The displays in the second line can be configured.

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

Third line

Display	Description
	Display of channel status. If several channels are present on the machine, the channel name is also displayed. 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 started. The program has been interrupted with "Stop".

Display	Description
DRYPRT	Display of active program controls: PRT: no axis motion DRY: Dry run feedrate M01: programmed stop 1 M101: programmed step 2 (name varies) SB1: Single block, coarse (program stops only after blocks which perform a machine function) SB2: Data block (program stops after each block) SB3: Single block, fine (program also only stops after blocks which perform a machine function in cycles)
Faulty NC block / user alarm Remaining dwell time:15 Sec.	Channel operational messages: Stop: An operator action is usually 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 45)

Touch operation (Page 32)

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 work offsets into consideration.

You can use the "MCS actual values" softkey to toggle between the machine coordinate system and the workpiece coordinate system.

The actual value display of the positions can also refer to the SZS coordinate system (settable zero system). However the positions are still output in the WCS.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

Maximize display

Press the ">>" and "Zoom act.val." softkeys.

Overview of display

Display	Meaning
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 feed acting on the axes, as well as the override, are displayed in the full-screen version.
Repos offset	The distances traversed in manual mode are displayed. This information is only displayed when you are in the "Repos" submode.
Footer	Display of active work offsets and transformations. The T, F, S values are also displayed in the full-screen version.

See also

Overview - Work offsets (Page 50)

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

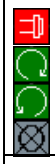
Display	Meaning
T	
Tool name	Name of current tool
Location	Location number of current tool
D	Cutting edge of the current tool In DIN-ISO mode the H number is displayed instead of the cutting edge number.
H	H number (tool offset data record for DIN-ISO mode) If there is a valid D number, this is also displayed.
Ø	Diameter of current tool

Feed data

Display	Meaning
F	
	Feed disable
	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	Meaning
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)

Display	Meaning
Icon 	Spindle status Spindle not enabled Spindle is turning clockwise Spindle is turning counterclockwise Spindle is stationary
Override	Display as a percentage
Spindle utilization rate	Display between 0 and 100% The upper limit value can be greater than 100% See machine manufacturer's specifications.

1.4.5 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.
- The program block which is just being processed appears colored.

Editing a program directly

In the Reset state, you can edit the current program directly.

1. Press the "INSERT" key.
2. Place the cursor at the relevant position and edit the program block.
   Direct editing is only possible for part programs in the NC memory, not for external execution.

3. Press the "INSERT" key to exit the program and the editing mode again.

See also

Correcting a program (Page 81)

1.4.6 Operation via softkeys and buttons

Operating areas/operating modes

The user interface consists of different windows featuring eight horizontal and eight 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 six operating areas (machine, parameter, program, program manager, diagnosis, startup) and five operating modes or submodes (JOG, MDA, AUTO, TEACH In, REF POINT, REPOS).

Changing the 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.

Select the "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 user interface, you can change the horizontal softkey bar within an operating area. To do so, press the menu forward key.

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

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

Use the ">>" softkey to open a new vertical softkey bar.

Use the "<<" softkey to return to the previous vertical softkey bar.

Use this softkey to close the open 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 15)

Channel switchover (Page 45)

1.4.7 Entering or selecting parameters

When setting up the machine and during programming, you must enter values for various parameters in the input fields. The background color of the fields provides information on the status of the input field.

Orange background	Input field is selected
Light orange background	Input field is in Edit mode
Pink background	Entered value is incorrect

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.

The selection symbol is displayed in the tooltip: 

Associated selection fields

There are selection fields for various parameters:

- Selection of units
- Changeover between absolute and incremental dimensions

Procedure

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.

   - OR -

   Press the "INSERT" key.

   The selection options are displayed in a list.

2. Select the desired setting with the "Cursor down" and "Cursor up" keys.

3. If required, enter a value in the associated input field.
4. 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" softkey.

- OR -

Press the "Left cursor" key.

1.4.8 Pocket calculator

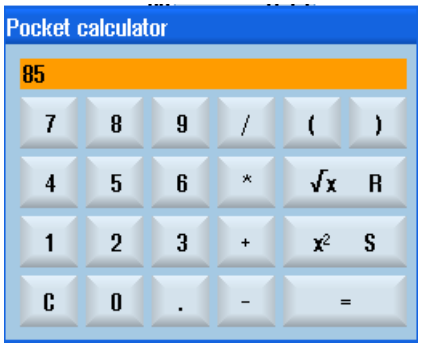
You can use the pocket calculator to quickly calculate parameter values during programming. If, for example, the diameter of a workpiece is only dimensioned indirectly in the workpiece drawing, i.e., the diameter must be derived from the sum of several other dimension specifications, you can calculate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

Calculation methods

The following arithmetic operations are available:

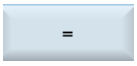
- Addition
- Subtraction
- Multiplication
- Division
- Calculation with parentheses
- Square root of x
- x squared

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" softkey.

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.

See also

Entering or selecting parameters (Page 27)

1.4.9 Context menu

When you right-click, the context menu opens and provides the following functions:

- Cut Ctrl+X
- Copy Ctrl+C
- Paste Ctrl+V

1.4.10 Touch operation

If you have an operator panel with a touch screen, you can perform the following functions with touch operation:

Operating area switchover


...

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

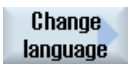
Channel switchover

You can switch over to the next channel by touching the channel display in the status display.

1.4.11 Changing the user interface language

Procedure

1. Select the "Startup" operating area.



2. Press the "Change language" softkey.
   The "Language selection" window opens. The language set last is selected.
3. Position the cursor on the desired language.

4. Press the "OK" softkey.

- OR -

Press the <INPUT> key.

The user interface changes to the selected language.

Note

Changing the language directly on the input screens

You can switch between the user interface languages available on the controller directly on the user interface by pressing the key combination <CTRL + L>.

1.4.12 Editing Asian characters

You can edit Asian characters.

You can select a character by using the Pinyin phonetic notation, which enables Chinese characters to be expressed by combining Latin letters.

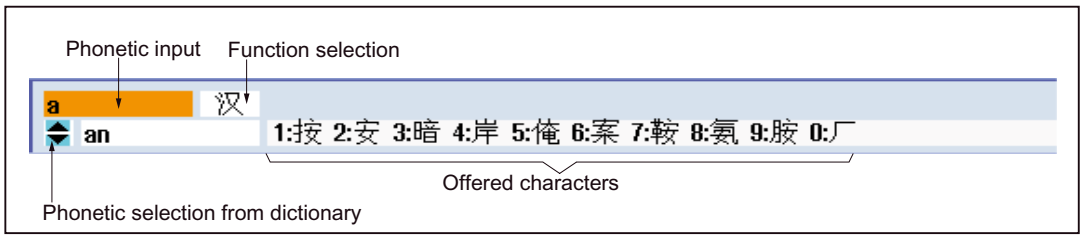
The editor is available for the following Asian languages:

- Simplified Chinese
- Traditional Chinese
- Korean

Note

You require a special keyboard to enter Korean characters.

Structure of editor



Functions

汉	Pinyin input
新词	Editing of the dictionary
A	Input of Latin letters

Requirement

The control has been set to Chinese or Korean.

Procedure

Editing characters

1. Open the screen form and position the cursor on the entry field and press the <Alt +S> keys.
   The editor is displayed.
2. Enter the desired phonetic notation.

3. Click the "Cursor down" button to access the dictionary.
4. Repeated clicking of the "Cursor down" button displays all the entered phonetic notations and the associated selection characters.
5. Press the "BACKSPACE" softkey to delete entered phonetic notations.
6. Press the number key to insert the associated character.
   When a character is selected, the editor records the frequency with which it is selected for a specific phonetic notation and offers this character at the top of the list when the editor is next opened.

Editing the dictionary

1. Select the dictionary editing function in the selection box.
   The editor provides a further line in which the combined characters and phonetic notations are displayed.
2. Enter the desired phonetic notation in the phonetic input field.
   Various characters are displayed for this phonetic notation, from which you can select a character by entering either of the appropriate number (1 to 9).
   You can toggle the input cursor between the compound phonetic notations field and the phonetic input field by pressing the <TAB> key.
   Combined characters are cancelled via the <BACKSPACE> key.

3. Press the <SELECT> key to take a combined phonetic notation into the dictionary.

1.4.13 Protection levels

The input and modification of data in the control system is protected by passwords at sensitive places.

Access protection via protection levels

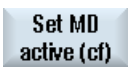
The input or modification of data for the following functions depends on the protection level setting:

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

For additional information, please refer to the following documentation:

HMI sl / SINUMERIK 840D sl Commissioning Manual

Softkeys

Startup operating area	Protection levels
	End user (protection level 3)
	Keyswitch 3 (protection level 4)
	Keyswitch 0 (protection level 7)
	Keyswitch 3 (protection level 4)
	Keyswitch 3 (protection level 4)
	Keyswitch 3 (protection level 4)
	Service (protection level 2)

Startup operating area	Protection levels
Change password	End user (protection level 3)
Delete password	End user (protection level 3)

1.4.14 Online help in HMI sl

A comprehensive context-sensitive online help is stored in the control system.

- A brief description is provided for each window and, if required, step-by-step instructions for the operating sequences.
- A detailed help is provided in the editor for every entered G code. You can also display all G functions and take over a selected command directly from the help into the editor.
- A help page with all parameters is provided on the input screen in the cycle programming.
- Lists of the machine data
- Lists of the setting data
- Lists of the drive parameters
- List of all alarms

Procedure

Calling context-sensitive online help

1. You are in an arbitrary window of an operating area.
2. Press the <Help> key or on an MF2 keyboard, the <F12> key.

   The help page of the currently selected window is opened in a subscreen.

3. Press the "Full screen" softkey to use the entire user interface for the display of the online help.

   Press the "Full screen" softkey again to return to the subscreen.

4. If further helps are offered for the function or associated topics, position the cursor on the desired link and press the "Follow reference" softkey.

   The selected help page is displayed.
5. Press the "Back to reference" softkey to jump back to the previous help.

Calling a topic in the table of contents

1. Press the "Table of contents" softkey.
   Depending on which technology you are using, the "HMI sl Milling", "HMI sl Turning" or "HMI sl Universal" Operating and Commissioning Manuals as well as the "Programming" Manual are displayed.

2. Select the desired manual with the "Cursor down" and "Cursor up" keys.

3. Press the "Cursor right" or "Input" key or double-click to open the book and the section.

4. Navigate to the desired topic with the "Cursor down" key.

5. Press the "Follow reference" softkey or the <Input> key to display the help page for the selected topic.

6. Press the "Current topic" softkey to return to the original help.

Searching for a topic

1. Press the "Search" softkey.
   The "Search in Help for: " window appears.
2. Activate the "Full text " checkbox to search in all help pages.
   If the checkbox is not activated, a search is performed in the table of contents and in the index.

3. Enter the desired keyword in the "Text" field and press the "OK" softkey.
   If you enter the search term on the machine control panel, replace an accented character with an asterisk (*) as dummy.
   All entered terms and sentences are sought with an AND operation. In this way, only documents and entries that satisfy all the search criteria are displayed.

4. Press the "Keyword index" softkey if you only want to display the index of the operating and programming manual.

Displaying alarm descriptions and machine data

1. If messages or alarms are pending in the "Alarms", "Messages" or "Alarm Log" window, position the cursor at the appropriate display and press the <Help> or the <F12> key.

   The associated alarm description is displayed.

2. If you are in the "Startup" operating area in the windows for the display of the machine, setting and drive data, position the cursor on the desired machine data or drive parameter and press the <Help> or the <F12> key.

   The associated data description is displayed.

Displaying and inserting a G code command in the editor

1. A program is opened in the editor.

   Position the cursor on the desired G code command and press the <Help> or the <F12> key.

   The associated G code description is displayed.

2. Press the "Display all G funct." softkey.

3. With the aid of the search function, select, for example, the desired G code command.

4. Press the "Transfer to editor" softkey.

   The selected G function is taken into the program at the cursor position.

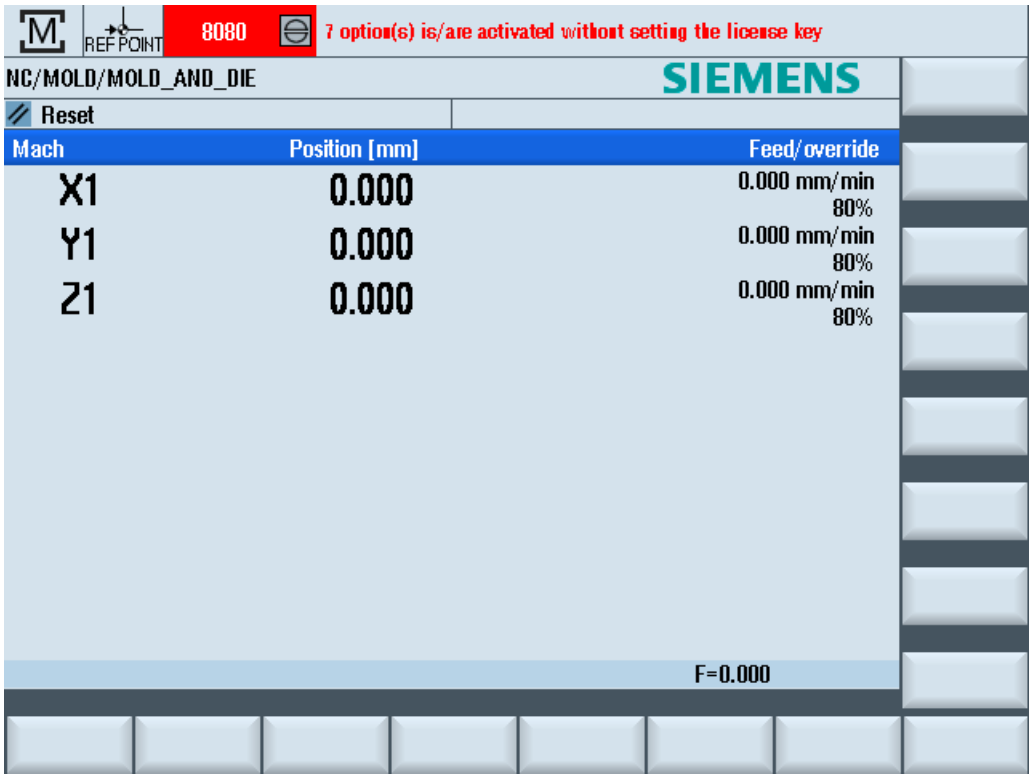
5. Press the "Exit help" softkey again to close the help.

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 Approaching a reference point

2.2.1 Referencing 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.

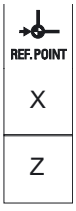
Machine manufacturer

Please refer to the machine manufacturer's specifications.

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!

Procedure

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".

2.2.2 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 acknowledgement for an axis after it has approached the reference point.

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

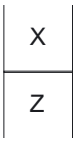
Option

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

Procedure

1. Select the "Machine" operating area.

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 and stops. The coordinate of the reference point is displayed.
   The axis is marked with ⊕.

5. Press the "User enable" softkey.
   The "User Acknowledge" window opens.

   It shows a list of all machine axes with their current position and SI position.
5. Position the cursor in the "Acknowledgement" field for the axis in question.

6. Activate the acknowledgement with the "SELECT" key.

   The selected axis is marked with an "x" meaning "safely referenced" in the "Acknowledgement" column.

   By pressing the "SELECT" key, you deactivate the acknowledgement again.

2.3 Modes of operation

2.3.1 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.

Selecting "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).

Selecting "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.

Selecting "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.

"TEACH IN" submode

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.

Selecting "Teach In"

Press the "TEACH IN" key.

2.3.2 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.

Changing the channel

Press the "CHANNEL" key.

The channel changes over to the next channel or a channel menu appears.

Channel switchover via touch operation

On the HT 8 and when using a touch screen operating panel, you can switch to the next channel via touch operation in the status display.

See also

HT 8 overview (Page 201)

2.4 Settings for the machine

2.4.1 Switching over the 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.

2.4.2 Switching the unit of measurement

You can set millimeters or inches as the unit of measurement. Switching the unit of measurement always applies to the entire machine. All required information is automatically converted to the new unit of measurement, for example:

- Positions
- Tool offsets
- Work offsets

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

Proceed as follows

1. Select "JOG" or "AUTO" mode in the "Machine" operating area.

2. Press the menu forward key and the "Settings" softkey.
   A new vertical softkey bar appears.

3. Press the "Switch to inch" softkey.
   A prompt asks you whether you really want to switch over the unit of measurement.

4. Press the "OK" softkey.

   The softkey label changes to "Switch to metric".
   The unit of measurement applies to the entire machine.

5. Press the "Switch to metric" softkey to set the unit of measurement of the machine to metric again.

2.4.3 Setting the work offset

You can enter a new position value in the actual value display for individual axes when a settable work offset is active.

The difference between the position value in the machine coordinate system MCS and the new position value in the workpiece coordinate system WCS is saved permanently in the currently active work offset (e.g. G54).

Requirement

The control is in the workpiece coordinate system.

The actual value can be set in both the Reset and Stop state.

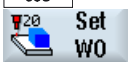
Note

Setting the WO in the Stop state

If you enter the new actual value in the Stop state, the changes made are only visible and only take effect when the program is continued.

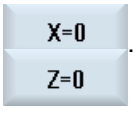
Procedure

1. Select "JOG" mode in the "Machine" operating area.



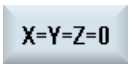
2. Press the "Set WO" softkey.

3. Enter the new position value for X, Y or Z directly in the actual value display (you can toggle between the axes with the cursor keys) and press the "Input" key to confirm the entries.



- OR -

Press softkeys "X=0","Y=0" or "Z=0" to set the relevant position to zero.



- OR -

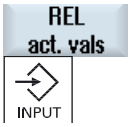
Press softkey "X=Y=Z=0" to set all axis positions to zero simultaneously.

Resetting the actual value

Press the "Delete active WO" softkey.
The offset is deleted permanently.

NOTICE
Irreversible active work offset
The current active work offset is irreversibly deleted by this action.

Relative actual value



1. Press the "REL actual values" softkey.
2. Enter the axis positions and press the "Input" key.

Note

The new actual value is only displayed. The relative actual value has no effect on the axis positions and the active work offset.

The softkey is only available if the relevant machine data is set.

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

2.5 Work offsets

2.5.1 Overview - Work offsets

Following reference point approach, the actual value display for the axis coordinates is based on the machine zero (M) of the machine coordinate system (MCS). The program for machining the workpiece, however, is based on the workpiece zero (W) of the workpiece coordinate system (WCS). The machine zero and workpiece zero are not necessarily identical. The distance between the machine zero and the workpiece zero depends on the workpiece type and how it is clamped. This work offset is taken into account during execution of the program and can be a combination of different offsets.

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 actual value display of the positions can also refer to the SZS coordinate system (settable zero system). The position of the active tool relative to the workpiece zero is displayed.

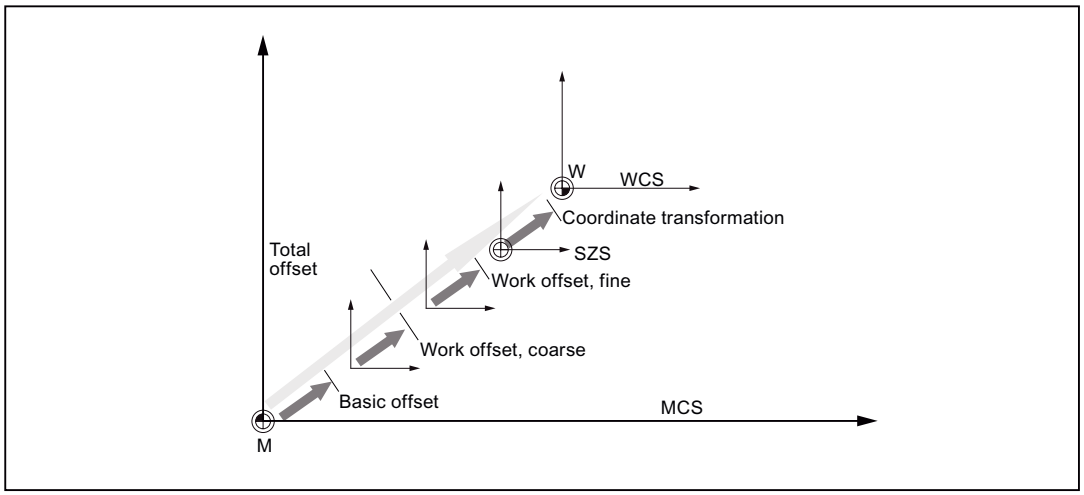


Figure 2-1 Work offsets

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 "Work offset - Base" window.

Coarse and fine offsets

Every work offset (G54 to G57, G505 to G599) consists of a coarse offset and a fine offset. You can call the work 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 "Work Offset - Active" window.

This window is generally used only for monitoring.

Display of active work offsets

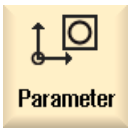
Work offsets	
Basic reference	Display of active system offsets. You can alter the data under certain circumstances, i.e. you can correct a zero point. Access to the system offsets is protected via a keyswitch.
Total base WO	Display of all active base offsets, as well as rotation, scaling and mirroring. You cannot edit these values here.
G500	Display of all work offsets activated with G54 - G599, as well as rotation, scaling and mirroring. You cannot edit these values here.
Programmed WO	Display of all additional work offsets programmed with $P_PFRAME= as well as rotation, scaling and mirroring.
Cycle reference	Display of all additional work offsets activated via $MC_MM_SYSTEM_FRAME_MASK.
Total WO	Display of the active work offset, representing the sum of all work offsets, as well as rotation, scaling and mirroring.

The availability of the offsets depends on the setting.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Parameter" operating area.

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

Note

Further details on work 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.

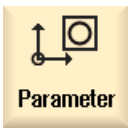
2.5.3 Displaying and editing base zero offset

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.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure



1. Select the "Parameter" operating area.

2. Press the "Work offset" softkey.

3. Press the "Base" softkey.
   The "Work 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 "Work Offset - G54..G599" window.

The currently active work offsets are displayed on a green background.

Rotation, scaling and mirroring are displayed.

Procedure

1. Select the "Parameter" operating area.

2. Press the "Work offset" softkey.

3. Press the "G54…G599" softkey.
   The "Work Offset - G54..G599" window is opened.

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

Note

Activate settable work offsets

The settable work 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 work offset, you can display and edit all data for all axes. You can also delete work offsets.

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

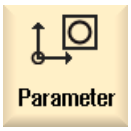
- Coarse and fine offsets
- Rotation
- Scaling
- Mirroring

Machine manufacturer
Please refer to the machine manufacturer's specifications.

Note

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

Procedure



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 work offset to view its details.

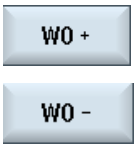
5. Press the "Details" softkey.

   A window opens, depending on the selected work offset, e.g. "Work 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.



Press the "WO +" or "WO -" softkey to select the next or previous offset, respectively, within the selected area ("Active", "Base", "G54 to G599") without first having to switch to the overview window.

If you have reached the end of the range (e.g. G599), you will switch automatically to the beginning of the range (e.g. G54).

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

Machine manufacturer

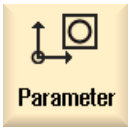
Please refer to the machine manufacturer's specifications.

Press the "Back" softkey to close the window.

2.5.6 Deleting a work offset

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 Monitoring axis and spindle data

2.6.1 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.

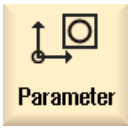
In this way, you are able to restrict the traversing range of the axes in addition to the limit switches.

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.

2.6.2 Editing spindle data

The speed limits set for the spindles that must not be under- or overshot are displayed in the "Spindles" window.

You can limit the spindle speeds in fields "Minimum" and "Maximum" within the limit values defined in the relevant machine data.

Spindle speed limitation at constant cutting rate

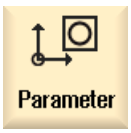
In field "Spindle speed limitation at G96", the programmed spindle speed limitation at constant cutting speed is displayed together with the permanently active limitations.

This speed limitation, for example, prevents the spindle from accelerating to the max. spindle speed of the current gear stage (G96) when performing tapping operations or machining very small diameters.

Note

The "Spindle data" softkey only appears if a spindle is configured.

Procedure



1. Select the "Parameter" operating area.

2. Press softkeys "Setting data" and "Spindle data".
   The "Spindles" window opens.

3. If you want to change the spindle speed, place the cursor on the "Maximum", "Minimum", or "Spindle speed limitation at G96" and enter a new value.

2.7 Manual mode

2.7.1 General

Always use "JOG" mode when you want to set up the machine for the execution of a program or to carry out simple traversing movements on the machine:

- Synchronize the measuring system of the controller with the machine (reference point approach)
- Set up the machine, i.e. activate manually-controlled motions on the machine using the keys and handwheels provided on the machine control panel.
- You can activate manually controlled motions on the machine using the keys and handwheels provided on the machine control panel while a part program is interrupted.

2.7.2 Selecting a tool and spindle

2.7.2.1 T, S, M windows

For the preparatory actions in manual mode, tool selection and spindle control are both performed centrally in a screen form.

In manual mode, you can select a tool either by its name or its location number. If you enter a number, a search is performed for a name first, followed by a location number. This means that if you enter "5", for example, and no tool with the name "5" exists, the tool is selected from location number "5".

Note

Using the location number, you can thus swing around an empty space into the machining position and then comfortably install a new tool.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Display	Meaning
T	Input of the tool (name or location number) You can select a tool from the tool list via the "Tool" softkey.
D	Cutting edge number of the tool (1 - 9)
Spindle	Spindle selection, identification with spindle number
Spindle M function	Spindle off: Spindle is stopped
	CCW rotation: Spindle turns in counterclockwise direction
	CW rotation: Spindle turns in clockwise direction
	Spindle positioning: Spindle is moved to the desired position.
Other M functions	Input of machine functions Refer to the machine manufacturer's table for the correlation between the meaning and number of the function.
G work offset	Selection of the work offset (basic reference, G54 - 57) You can select work offsets from the tool list of settable work offsets via the "Work offset" softkey.
Unit of measurement	Selection of the unit of measurement (inch, mm) The setting made here has an effect on the programming.
Machining plane	Selection of the machining plane (G17(XY), G18 (ZX), G19 (YZ))
Gear stage	Specification of the gear stage (auto, I - V)
Stop position	Input of the spindle position in degrees

Note

Spindle positioning

You can use this function to position the spindle at a specific angle, e.g. during a tool change.

- A stationary spindle is positioned via the shortest possible route.
- A rotating spindle is positioned as it continues to turn in the same direction.

2.7.2.2 Selecting a tool

Procedure

1. Select the "JOG" operating mode.

2. Press the "T, S, M" softkey.

3. Enter the name or the number of the tool T in the input field.

   - OR -

   Press the "Tool" softkey to open the tool list, position the cursor on the desired tool and press the "To Manual" softkey.

   The tool is transferred to the "T, S, M... window" and displayed in the field of tool parameter "T".

4. Select tool edge D or enter the number directly in field "D".

5. Press the "CYCLE START" key.

   The tool is loaded into the spindle.

2.7.2.3 Starting and stopping a spindle manually

Procedure

1. Select the "JOG" operating mode.

2. Press the "T, S, M" softkey.
3. Select the desired spindle (e.g. S1) and enter the desired spindle speed (rpm) in the adjacent input field.
   The spindle remains stationary.

4. If the machine has a gearbox for the spindle, set the gear stage (e.g. auto).

5. Select a spindle direction of rotation (clockwise or counterclockwise) in the "Spindle M function" field.

6. Press the "CYCLE START" key.
   The spindle rotates.

7. Select the "Stop" setting in the "Spindle M function" field.

   Press the "CYCLE START" key.
   The spindle stops.

Note

Changing the spindle speed

If you enter the speed in the "Spindle" field while the spindle is rotating, the new speed is applied.

2.7.2.4 Position spindle

Procedure

1. Select "JOG" mode.

2. Press the "T, S, M" softkey.

3. Select the "Stop Pos." setting in the "Spindle M function" field.
   The "Stop Pos." entry field appears.
4. Enter the desired spindle stop position.
   The spindle position is specified in degrees.

5. Press the "CYCLE START" key.

   The spindle is moved to the desired position.

Note

You can use this function to position the spindle at a specific angle, e.g. during a tool change.

- A stationary spindle is positioned via the shortest possible route.
- A rotating spindle is positioned as it continues to turn in the same direction.

2.7.3 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.

2.7.3.1 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

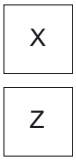
1. Select the "Machine" operating area.

2. Press the "JOG" key.

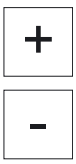
3. Press keys 1, 10, etc. up to 10000 in order to move the axis in a defined increment.

   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.

2.7.3.2 Traversing axes by a variable increment

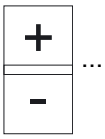
Proceed as follows

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.

2.7.4 Positioning axes

In manual mode, you can traverse individual or several axes to certain positions in order to implement simple machining sequences.

The feedrate / rapid traverse override is active during traversing.

Procedure

1. If required, select a tool.
2. Select the "JOG" operating mode.

3. Press the "Positions" softkey.
4. Specify the desired value for the feedrate F.
   - OR -

Press the "Rapid traverse" softkey.

The rapid traverse is displayed in field "F".

5. Enter the target position or target angle for the axis or axes to be traversed.

6. Press the "CYCLE START" key.

   The axis is traversed to the specified target position.

   If target positions were specified for several axes, the axes are traversed simultaneously.

2.7.5 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 the "Machine" operating area.

2. Press the "JOG" key.

3. Press the menu forward key and the "Settings" softkey.

   The "Settings for manual operation" window is opened.

2.8 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 the "Machine" operating area.

...


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


...


2. Press the menu forward key and the "Handwheel" softkey.
   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.
   Selecting an axis also activates the handwheel (e.g., "X" is assigned to handwheel no. 1 and is activated immediately).

5. Press the "Handwheel" softkey again.

   - OR -

Press the "Back" softkey.

The "Handwheel" window closes.

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 -

...

To open the "Axis" selection box using the "INSERT" key, navigate to the empty field, and press the "INPUT" key.

Clearing an axis selection also clears the handwheel selection (e.g., "X" is cleared for handwheel no. 1 and is no longer active).

2.9 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.

2.9.1 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 MDA" softkey.

   A changeover is made into the Program Manager.

   The "Load in MDA" window opens. The program manager is displayed in it.

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

5. Press the "OK" softkey.

   The window closes and the program is ready for operation.

2.9.2 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 "Store MDA" softkey.

   The "Store from MDA : Select storage location" window opens. It shows you a view of the program manager.

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

6. Press the "OK" 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.

2.9.3 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
- Testing the program
  Settings under program control
- Setting the test-run feedrate
  Settings under program control

See also

Program control (Page 90)

2.9.4 Deleting an MDA program

Requirement

The MDA editor contains a program that you created in the MDI window or loaded from the program manager.

Procedure

Press the "Delete MDI buffer" softkey.

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.

Requirements

The following requirements must be met before executing a program:

- The measuring system of the controller is referenced with the machine.
- The necessary tool offsets and work 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", "Local drive", "USB" or set-up network drives.

3. Press the "Select" 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

Starting the 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 machining

Press the "CYCLE STOP" key.
Machining stops immediately. Individual program blocks are not executed to the end. On the next start, machining is resumed from the point where it left off.

Canceling machining

Press the "RESET" key.
Execution of the program is interrupted. On the next start, machining will start from the beginning.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

See also

EXTCALL (Page 199)

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 "Select" 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 testing 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 execution of a program on the machine.

Move by single block

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

SB mode	Scope
SB1 Single block, coarse	The machining stops after every machine block (except for cycles)
SB2 Data block	The machining stops after every block, i.e. also for data blocks (except for cycles)
SB3 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.

Procedure

1. Press the "Prog. ctrl." 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 77)

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.
- The program block which is just being processed appears colored.

Editing a program directly

In the Reset state, you can edit the current program directly.

1. Press the "INSERT" key.
2. Place the cursor at the relevant position and edit the program block.

   Direct editing is only possible for part programs in the NC memory, not for external execution.

3. Press the "INSERT" key to exit the program and the editing mode again.

3.3.2 Displaying a basic block

If you want precise information about axis positions and important G functions during testing or program execution, you can call up the basic block display. This is how you can check, when using cycles, for example, whether the machine is actually traversing.

Positions programmed by means of variables or R parameters are resolved in the basic block display and replaced by the variable value.

You can use the basic block display both in test mode and when machining the workpiece on the machine. All G code commands that initiate a function on the machine are displayed in the "Basic Blocks" window for the currently active program block:

- Absolute axis positions
- G functions for the first G group
- Other modal G functions
- Other programmed addresses
- M functions

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

1. A program is selected for execution and has been opened in the "Machine" operating area.

2. Press the "Basic blocks" softkey.

   The "Basic Blocks" window opens.

3. Press the <SINGLE BLOCK> key if you wish to execute the program block-by-block.

4. Press the <CYCLE START> key to start the program execution.

   The axis positions to be approached, modal G functions, etc., are displayed in the "Basic Blocks" window for the currently active program block.

5. Press the "Basic blocks" softkey once again to hide the window again.

3.3.3 Display program level

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

Display of 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.

Procedure

Press the "Program levels" softkey.

The "Program levels" window appears.

3.4 Correcting a program

As soon as a syntax error in the part program is detected by the controller, 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 mode

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

- Reset mode

  All program lines can be edited.

Note

The "Program correction" function is only available for part programs in the NC memory, not for external execution.

Requirement

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

Procedure

1. The program to be corrected is in the Stop or Reset mode.
2. Press the "Prog. corr." softkey

   The program is opened in the editor.

   The program preprocessing and the current block are displayed. The current block is also updated in the running program, but not the displayed program section, i.e. the current block moves out of the displayed program section.

   If a subprogram is executed, it is not opened automatically.
3. Make the necessary corrections.

4. Press the "NC Execute" softkey.

   The system switches back to the "Machine" operating area and selects "AUTO" mode.

5. Press the "CYCLE START" key to resume program execution.

Note

Exit the editor using the "Close" softkey to return to the "Program manager" operating area.

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".

Resuming program execution

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

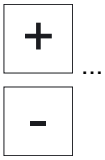
The following prerequisites must be met when repositioning the axes:

- 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 program execution
- Specify a target position, e.g. during remachining

Determining a search target

- User-friendly search target definition (search positions)
  - Direct specification of the search target by positioning the cursor in the selected program (main program)
  - Search target via text search
  - The search target is the interruption point (main program and subprogram)

    The function is only available if there is an interruption point. After a program interruption (CYCLE STOP or RESET), the controller saves the coordinates of the interruption point.
  - The search target is the higher program level of the interruption point (main program and subprogram)

    The level can only be changed if it was previously possible to select an interruption point in a subprogram. It is then possible to change the program level up to the main program level and back to the level of the interruption point.
- Search pointer
  - Direct entry of the program path

    Note

    Searching for point in subprogram

    You can search for a specific point in subprograms with the search pointer if there is no interruption point.

Cascaded search

You can start another search from the "Search target found" state. The cascading can be continued any number of times after every search target found.

Note

Another cascaded block search can be started from the stopped program execution only if the search target has been found.

References

Function Manual Basic Functions; Block Search

Requirements

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.

Toggling between search pointer and search positions

Press the "Search pointer" softkey again to exit the "Search Pointer" window and return to the "Program" window to define search positions.

- OR -

By pressing the "Back" softkey.

You have now exited the block search function.

See also

Block search mode (Page 88)

Selecting a program (Page 77)

3.6.2 Continuing program from search target

To continue the program at the desired position, press the "CYCLE START" key twice.

- The first CYCLE START outputs the auxiliary functions collected during the search. The program is then in the Stop state.
- Before the second CYCLE START, you can use the "Overstore" function to create states that are required, but not yet available, for the further program execution.

  By changing to the JOG REPOS mode, you can also manually traverse the tool from the current position to the setpoint position, if the setpoint position is not to be automatically approached after the program start

3.6.3 Simple search target definition

Requirement

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

Procedure

1. Press the "Block search" softkey.

2. Place the cursor on a particular program block.

   - OR -

   Press the "Find text" softkey, select the search direction, enter the search text and confirm with "OK".

3. Press the "Start search" softkey.

   The search starts. Your specified search mode will be taken into account.

   The current block will be displayed in the "Program" window as soon as the target is found.

4. If the located target (for example, when searching via text) does not correspond to the program block, press the "Start search" softkey again until you find your target.

   Press the "CYCLE START" key twice.

   Processing is continued from the defined position.

3.6.4 Defining an interruption point as search target

Requirement

A program was selected in "AUTO" mode and interrupted during execution through "CYCLE STOP" or "RESET".

Procedure

1. Press the "Block search" softkey.

2. Press the "Interrupt point" softkey.
   The interruption point is loaded.

3. If the "Higher level" and "Lower level" softkeys are available, use these to change the program level.

4. Press the "Start search" softkey.
   The search starts. 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.

5. Press the "CYCLE START" key twice.
   The execution will continue from the interruption point.

3.6.5 Entering the search target via search pointer

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

Requirement

The program is selected and the controller 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 subprograms.

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 subprogram 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 subprogram is called in the main program in two different places, you must also specify a target in program level 1 (main program).

Procedure

1. Press the "Block search" softkey.

2. Press the "Search pointer" softkey.
3. Enter the full path of the program as well as the subprograms, if required, in the input fields.

4. Press the "Start search" softkey.

   The search starts. 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.

5. Press the "CYCLE START" key twice.

   Processing is continued from the defined location.

Note

Interruption point

You can load the interruption point in search pointer mode.

3.6.6 Parameters for block search in the search pointer

Parameter	Meaning
Number of program level	
Program:	The name of the main program is automatically entered
Ext:	File extension
P:	Pass counter If a program section 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 Subprg. Subprogram call Line Line number
Search target	Point in the program at which machining is to start

3.6.7 Block search mode

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

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

Search variants

Block search mode	Meaning
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.
With calculation - with approach	It is used to be able to approach the contour in any circumstance. The end position of the block prior to the target block is found with "CYCLE START". The program runs in the same way as in normal program processing.
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.).

Machine manufacturer
Please refer to the machine manufacturer's specifications.

References

For additional information, please refer to the following documentation:
HMI sl / SINUMERIK 840D sl Commissioning Manual

Procedure

1. Select the "Machine" operating area.

2. Press the "AUTO" key.

3. Press the "Block search" and "Block search 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.
SB	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.

Activating 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:

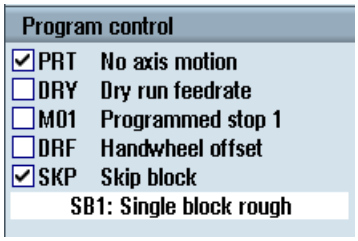
If a program control is activated, the abbreviation of the corresponding function appears in the status display as response.

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.



See also

Defining the dry run feedrate (Page 122)

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

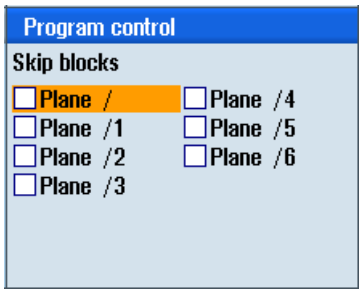
1. Select the "Machine" operating area.

...

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 Overstore

This function allows you to overstore technological parameters (for example, auxiliary functions, axis feed, spindle speed, programmable instructions, etc.) for a program run in the main memory of the NCK.

When next started, the program will be executed as originally programmed.

Requirement

The program to be corrected is in the Stop or Reset mode.

Procedure

1. Open the program in the "AUTO" mode.

2. Press the "Overstore" softkey.
   The "Overstore" window opens.
3. Enter the required data and NC block.

4. Press the <CYCLE START> key.
   The blocks you have entered are stored. You can observe execution in the "Overstore" window.
   After the entered blocks have been executed, you can append blocks again.
   You cannot change the operating mode while you are in overstore mode.

5. Press the "Back" softkey.
   The "Overstore" window closes.

6. Press the <CYCLE START> key again.
   The program selected before overstoring continues to run.

Note

Block-by-block execution

Using the <SINGLE BLOCK> key, the program can be overstored block-by-block.

Deleting blocks

Press the "Delete blocks" softkey to delete program blocks you have entered.

3.9 Editing a program

3.9.1 Overview - Program editor

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 called via the "Open" softkey as well as with the "INPUT" or "Cursor right" key in the "Program manager" operating area.
- The editor opens in the "Program" operating area with the last executed part program, when this was not explicitly exited via the "Close" softkey.

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.

If you are editing on a local drive or external drives, you can also exit the editor without saving, depending on the setting.

Exit the program correction mode using the "Close" softkey to return to the "Program manager" operating area.

See also

Editor settings (Page 99)

Opening and closing a program (Page 179)

Correcting a program (Page 81)

3.9.2 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.

Procedure

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. Select "Whole words" if you want to search for whole words only.

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

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 "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.

3.9.3 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" softkey.
   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 the "SELECT" button.

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.

3.9.4 Copying/pasting/deleting program blocks

Requirement

The program is opened in the editor.

Procedure

1. Press the "Mark" softkey.

   - OR -

   Press the "SELECT" key.

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 pasted.

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 paste the contents in another program, as well.

3.9.5 Renumbering a program

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.

Procedure

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

2. Press the "Renumber" 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.

3.9.6 Editor settings

Enter the default settings in the "Settings" window that are to take effect automatically when the editor is opened.

Presettings

Settings	Meaning
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 ¶ is 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.
Automatic save (only local and external drives)	Yes: The changes are saved automatically when you change to another operating area. No: You are prompted to save when changing to another operating area. Save or reject the changes with the "Yes" or "No" softkeys.

Note

All entries that you make here are effective immediately.

Procedure

1. Select the "Program" operating area

You have activated the editor.

2. Press the ">>" and "Settings" softkeys.
   The "Settings" window appears.

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

3.10 Simulating machining

3.10.1 Overview

During simulation, the current program is calculated in its entirety and the result displayed in graphic form. The result of programming is verified without traversing the machine axes. Incorrectly programmed machining steps are detected at an early stage and incorrect machining on the workpiece prevented.

Graphical representation

The simulation represented on the screen uses the correct workpiece and tool proportions.

The blank dimensions that are entered in the program editor are used for the workpiece.

The traversing paths of the tool are shown in color.

Simulation display

You can choose one of the following types of display:

- Material removal simulation

  During simulation or simultaneous recording you can follow stock removal from the defined blank.

- Path display

  You have the option of including the display of the path. The programmed tool path is displayed.

  If a blank is not defined, automatically only the tool path is displayed.

Display variants

You can choose between three variants of graphical display:

- Simulation before machining of the workpiece

  Before machining the workpiece on the machine, you can perform a quick run-through in order to graphically display how the program will be executed. The machine axes do not move.

- Simultaneous recording before machining of the workpiece

  Before machining the workpiece on the machine, you can graphically display how the program will be executed at dry run feedrate. The machine axes do not move.

- Simultaneous recording during machining of the workpiece

  You can follow machining of the workpiece on the screen while the program is being executed on the machine.

Views

The following views are available for all three variants:

- Top view
- 3D view
- 4-window view
- Side view

Status display

The current axis coordinates, the override, the current tool with cutting edge, the current program block, the feedrate and the machining time are displayed.

In all views, a clock is displayed during graphical processing. The machining time is displayed in hours, minutes and seconds. It is approximately equal to the time that the program requires for processing including the tool change.

Software options

You require the option "3D simulation of the finished part" for the 3D view and the 4-window view.

You require the option "Simultaneous recording (real-time simulation)" for the "Simultaneous recording" function.

3.10.2 Simulation prior to machining of the workpiece

Before machining the workpiece on the machine, you have the option of performing a quick run-through in order to graphically display how the program will be executed. This provides a simple way of checking the result of the programming.

Feedrate override

Feedrate override is also active during simulation.

You can change the feedrate during the simulation via the user interface.

0 %: The simulation stops.

≥ 100 %: The program is executed as quickly as possible.

See also

Changing the feedrate (Page 109)

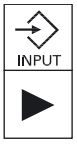
Simulating the program block by block (Page 110)

3.10.2.1 Start simulation

Procedure

1. Select the "Program manager" operating area.

2. Select the desired storage location and position the cursor on the program to be simulated.



3. Press the <INPUT> or <Cursor right> key.

   - OR -

   Double-click the program.

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

4. Press the "Simulation" and "Start" softkeys.

   The program execution is displayed graphically on the screen. The machine axes do not move.

5. Press the "Stop" softkey if you wish to stop the simulation.

   - OR -

   Press the "Reset" softkey to cancel the simulation.

   The unmachined workpiece blank is displayed again.

6. Press the "Start" softkey to restart the simulation.

3.10.3 Simultaneous recording prior to machining of the workpiece

Before machining the workpiece on the machine, you can graphically display the execution of the program on the screen to monitor the result of the programming.

Software option

You require the option "Simultaneous recording (real-time simulation)" for the simultaneous recording.

You can replace the programmed feedrate with a dry run feedrate to influence the speed of execution.

If you would like to view the current program blocks again instead of the graphical display, you can switch to the program view.

3.10.3.1 Starting simultaneous recording

Procedure

1. Load a program in the "AUTO" mode.
2. Press the "Prog. ctrl." softkey and activate the checkboxes "PRT no axis movement" and "DRY run feedrate".

   The program is executed without axis movement. The programmed feedrate is replaced by a dry run feedrate.
3. Press the "Sim. rec." softkey.

4. Press the <CYCLE START> key.

   The program execution is displayed graphically on the screen.

5. Press the "Prog. ctrl." softkey and deactivate the "DRY run feedrate" checkbox.

   Simultaneous recording is performed with the programmed feedrate.

3.10.4 Simultaneous recording during machining of the workpiece

If the view of the work space is blocked by coolant, for example, while the workpiece is being machined, you can also track the program execution on the screen.

You can switch on the simultaneous recording of the machining process even when the machining of the workpiece is already running on the machine.

Software options

You require the option "Simultaneous recording (real-time simulation)" for the simultaneous recording.

Procedure

1. Load a program in the "AUTO" mode.
2. Press the "Sim. rec." softkey.

3. Press the "CYCLE START" key.

   The machining of the workpiece is started and graphically displayed on the screen.

3.10.5 Different views of a workpiece

In the graphical display, you can choose between different views so that you constantly have the best view of the current workpiece machining, or in order to display details or the overall view of the finished workpiece.

The following views are available:

- Top view
- 3D view
- 4-window view
- Side view

3.10.5.1 Plan view

1. Start the simulation.
2. Press the "Top view" softkey.

   The workpiece is shown from above in the top view.

   A depth display indicates the current depth at which machining is currently taking place.

   The rules for depth display in this graphical representation is: "The deeper, the darker".

Changing the display

You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

See also

Enlarging or reducing the graphical representation (Page 111)

Panning a graphical representation (Page 112)

Modifying the viewport (Page 113)

3.10.5.2 3D view

1. Start the simulation.
2. Press the "3D view" softkey.

Software option

You require the option "3D simulation (finished part)" for the simulation.

Changing the display

You can increase or decrease the size of the simulation graphic, move it, turn it, or change the segment.

Displaying and moving cutting planes

You can display and move cutting planes X, Y, and Z.

See also

Enlarging or reducing the graphical representation (Page 111)

Panning a graphical representation (Page 112)

Rotating the graphical representation (Page 112)

Modifying the viewport (Page 113)

3.10.5.3 4-window view

1. Start the simulation.
2. Press the "Additional views" and "4-window view" softkeys.

The 4-window view shows the top view (top left window) and side view (top right window and bottom left window) as well as a 3D view (bottom right window) with the appropriate option.

Changing the display

You can increase or decrease the size of the simulation graphic, move it, turn it, or change the segment.

The segment changes for the top view and side view synchronously.

3.10.5.4 Side view

1. Start the simulation.
2. Press the "Additional views" and "Side view" softkeys.

Changing the display

You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

3.10.6 Graphical display

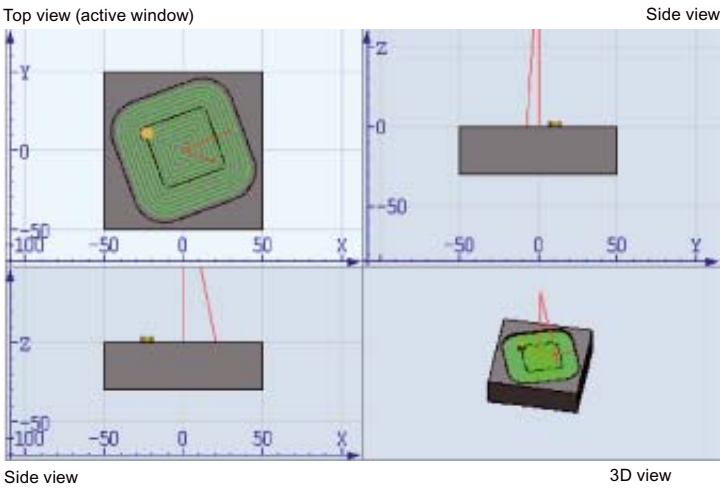


Figure 3-1 4-window view

Active window

The currently active window has a lighter background than the other view windows.

You can change the workpiece display here, e.g. increase or decrease the size, turn it and move it.

Some of the actions that you perform in the active window also have a simultaneous effect in other view windows.

Display of the traversing paths

- Rapid traverse = red
- Feed = green

3.10.7 Editing the simulation display

3.10.7.1 Blank display

You can change the blank defined in the program.

Procedure

1. The simulation or the simultaneous recording is started.
2. Press the ">>" and "Blank" softkeys.
   The "Blank Input" windows opens and displays the pre-assigned values.
3. Enter the desired values for the dimensions.

4. Press the "Accept" softkey to confirm your entries. The new dimensions are taken into account in the display of the workpiece.

3.10.7.2 Showing and hiding the tool path

The path display follows the programmed tool path of the selected program. The tool paths can be shown or hidden as required.

Procedure

1. The simulation or the simultaneous recording is started.

2. Press the "<<" softkey to switch to the main screen, if required.

2. Press the ">>" softkey.
   The tool path or tool paths for several tools are displayed in the active view. The path is continuously updated as a function of the tool movement.

3. Press the softkey to hide the tool paths.
   The tool paths are still generated in the background and can be shown again by pressing the softkey again.

4. Press the '"Delete tool path" softkey.
   All tool paths so far generated, including tool paths generated in the background are deleted.

3.10.8 Program control during simulation

3.10.8.1 Changing the feedrate

You can change the feedrate at any time during the simulation.

You can track the changes in the dialog line.

Note

If you are working with the "Simultaneous recording" function, the override rotary switch on the control panel is used.

Procedure

1. Simulation is started.
2. 

Press the "Program control" softkey.
3. 

Press the "Override +" or "Override -" softkey to increase or decrease the feedrate by 5%.

- OR -

Press the "Override 100%" softkey to set the feedrate to the maximum value.

- OR -

4. 
Press the "<<" softkey to return to the main screen and perform the simulation with changed feedrate.

3.10.8.2 Simulating the program block by block

You can control the program execution during simulation, i.e. execute a program block by block, as when executing a program.

Procedure

1. Simulation is started.
2. 

Press the "Program control" and "Single block" softkeys.
3. 


Press the "<<" and "Start SBL" softkeys.
The pending block of the program is simulated and then stops.
4. 

Press "Start SBL" as many times as you want to simulate a single program block.
5. 

Press the "Program control" and the "Single block" softkeys to exist the single block mode.

3.10.9 Modifying and adapting the simulation graphics

3.10.9.1 Enlarging or reducing the graphical representation

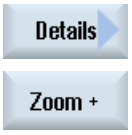
Requirement

The simulation or the simultaneous recording is started.

Procedure

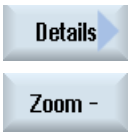
1. Press the <+> and <-> keys if you want to enlarge or reduce the graphical representation now displayed.
   The graphical representation is enlarged or reduced from the center.

- OR -



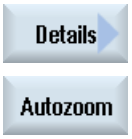
Press the "Details" and "Zoom +" softkeys if you want to increase the size of the segment.

- OR -



Press the "Details" and "Zoom -" softkeys if you want to decrease the size of the segment.

- OR -



Press the "Details" and "Auto zoom" softkeys if you want to automatically adapt the segment to the size of the window.
The automatic scaling function "Fit to size" takes account of the largest expansion of the workpiece in the individual axes.

Note

Program change during viewing

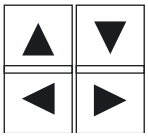
If you have enlarged or reduced a segment and change the program during simulation or simultaneous recording, the view switches back to the automatic scaling for the current blank.

3.10.9.2 Panning a graphical representation

Requirement

The simulation or the simultaneous recording is started.

Procedure



1. Press a cursor key if you want to move the graphic up, down, left, or right.

3.10.9.3 Rotating the graphical representation

In the 3D view and in the 4-window view, you can rotate the workpiece to view it from all sides.

Requirement

Simulation is started and the 3D view or the 4-window view is selected.

Procedure

1. Press the "Details" softkey.

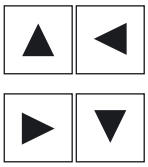
2. Press the "Rotate view" softkey.


...


...

2. Press the "Arrow right", "Arrow left", "Arrow up", "Arrow down", "Arrow clockwise" and "Arrow counterclockwise" softkeys to change the position of the workpiece.

- OR -



Keep the <Shift> key pressed and then turn the workpiece in the desired direction using the appropriate cursor keys.

3.10.9.4 Modifying the viewport

If you would like to move, enlarge or decrease the size of the segment of the graphical display, e.g. to view details or display the complete workpiece, use the magnifying glass.

Using the magnifying glass, you can define your own segment and then increase or decrease its size.

Requirement

The simulation or the simultaneous recording is started.

Procedure

1. Press the "Details" softkey.
2. Press the "Magnifying glass" softkey.

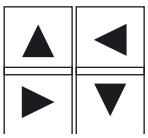
   A magnifying glass in the shape of a rectangular frame appears.

3. Press the "Magnify +" or <+> softkey to enlarge the frame.

   - OR -

   Press the "Magnify -" or <-> softkey to reduce the frame.

   - OR -



   Press a cursor key to move the frame up, down, left or right.

3.10.10 Displaying simulation alarms

Special alarms might occur during simulation. If an alarm occurs during a simulation run, a window opens in the operating window to display it.

The alarm overview contains the following information:

- Date and time
- Deletion criterion

  Specifies with which softkey the alarm is acknowledged
- Alarm number
- Alarm text

Requirement

Simulation is running and an alarm is active.

Procedure

1. Press the "Program control" and "Alarm" softkeys.
   The "Simulation Alarms" window is opened and a list of all pending alarms is displayed.

Press the "Acknowledge alarm" softkey to reset the simulation alarms indicated by the Reset or Cancel symbol.
The simulation can be continued.
- OR -

Press the "Simulation Power On" softkey to reset a simulation alarm indicated by the Power On symbol.
The simulation is terminated and loaded again.

Acknowledgement symbols

Symbol	Meaning
	Cancel symbol
	Reset symbol
	Power On symbol

3.11 Display G and auxiliary functions

3.11.1 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 controller 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 refer to the machine manufacturer's specifications.

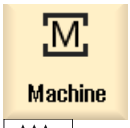
G groups displayed by default

Group	Meaning
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	Plane selection (e.g. G17, G18)
G group 7	Tool radius compensation (e.g. G40, G42)
G group 8	Settable work offset (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)

G groups displayed by default (ISO code)

Group	Meaning
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	Plane selection (e.g. G17, G18)
G group 7	Tool radius compensation (e.g. G40, G42)
G group 8	Settable work offset (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)

Procedure



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 refer to the machine manufacturer's specifications.

References

For more information about configuring the displayed G groups, refer to the following document:

HMI sl / 840D sl Commissioning Manual

3.11.2 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 controller is displayed.

Additional information in the footer

The following additional information is displayed in the footer:

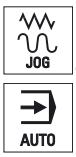
- Current transformations

Display	Meaning
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.

- Current zero offsets
- Spindle speed
- Path feedrate
- Active tool

Procedure

1. Select the "Machine" operating area.



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

3. Press the ">>" and "All G functions" softkeys.
   The "G Functions" window is opened.

3.11.3 Auxiliary functions

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

Displayed auxiliary functions

Up to five current M functions and three H functions are displayed in the "Auxiliary Functions" window.

Procedure

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.

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

You get a list with all currently active synchronized actions.

In this list, the synchronized action programming is displayed in the same form as in the part program.

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	Meaning
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 once, irrespective of the identification number.

Procedure

1. Select the "Machine" operating area.

2. Press the "AUTO" key.

3. Press the menu forward key and the "Synchron." softkey.
   The "Synchronized Actions" window appears.

3.12 Displaying the program runtime and counting workpieces

To gain an overview of the program runtime and the number of machined workpieces, open the "Times, Counter" window.

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

Displayed times

- Program

  Pressing the softkey the first time shows how long the program has already been running.

  At every further start of the program, the time required to run the entire program the first time is displayed.

  If the program or the feedrate is changed, the new program runtime is corrected after the first run.

- Program remainder

  Here you can see how long the current program still has to run. In addition, you can

  follow how much of the current program has been completed in percent on a progress bar.

  The display only appears when the program is run a second time.

If you are executing the program from an external location, the program loading progress is displayed here.

- Influencing the time measurement

  The time measurement is started with the start of the program and ends with the end of the program (M30) or with an agreed M function.

  When the program is running, the time measurement is interrupted with "CYCLE STOP" and continued with "CYCLE START".

  The time measurement starts at the beginning with "RESET" and subsequent "CYCLE START".

  The time measurement stops with "CYCLE STOP" or a feedrate override = 0.

Counting workpieces

You can also display program repetitions and the number of completed workpieces. For the worpiece count, enter the actual and planned workpiece numbers.

Workpiece count

Completed workpieces can be counted via the end of program command (M30) or an M command.

Procedure

1. Select the "Machine" operating area.

2. Press the "AUTO" key.

3. Press the "Times, Counter" softkey.
   The "Times, Counter" window opens.

4. Select "Yes" under "Count workpieces" if you want to count completed workpieces.

5. Enter the number of workpieces needed in the "Desired workpieces" field.
   The number of workpieces already finished is displayed in "Actual workpieces". This value can be corrected if necessary.
   After the defined number of workpieces is reached, the current workpieces display is automatically reset to zero.

3.13 Setting for automatic mode

3.13.1 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".

Procedure

1. Select the "Machine" operating area.

2. Press the "AUTO" key.

3. Press the menu forward key and 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.

See also

Program control (Page 90)

User data

4

4.1 Overview

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.

4.2 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

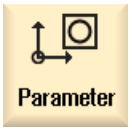
The number of channel-specific R parameters is defined in a machine data element.

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

1. Press the ">>" and "Delete area" softkeys.
   The "Delete R parameters" window appears.

...

2. Enter the R parameter(s) whose channel-specific values you would like to delete and press the "OK" softkey.

   - OR -

   Press the "Delete all" softkey and confirm the prompt with "OK" if you really want to delete all R parameters.

...

A value of 0 is assigned to the selected R parameters or to all R parameters.

4.3 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)

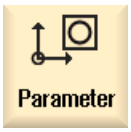
Example

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 ">>" 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.

4.4 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

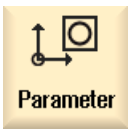
A channel-specific GUD variable is defined with the following:

- 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.

4.5 Displaying local user data (LUD)

Local user data

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

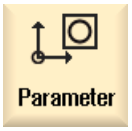
The control displays the LUDs after the start of program processing. The display is available until the end of program processing.

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.

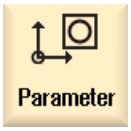
3. Press the "Local LUD" softkey.

4.6 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.

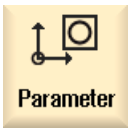
2. Press the "User variable" softkey.

3. Press the "Program PUD" softkey.

4.7 Searching for user data

You can search for R parameters and user data.

Procedure



1. Select the "Parameter" operating area.

...

2. Press the "R parameters", "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.

4.8 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 "Startup" operating area.

2. Press the "System data" softkey.
3. In the data tree, 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.

Teaching in a program 5

5.1 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.

5.2 General sequence

General sequence

Select the desired program block, press the relevant softkey "Teach position", "Rap. tra. G01", "Straight line G1" or "Circ. interm. 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.

Note

All defined axes are "taught in" in the first teach-in block. In all additional teach-in blocks, only axes modified by axis traversing or manual input are "taught in".

If you exit teach-in mode, this sequence begins again.

Operating mode or operating area switchover

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

5.3 Inserting a block

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.

5.3.1 Teach-in via Windows

5.3.1.1 General information

The cursor must be positioned on an empty line.

The windows for pasting program blocks contain input and output 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.

When first selected, the input fields are empty unless axes were already traversed before the window was selected.

All data from the input/output fields are transferred to the program via the "Accept" softkey.

Requirement

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

Proceed as follows

1 Select the "Machine" operating area.

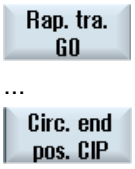
...

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.

5.3.1.2 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.

5.3.1.3 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.

5.3.1.4 Teaching in circle intermediate and 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.

5.3.1.5 A-spline teach in (option)

For Akima-spline interpolation, you enter interpolation points that are connected by a smooth curve.

Enter a starting point and specify a transition at the beginning and end.

You teach-in each interpolation point via "Teach in of position".

Note

The relevant option bit must be set to enable you to program a spline interpolation.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

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. Press the ">>" and "ASPLINE" softkeys.
   The "Akima-spline" window opens with the input fields.

6. Traverse the axes to the required position and if necessary, set the transition type for the starting point and end point.

7. Press the "Accept" softkey.
   A new program block will be inserted at the cursor position.

   - OR -

   Press the "Cancel" softkey to cancel your input.

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.

5.3.2 Input parameters for teach-in blocks

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

Parameter	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

Parameter	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

Transition types for teach-in of position and teach-in of G0 and G1, and ASPLINE

The following parameters are offered for the transition:

Parameter	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

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

The following motion parameters are offered:

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

Transition behavior at the beginning and end of the spline curve

The following motion parameters are offered:

Parameter	Description
Start	
BAUTO	Automatic calculation
BNAT	Curvature is zero or natural
BTAN	Tangential
End	
EAUTO	Automatic calculation
ENAT	Curvature is zero or natural
ETAN	Tangential

5.4 Editing a block

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

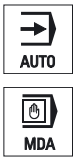
If you wish to change any variable in a block in the program block window other than the position and its parameters, then we recommend alphanumerical input.

Requirement

The program to be edited is selected.

Proceed as follows

1. Select the "Machine" operating area.

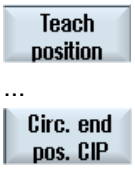


2. Press the "AUTO" or "MDA" key.

3. Press the "TEACH IN" key.

4. Press the "Teach prog." softkey.

5. Click the program block to be edited.



6. 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.

7. 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.

5.5 Selecting a block

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.

With teach-in, you can also change program areas that have already been executed. This automatically disables program processing.

You must press reset or select a block to resume the program.

Requirement

The program to be processed 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. Place the cursor on the desired program block.

6. Press the "Block selection" softkey.

5.6 Deleting a block

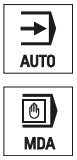
You have the option of deleting a program block entirely.

Requirement

"AUTO" mode: The program to be processed 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.

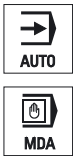
6. Press the ">>" and "Delete block" softkeys.
   The program block on which the cursor is positioned is deleted.

5.7 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. Press the ">>" and "Settings" softkeys.
   The "Settings" window appears.

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

Tool management

6

6.1 Lists for the tool management

All tools and also all magazine locations that have been created or configured in the NC are displayed in the lists in the Tool area.

All lists display the same tools in the same order. When switching between the lists, the cursor remains on the same tool in the same screen segment.

The lists have different parameters and softkey assignments. Switching between lists is a specific change from one topic to the next.

- **Tool list**

  All parameters and functions required to create and set up tools are displayed.

- **Tool wear**

  All parameters and functions that are required during operation, e.g. wear and monitoring functions, are listed here.

- **Magazine**

  Magazine and magazine location-related parameters and functions for the tools and magazine locations are listed here.

- **Tool data OEM**

  This list can be freely defined by the OEM.

Sorting the lists

You can change the sorting within the lists:

- According to magazine
- According to tool identifier (alphabetic)
- According to tool type
- According to T number (numeric tool identifier)

6.2 Magazine management

Depending on the configuration, the tool lists support a magazine management.

Magazine management functions

- Press the "Magazine" horizontal softkey to obtain a list that displays tools with magazine-related data.
- The Magazine / Magazine location column is displayed in the lists.
- In the default setting, the lists are displayed sorted according to magazine location.
- The magazine selected via the cursor is displayed in the title line of each list.
- The "Magazine selection" vertical softkey is displayed in the tool list.
- You can load and unload tools to and from a magazine via the tool list.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

6.3 Tool types

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

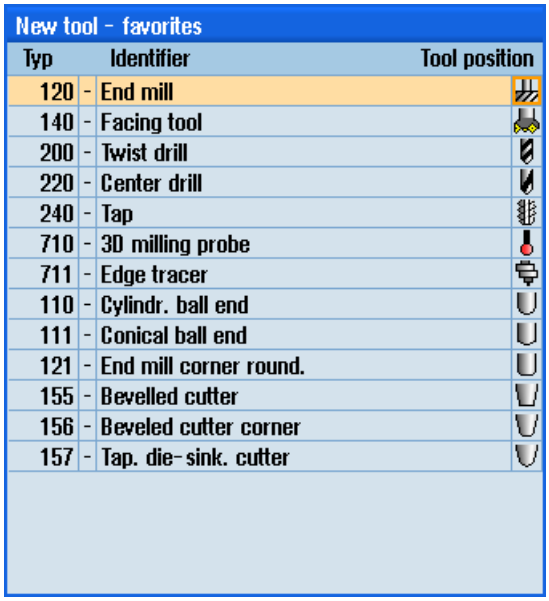


Figure 6-1 Example of Favorites list

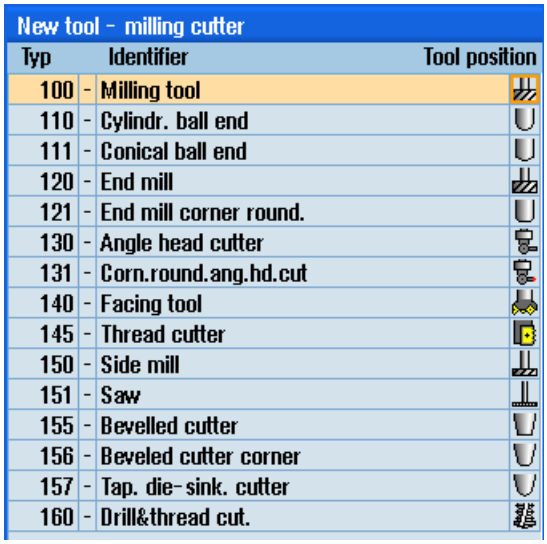


Figure 6-2 Available tools in the "New tool - milling cutter" window

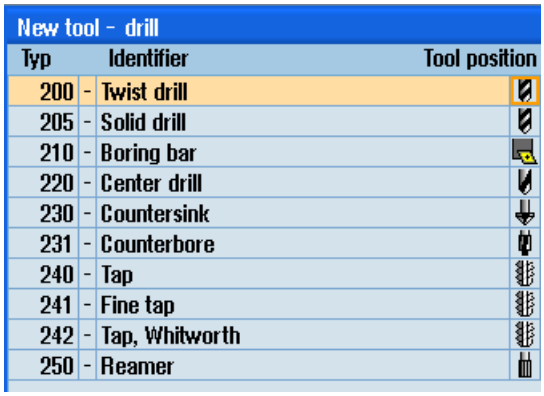


Figure 6-3 Available tools in the "New tool - drill" window

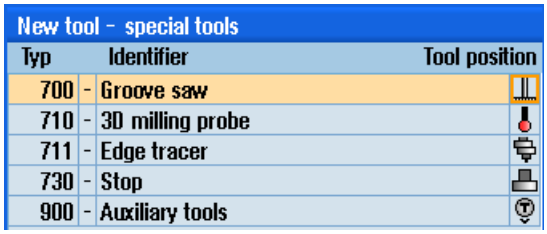


Figure 6-4 Available tools in the "New tool - special tools" window

6.4 Tool dimensioning

This section provides an overview of the dimensioning of tools.

Tool types

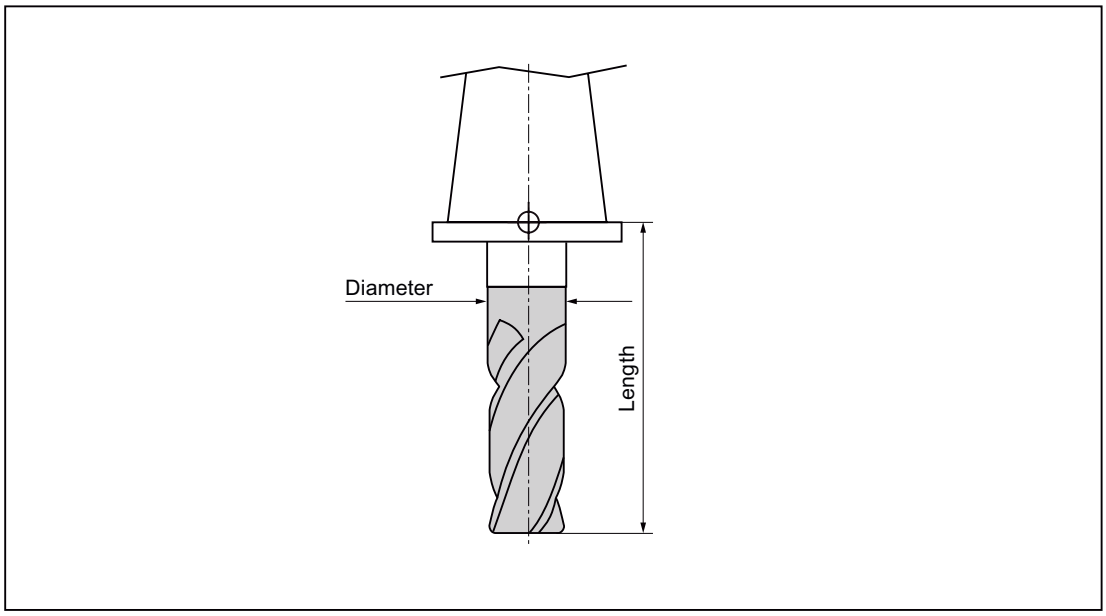


Figure 6-5 End mill (Type 120)

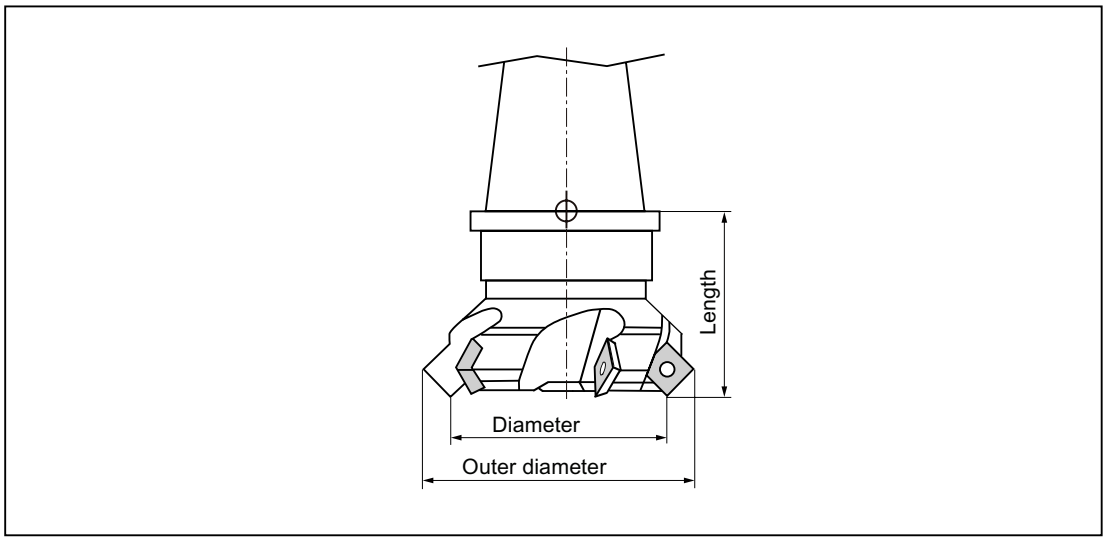


Figure 6-6 Face mill (Type 140)

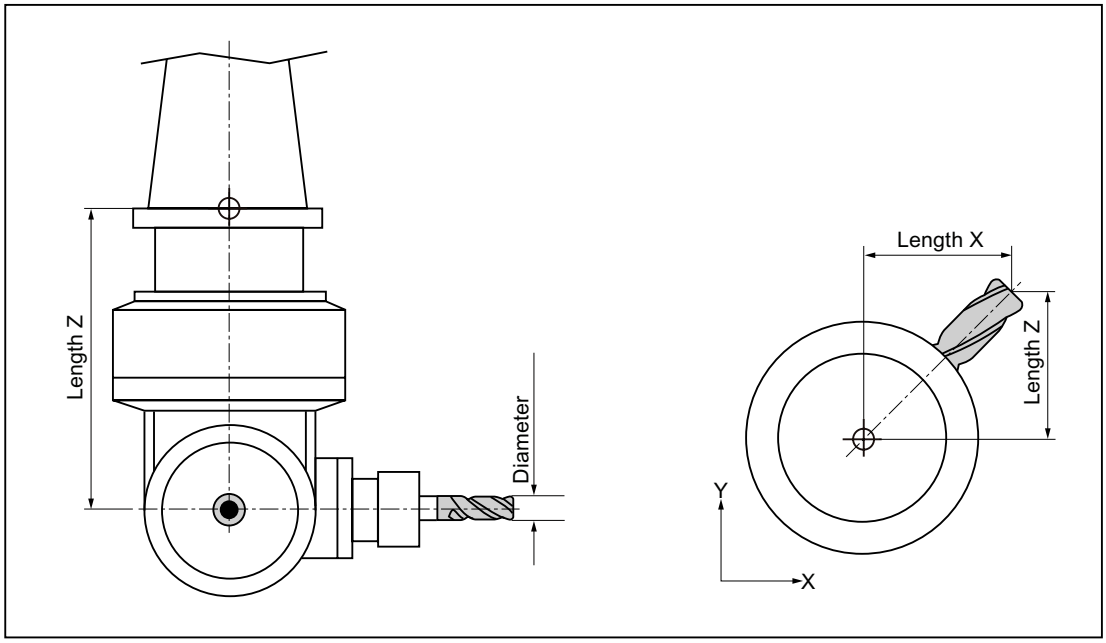


Figure 6-7 Angle head cutter (Type 130)

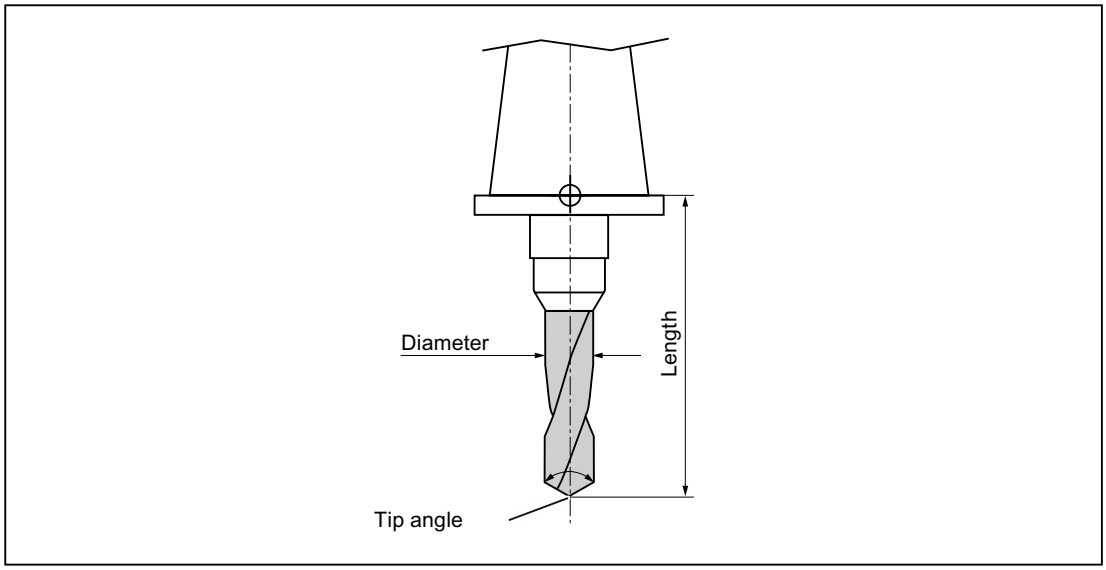


Figure 6-8 Drill (Type 200)

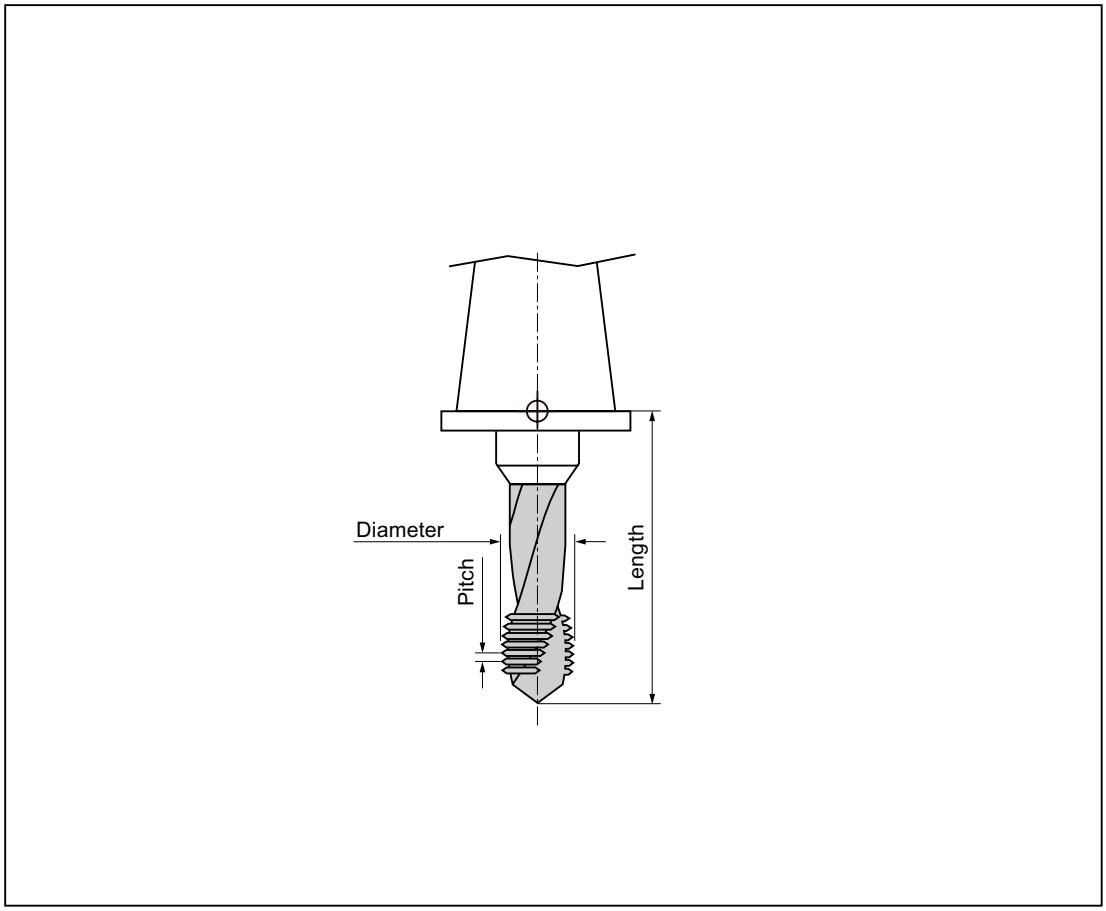


Figure 6-9 Tap (Type 240)

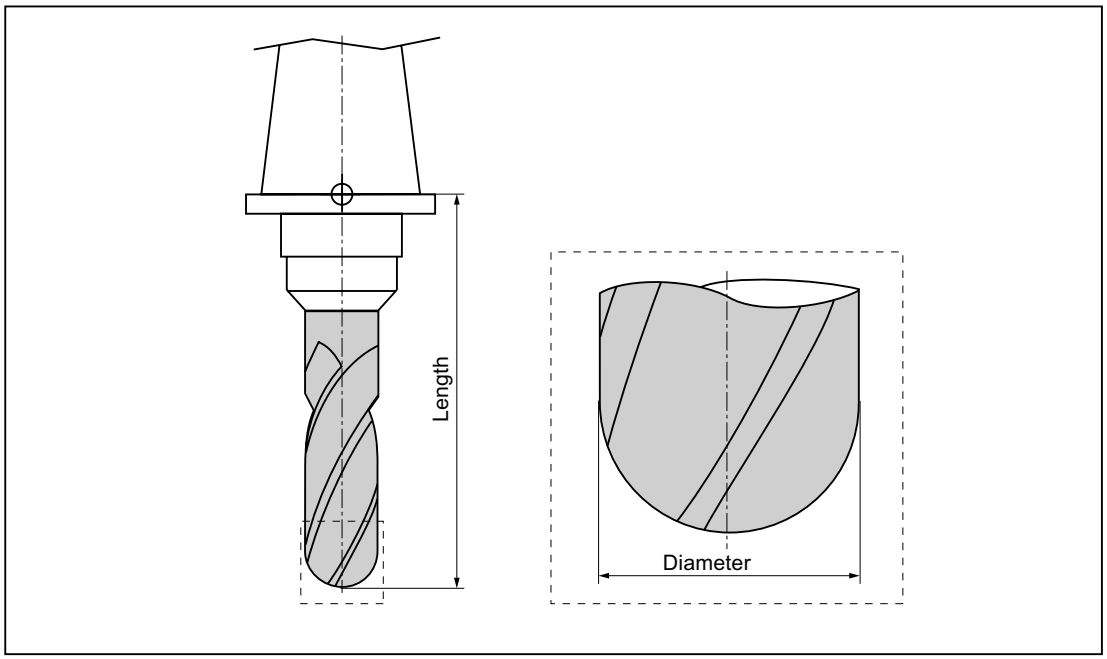


Figure 6-10 3D tool with an example of a cylindrical die-sinking cutter (Type 110)

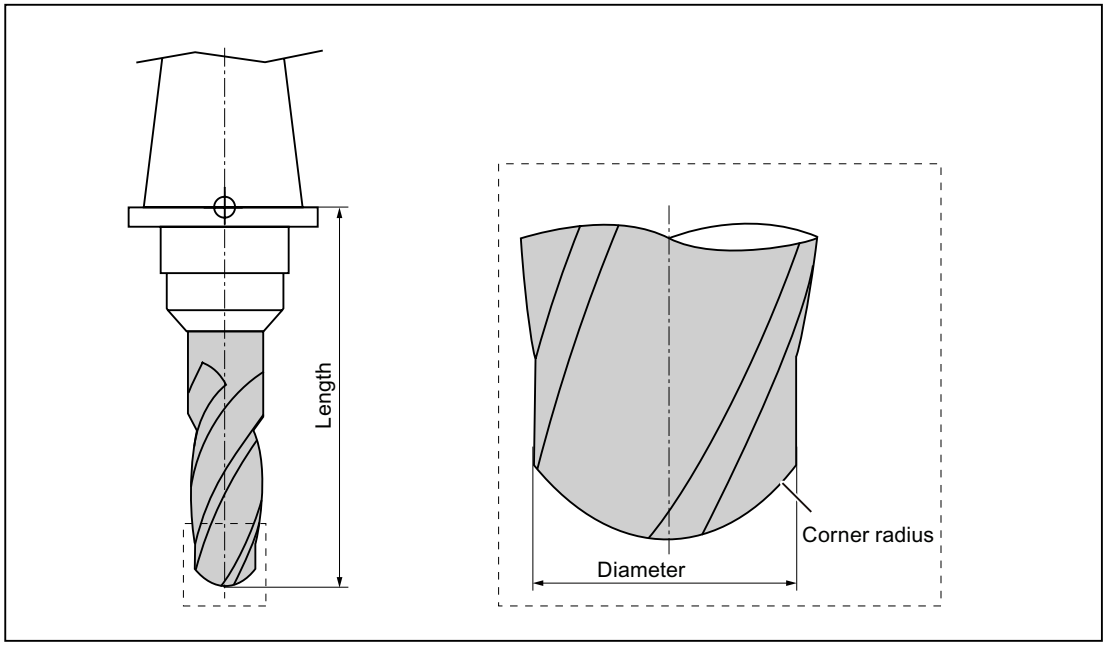


Figure 6-11 3D tool type with an example of a ballhead cutter (Type 111)

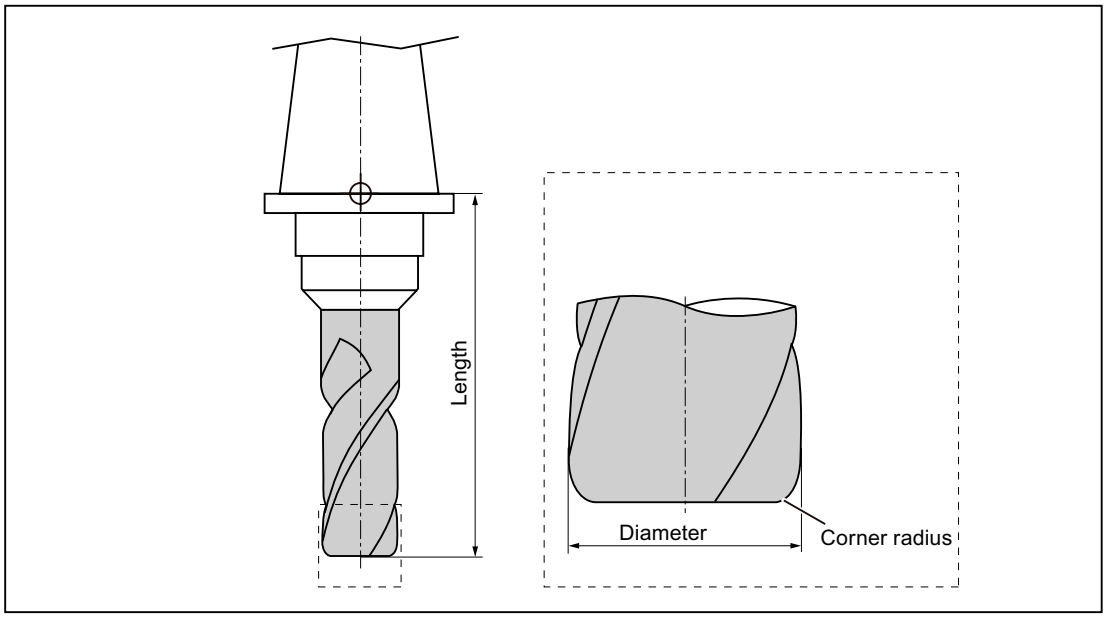


Figure 6-12 3D tool with an example of an end mill with corner rounding (Type 121)

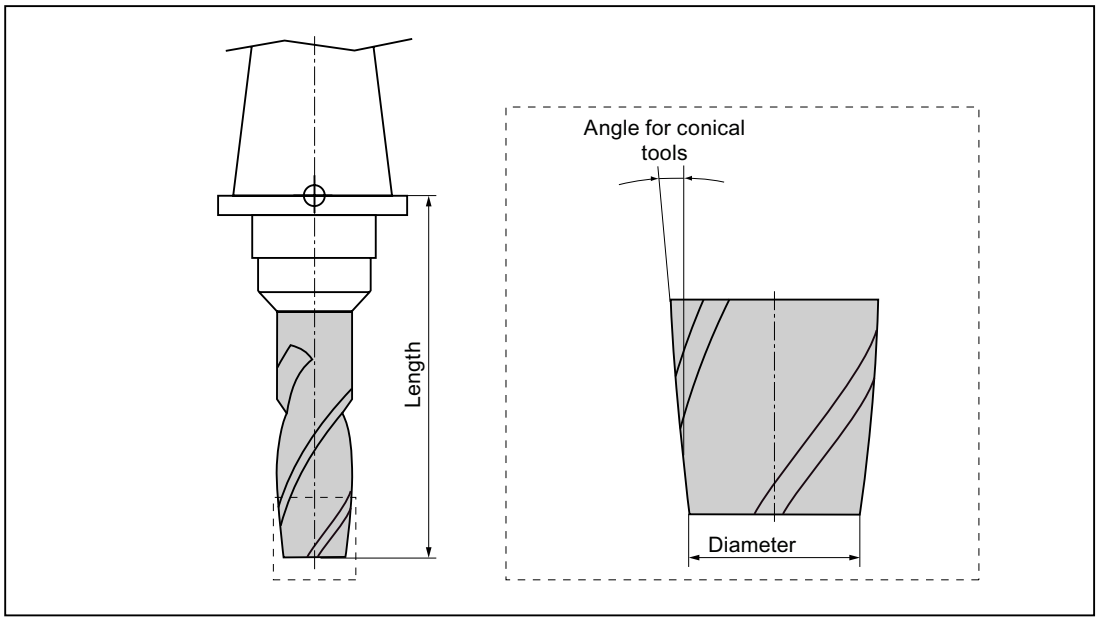


Figure 6-13 3D tool type with an example of a bevel cutter (Type 155)

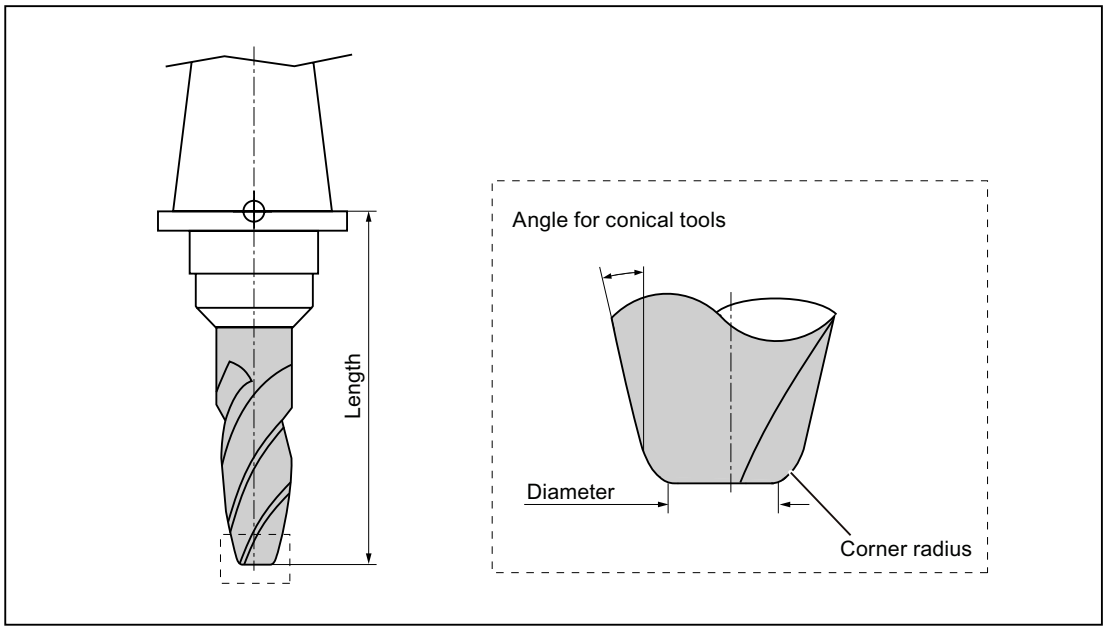


Figure 6-14 3D tool with an example of a bevel cutter with corner rounding (Type 156)

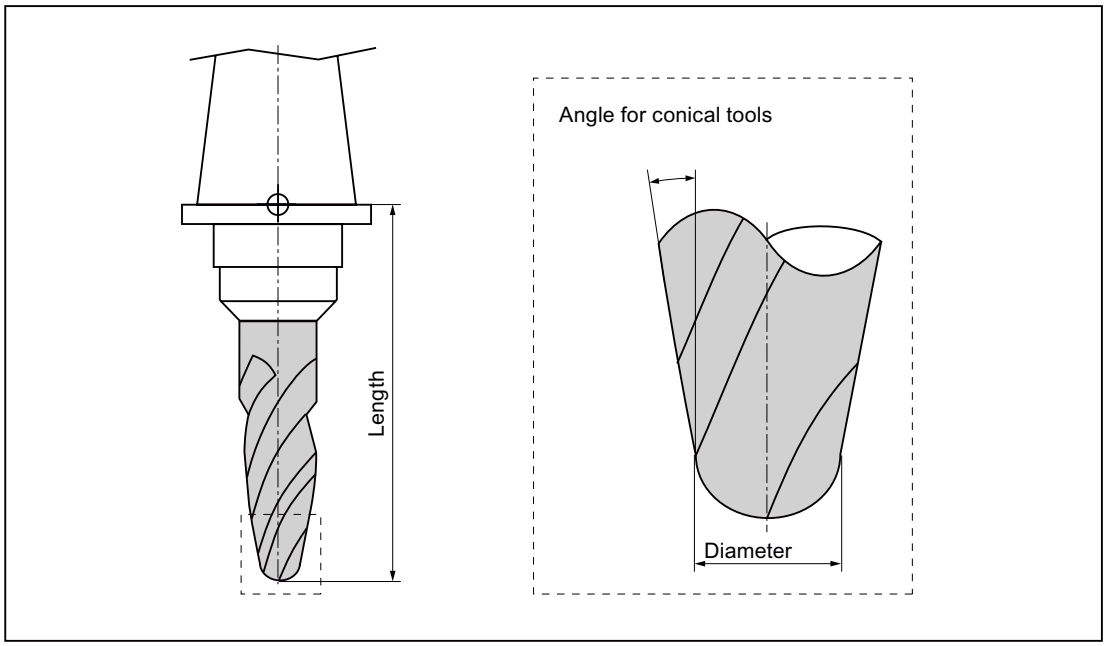


Figure 6-15 3D tool with an example of a tapered die-sinking cutter (Type 157)

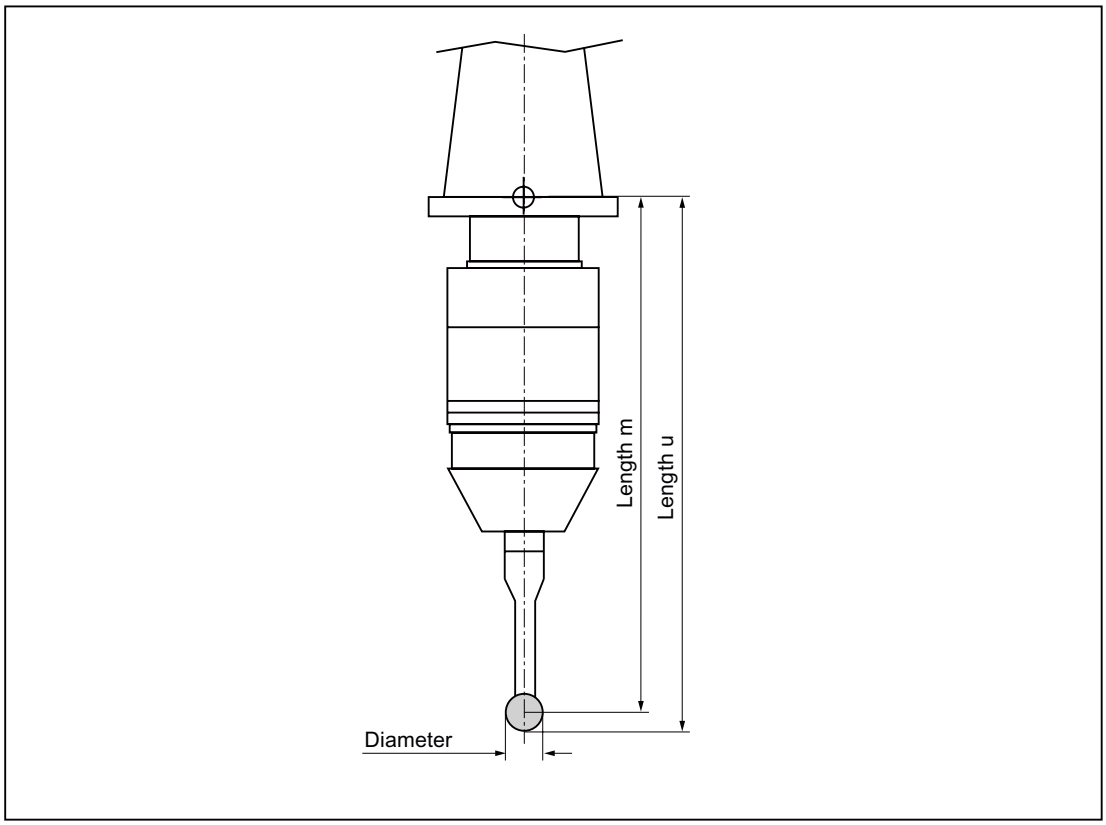


Figure 6-16 3D probe

Machine manufacturer

The tool length is measured to the center of the ball (length m) or to the ball circumference (length u).

Please refer to the machine manufacturer's specifications.

Note

A 3D probe must be calibrated before use.

6.5 Tool list

All parameters and functions that are required to create and set up the tools are displayed in the tool list.

Each tool is uniquely identified by the tool identifier and the replacement tool number.

Tool parameters

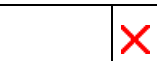
Column heading	Meaning
Location  *  * * If activated in magazine selection	Magazine/location number • Spindle location as an icon. • Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons. • Magazine location number. The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed.
Type	Tool type Specific tool offset data is displayed depending on the tool type (represented as an icon).
Tool name	The tool is identified by the name and the replacement tool number. You may enter the names as text or numbers.
ST	Replacement tool number (for replacement tool strategy)
D	Cutting edge number
Length	Tool length Geometry data, length
Radius	Tool radius
Tip angle or Pitch	Tip angle for Type 200 - twist drill and Type 220 - center drill Pitch for Type 240 - tap
N	Number of teeth for Type 110 - ball end mill for cylindrical die-sinking cutter, Type 111 - ball end mill for tapered die-sinking cutter, Type 120 - end mill, Type 121 - end mill with corner rounding, Type 130 - angle head cutter, Type 140 - facing tool, Type 150 - side mill, Type 155 - bevel cutter, Type 156 - bevel cutter with corner rounding, and Type 157 - tapered die-sinking cutter.

You use the configuration file to specify the selection of parameters in the list.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Icons in the tool list

Icon/ Designation		Meaning
Tool type		
Red "X"		The tool is disabled.
Yellow triangle pointing downward		The prewarning limit has been reached.
Yellow triangle pointing upward		The tool is in a special state. Place the cursor on the marked tool. A tooltip provides a short description.
Green border		The tool is preselected.
Magazine/location number		
Green double arrow		The magazine location is positioned at the change position.
Gray double arrow (configurable)		The magazine location is positioned at the loading position.
Red "X"		The magazine location is disabled.

Additional data

The following tool types require geometry data that is not included in the tool list display.

Tool type	Additional parameters
111 Conical ballhead cutter	Corner radius
121 End mill with corner rounding	Corner radius
130 Angle head cutter	Geometry length (length X, length Y, length Z) Wear length (Δ length X, Δ length Y, Δ length Z) Adapter length (length X, length Y, length Z) V (direction vector 1 - 6) Vector X, vector Y, vector Z
131 Angle head cutter with corner rounding	Geometry length (length X, length Y, length Z) Corner radius Wear length (Δ length X, Δ length Y, Δ length Z) Adapter length (length X, length Y, length Z) V (direction vector 1 - 6) Vector X, vector Y, vector Z

Tool type	Additional parameters
140 Face milling	External radius Tool angle
155 Bevel cutter	Taper angle
156 Bevel cutter with corner rounding	Corner radius Taper angle
157 Conical die-milling cutter	Taper angle

You can use the configuration file to specify the data to be displayed for specific tool types in the "Additional Data" window.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

1. Select the "Parameter" operating area.

2. Press the "Tool list" softkey.

3. Press the "Additional data" softkey to display further information for selected tools.

   The "Additional Data" window opens.

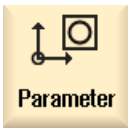
   The "Additional data" softkey is only active if a tool for which the "Additional Data" window is configured is selected.

6.5.1 Creating a new tool

When creating a new tool, the "New Tool - Favorites" window offers you a number of selected 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.

Procedure



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 magazine.

   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 -

If you want to create a tool that is not in the "Favorites" list, press the "Cutters 100-199", "Drills 200-299" or "Sp. tools 700-900" softkey.

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 tool is added to the tool list with a predefined name. If the cursor is located on an empty magazine location in the tool list, then the tool is loaded to this magazine location.

The tool creation sequence can be defined differently.

Multiple load points

If you have configured several loading points for a magazine, then the "Loading Point Selection" window appears when a tool is created directly in an empty magazine location or when the "Load" softkey is pressed.

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

Additional data

If configured accordingly, the "New Tool" window opens after the required tool has been selected and confirmed with "OK".

You can define the following data in this window:

- Names
- Tool location type
- Size of tool

References:

For a description of configuration options, refer to the

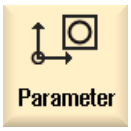
/IHsl/, HMI sl / SINUMERIK 840D sl Commissioning Manual

6.5.2 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 controller configuration.

Tool cutting edges that are not required can be deleted.

Procedure



1. Select the "Parameter" operating area.

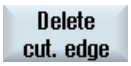
2. Press the "Tool list" softkey.
3. Position the cursor on the tool for which you would like to store more cutting edges.

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

5. Press the "New cutting edge" softkey.

   A new data set is stored in the list.

   The cutting edge number is incremented by one and the offset data is assigned the values of the cutting edge on which the cursor is positioned.
6. Enter the offset data for the second cutting edge.



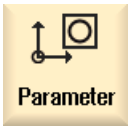
7. Repeat this process if you wish to create more tool edge offset data.
8. Position the cursor on the cutting edge that you want to delete and press the "Delete cutting edge" softkey.

   The data set is deleted from the list. The first tool cutting edge cannot be deleted.

6.5.3 Deleting a tool

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

Procedure



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.

   A safety prompt is displayed.

5. Press the "OK" softkey if you really want to delete the tool.

   Use this softkey to delete the tool.

   If the tool is in a magazine location, it is unloaded and then deleted.

Multiple load points - tool in magazine location

If you have configured several loading points for a magazine, then the "Loading Point Selection" window appears after pressing the "Delete tool" softkey.

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

6.5.4 Loading and unloading tools

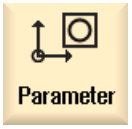
You can load and unload tools to and from a magazine via the tool list. When a tool is loaded, it is taken to a magazine location. When it is unloaded, it is removed from the magazine and stored in the tool list.

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 in the NC memory 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.

Procedure



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 "... 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.

   - OR -

   Press the "Spindle" softkey.

   The tool is loaded into the specified magazine location or spindle.

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 "Loading Point Selection" window appears after pressing the "Load" softkey.

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

Unloading tools

1. Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" softkey.
2. Select the required load point in the "Loading Point Selection" window.
3. Confirm your selection with "OK".

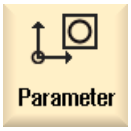
- OR -

Undo your selection with "Cancel".

6.5.5 Selecting a magazine

You can directly select the buffer memory, the magazine, or the NC memory.

Procedure



1. Select the "Parameter" operating area.

2. Press the "Tool list" softkey.

3. Press the "Magazine selection" softkey.

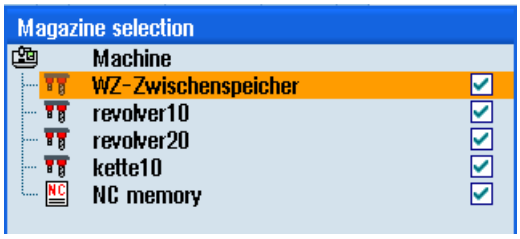
If there is only one magazine, you will move from one area to the next (i.e. from the buffer memory to the magazine, from the magazine to the NC memory, and from the NC memory back to the buffer memory) each time you press the softkey. The cursor is positioned at the beginning of the magazine each time.

- OR -

If there is more than one magazine, the "Magazine Selection" window opens. Position the cursor on the desired magazine in this window and press the "Go to" softkey.

The cursor jumps directly to the beginning of the specified magazine.

Hiding magazines



Deactivate the checkbox next to the magazines that you do not want to appear in the magazine list.

The magazine selection behavior with multiple magazines can be configured in different ways.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

References

For a description of configuration options, refer to the

HMI sl / SINUMERIK 840D sl Commissioning Manual

6.6 Tool wear

All parameters and functions that are required during operation are contained in the tool 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 controller then takes this information into account when calculating the tool length or radius compensation. This ensures a consistent level of accuracy during workpiece machining.

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.

Tool parameters

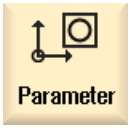
Column heading	Meaning
Location  *  * * If activated in magazine selection	Magazine/location number • Spindle location as an icon. • Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons. • The magazine location numbers. The magazine number is specified 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.
Type	Tool type Depending on the tool type (represented by an icon), certain tool offset data is enabled.
Tool name	The tool is identified by the name and the replacement tool number. You can enter the name as text or number.
ST	Replacement tool number (for replacement tool strategy).
D	Cutting edge number
Δ Length	Length wear
Δ Radius	Radius wear
T C	Selection of tool monitoring - by tool life (T) - by count (C) - by wear (W) The wear monitoring is configured via a machine data item. Please refer to the machine manufacturer's instructions.
Setpoint	Setpoint for tool life, workpiece count, or wear.
Prewarning limit	Specification of the tool life, workpiece count or wear at which a warning is displayed.

Column heading	Meaning
Tool life Workpiece count Wear * *Parameter depends on selection in TC	Tool life Number of workpieces Tool wear
G	The tool is disabled when the checkbox is selected.

Icons in the wear list

Icon/ Designation		Meaning
Tool type		
Red "X"		The tool is disabled.
Yellow triangle pointing downward		The prewarning limit has been reached.
Yellow triangle pointing upward		The tool is in a special state. Place the cursor on the marked tool. A tooltip provides a short description.
Green border		The tool is preselected.
Magazine/location number		
Green double arrow		The magazine location is positioned at the change position.
Gray double arrow (configurable)		The magazine location is positioned at the loading position.
Red "X"		The magazine location is disabled.

Procedure



1. Select the "Parameter" operating area.

2. Press the "Tool wear" softkey.

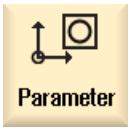
6.6.1 Reactivating a tool

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

Requirements

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

Procedure



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 canceled.

Reactivating and positioning

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

Reactivation of all monitoring types

When the "Reactivation of all monitoring types" function is configured, all the monitoring types set in the NC for a tool are reset during reactivation.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

References

HMI sl / SINUMERIK 840D sl Commissioning Manual

Multiple load points

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

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

6.7 Tool data OEM

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

References:

For more information, see the following references:
/IHsl/, HMI sl / SINUMERIK 840D sl Commissioning Manual

Procedure

1. Select the "Parameter" operating area.

2. Press the "OEM Tool" soft key.

6.8 Magazine

Tools are displayed with their magazine-related data in the magazine list. Here, you can take specific actions relating to the magazines and the magazine locations.

Individual magazine locations can be location-coded or disabled for existing tools.

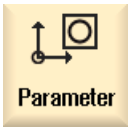
Tool parameters

Column heading	Meaning
Location  *  * * If activated in magazine selection	Magazine/location number • Spindle location as an icon. • 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. If there is only one magazine, only the location number is displayed.
Type	Tool type Depending on the tool type (represented by an icon), certain tool offset data is enabled.
Tool name	The tool is identified by the name and the replacement tool number. You can enter the name as text or number.
ST	Replacement tool number (for replacement tool strategy).
D	Cutting edge number.
G	Disabling of the magazine location.
Mag.loctype	Display of magazine location type.
Tool.loctype	Display of tool location type of tool.
Ü	Marking of a tool as oversized. The tool occupies two half locations left, two half locations right, one half location top and one half location bottom in a magazine.
P	Fixed location coding. The tool is permanently assigned to this magazine location.

Magazine list icons

Icon/ Designation		Meaning
Tool type		
Red "X"		The tool is disabled.
Yellow triangle pointing downward		The prewarning limit has been reached.
Yellow triangle pointing upward		The tool is in a special state. Place the cursor on the marked tool. A tooltip provides a short description.
Green border		The tool is preselected.
Magazine/location number		
Green double arrow		The magazine location is positioned at the change position.
Gray double arrow (configurable)		The magazine location is positioned at the loading position.
Red "X"		The magazine location is disabled.

Procedure



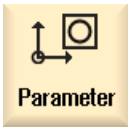
1. Select the "Parameter" operating area.

2. Press the "Magazine" softkey.

6.8.1 Position magazine

You can position magazine locations directly on the loading point.

Procedure



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 "Loading Point Selection" window appears after pressing the "Position magazine" softkey.

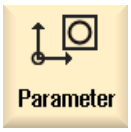
Select the desired loading point in this window and confirm your selection with "OK" to position the magazine location at the loading point.

6.8.2 Relocating a tool

Tools can be directly relocated within magazines to another magazine location, which means that you do not have to unload tools from the magazine in order to load them into a different location.

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

Procedure



1. Select the "Parameter" operating area.

2. Press the "Magazine" softkey.

3. Position the cursor on the tool that you wish to relocate to a different magazine location.

4. Press the "Relocate" softkey.

   The "... relocate from location ... to ..." window appears on the screen. The "... location" field is initialized with the number of the first empty magazine location.

5. Press the "OK" softkey to relocate the tool to the suggested location.

   - OR -

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

   - OR -

   Press the "Spindle" softkey to load a tool into the spindle and press the "OK" softkey.

   The tool is moved to the specified magazine location or the spindle.

Several magazines

If you have set up several magazines, then the "...relocate from magazine... location... to..." window appears after pressing the "Relocate" softkey.

Select the desired magazine and location, and confirm your selection with "OK" to load the tool.

6.9 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 quickly in the lists.

Procedure

1. Select the "Parameter" operating area.

2. Press the "Tool list", "Tool wear" or "Magazine" softkey.

...

3. Press the ">>" 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 "Acc. to type" softkey to display the tools arranged by tool type. Identical types (e.g. milling cutters) are sorted according to their radius value.

- OR -

Press the "Acc. to name" softkey to display the tool names in alphabetical order.

The replacement tool numbers are used to sort tools with the same names.

- OR -

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

The list is sorted according to the specified criteria.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Program management 7

7.1 Overview

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.

NOTICE
Execution from USB FlashDrive Direct execution from a USB FlashDrive is not recommended. There is no protection against contact problems, falling out, breakage through knocking or unintentional removal of the USB FlashDrive during operation. Disconnecting it during operation will result in the stopping of the machining and thus to the workpiece being damaged.

Storage for programs

Possible storage locations are:

- NC
- Local drive
- Network drives
- USB drives

Software options

You need the "Additional 256 MB HMI user memory on CF card of NCU" option in order to display the "Local drive" softkey.

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

Data exchange 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:

Directory

Program

All directories have a plus sign when the program manager is called for the first time.

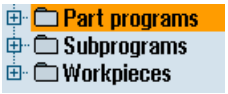


Figure 7-1 Program directory in the program manager

The plus sign in front of empty directories is removed after they have been read for the first time.

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 (without accents), numbers, and underscores.

- Type

  Directory: WPD

  Program: MPF

  Subprogram: SPF

  Initialization programs: INI

  Program lists:

  Tool data: TOA

Magazine assignment: TMA

Zero points: UFR

R parameters: RPA

Global user data/definitions: GUD

Setting data: SEA

Protection zones: PRO

Sag: CEC

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

7.1.1 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.

7.1.2 Local drive

The workpieces, main programs and subprograms 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).

Software options

You need the "Additional 256 MB HMI user memory on CF card of NCU" option in order to display the "Local drive" softkey.

Procedure

1. Select the "Program manager" operating area.

2. Press the "Local drive" software key.

7.1.3 USB drives

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

NOTICE
Execution from USB FlashDrive Direct execution from the USB FlashDrive is not recommended.

Procedure

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.

7.2 Opening and closing a program

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

With programs that are in the NCK memory, navigation is already possible when opening. The program blocks can only be edited when the program has been opened completely. You can follow the opening of the program in the dialog line.

With programs that are opened via local network, USB FlashDrive or network connections, navigation is only possible when the program has been opened completely. A progress message box is displayed when opening the program.

Procedure

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 program.

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

4. Now make the necessary program changes.

5. Press the "NC Select" softkey to switch to the "Machine" operating area and begin execution.

   When the program is running, the softkey is deactivated.

Closing the program

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

- OR -

If you are at the start of the first line of the program, press the "Cursor left" key to close the program and the editor.

To reopen a program you have exited with "Close", press the "Program" key.

Note

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

7.3 Executing a program

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

Program selection

Select the workpieces (WPD), main programs (MPF) or subprograms (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 refer to the machine manufacturer's specifications.

Procedure

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 "Select" softkey.

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

   - OR -

If the selected program is already opened in the "Program" operating area,

Press the "Execute NC" softkey.

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.

7.4 Creating a directory/program/joblist/program list

7.4.1 Creating a new directory

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

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

Note

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

All letters (except accented characters), 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.

Procedure

1. Select the "Program manager" operating area.

2. Select your chosen storage medium, i.e. a local or USB drive.

3. If you want to create a new directory in the local network, place the cursor on the topmost folder and press the "New" and "Directory" softkeys.
   The "New Directory" window appears.

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

7.4.2 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. If necessary, select a template if any are available.

5. Enter the desired workpiece name, select a template, if necessary, 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.

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

   The program will open in the editor.

7.4.3 Creating a new G code program

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

Procedure

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" softkey.

   The "New G Code Program" window appears.

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

   If you are in the NC memory and have selected the file "Subprograms" or "Part programs", you can create only one subprogram (SPF) or main program (MPF) respectively.

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 all letters (except accented characters), digits and underscores (_).

7.4.4 Storing any new file

In each directory or subdirectory you can create a file in any format that you specify.

This does not apply to the NC memory. Here you can create the following file types under a workpiece using the "Any" softkey.

Procedure

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. Select a file type from the "Type" selection field (for example, "Definitions GUD") and enter the name of the file to be created when you have selected a workpiece directory in the NC memory.

   The file automatically has the selected file format.

   - OR -

Enter a name and file format for the file to be created (e.g. My_Text.txt).

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 (_).

5. Press the "OK" softkey.

7.4.5 Creating a Joblist

For every workpiece, you can create a job list for extended workpiece selection.

In the job list, you specify instructions for program selection in different channels.

Syntax

The job list contains the SELECT instructions.

SELECT <program> CH=<channel number> [DISK]

The SELECT instruction selects a program for execution in a specific NC channel. The selected program must be loaded into the working memory of the NC. The DISK parameter enables the selection of external execution (CF card, USB data carrier, network drive).

- <Program>

  Absolute or relative path specification of the program to be selected.

  Examples:

  – //NC/WCS.DIR/SHAFT.WPD/SHAFT1.MPF

  – SHAFT2.MPF

- <Channel number>

  Number of the NC channel in which the program is to be selected.

  Example:

  CH=2

- [DISK]

  Optional parameter for programs that are not in the NC memory and are to be executed "externally".

  Example:

  SELECT //remote/myshare/shaft3.mpf CH=1 DISK

Comments

Comments are identified in the job list by ";" at the start of the line or by round brackets.

Template

You can select a template from Siemens or the machine manufacturer when creating a new job list.

Executing a workpiece

If the "Select" softkey is selected for a workpiece, the syntax of the associated job list is checked and then executed. The cursor can also be placed on the job list for selection.

Procedure

1. Select the "Program manager" operating area.

2. Press the "NC" softkey, and in directory "Workpieces" place the cursor on the program for which you wish to create a job list.

3. Press the "New" and "Any" softkeys.
   The "Any New Program" window opens.

4. Select entry "Job list JOB" from the "Type" selection field and enter a name and press the "OK" softkey.

7.4.6 Creating a program list

You can also enter programs in a program list that are then selected and executed from the PLC.

The program list may contain up to 100 entries.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

Procedure

1. Select the "Program manager" operating area.

2. Press the menu forward key and the "Program list" softkey.
   The "Program List" window appears.

3. Place the cursor in the desired line (program number).

4. Press the "Select program" softkey.
   The "Programs" window opens. The data tree of the NC memory with workpiece, part program and subprogram directory is displayed.

5. Place the cursor on the desired program and press the "OK" softkey.
   The selected program is inserted in the first line of the list together with its path.
   - OR -
   Enter the program name directly in the list.
   If you are making entries manually, check that the path is correct (e.g. //NC/WKS.DIR/MEINPROGRAMM.WPD/MEINPROGRAMM.MPF).
   //NC and the extension (.MPF) may be added automatically.
   With multi-channel machines, you can specify in which channel the program is to be selected.

6. To remove a program from the list, place the cursor on the appropriate line and press the "Delete" softkey.
   - OR -

   To delete all programs from the program list, press the "Delete all" softkey.

7.5 Creating templates

You can store your own templates to be used for creating part programs and workpieces. These templates provide the basic framework for further editing.

You can use them for any part programs or workpieces you have created.

Storage location for templates

The templates used to create part programs or workpieces are stored in the following directories:

HMI Data/Templates/Manufacturer/Part programs or Workpieces

HMI Data/Templates/User/Part programs or Workpieces

Procedure

1. Select the "Startup" operating area.

2. Press the "System data" softkey.

3. Position the cursor on the file that you wish to store as a template and press the "Copy" softkey.

4. Select the directory in which you want to store the data - "Part programs" or "Workpieces" - and press the "Paste" softkey.

   Stored templates can be selected when a part program or a workpiece is being created.

7.6 Displaying the program in the Preview.

You can show the content on a program in a preview before you start editing.

Procedure

1. Select the "Program manager" operating area.

2. Select a storage location and place the cursor on the relevant program.

3. Press the ">>" and "Preview window" softkeys.
   The "Preview: ..." window opens.

4. Press the "Preview window" softkey again to close the window.

7.7 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 canceled.

Procedure

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.

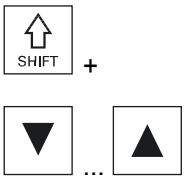
4. Select the required directories/programs with the cursor keys or mouse.

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 canceled.

Selecting via keys

Key combination	Meaning
	Renders or expands a selection. You can only select individual elements.
	Renders a consecutive selection.
	A previously existing selection is canceled.

Selecting with the mouse

Key combination	Meaning
Left mouse	Click on element: The element is selected. A previously existing selection is canceled.
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.

7.8 Copying and pasting a directory/program

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 pasted in a different location.

Note

You can only paste 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 accented characters), 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.

Procedure

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" softkey.

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

5. Press the "Paste" softkey.

   If a directory/program of the same name already exists in this directory, you are are informed. You are requested to assign a new name, otherwise the directory/program is assigned a name by the system.

   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 paste 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.

7.9 Deleting a directory/program

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.

Procedure

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 ">>" and "Delete" softkeys.

A prompt appears as to 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.

7.10 Changing 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.

Changing access rights for NC data

Access rights for execution, writing, listing and reading are displayed in the "Properties" window.

- Execute: This is used for the selection
- Write: Controls the changing and deletion of a file or a directory

You can set the access rights from keyswitch 0 to the current access level. If an access level is higher than the current access level, it cannot be changed.

Procedure

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. Press the ">>" and "Properties" softkeys.
   The "Properties from ..." window appears.

4. Enter any necessary changes.

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

7.11 Backing up data

7.11.1 Creating an archive

If you only want to backup specific data, then you can select the desired files directly from the data tree and generate an archive.

You can display the contents of the selected files (XML, ini, hsp, syf files, programs) in a preview.

You can display information about the file, such as path, name, date of creation and change, in a Properties window.

Requirement

The access rights depend on the relevant areas and range from protection level 7 (keyswitch position 0) to protection level 2 (password: Service).

Storage locations

- On the CompactFlash card under
  /**User**/sinumerik/data/archive, or
  /**OEM**/sinumerik/data/archive
- All configured logical drives (USB, network drives)

Software option

You need the "Additional 256 MB HMI user memory on CF card of NCU" option in order to store the archives on the CompactFlash card in the "User" area (User/sinumerik/data/...).

NOTICE
USB FlashDrive
USB FlashDrives are not suitable as persistent memory media.

Procedure

1. Select the "Startup" operating area.

2. Press the "System data" softkey.
   The data tree opens.
3. In the data tree, select the required files from which you want to generate an archive.
   - OR -

   If you want to back up several files or directories, press the "Select" softkey and, using the cursor keys or the mouse, select the required directories or files.

4. If you press the ">>" softkey, further softkeys are displayed on the vertical bar.

   Press the "Preview window" softkey.
   The contents of the selected file are displayed in a small window.
   Press the "Preview window" softkey again to close the window.

   Press the "Properties" softkey.
   Information about the selected file is displayed in a small window.
   Press the "OK" softkey to close the window.

   Press the "Generate archive" softkey.
   The "Generate Archive: Select Archiving" window opens.
   All the files to be archived and the storage path are displayed.
   - OR -

5. Select the required location for archiving and press the "New directory" softkey to create a suitable subdirectory.
   The "New Directory" window appears.

6. Enter the required name and press the "OK" softkey.
   The directory is created subordinate to the selected folder.

7. Press the "OK" softkey.
   The "Generate Archive: Name" window opens.

8. Enter the required name and press the "OK" softkey.
   An archive file in .arc format is created in the selected directory.

7.11.2 Reading in an archive

If you want to read in a specific archive, you can select this directly from the data tree.

Procedure

1. Select the "Startup" operating area.

2. Press the "System data" softkey.

3. Below the "Archive" directory in the data tree, select the file you want to read in.

4. Press the "Read in" softkey.

...

5. Press the "OK" or "Overwrite all" softkey to overwrite existing files.

   - OR -

   Press the "Do not overwrite" softkey if you do not want to overwrite existing files.

   - OR -

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

   The "Read In Archive" window opens and a progress message box appears for the read-in process.

6. Press the "Cancel" softkey to cancel the read-in process.

7.12 EXTCALL

The EXTCALL command can be used to access files on a local drive, USB data carriers or network drives from a part program.

The programmer can set the source directory with the setting data SD42700 EXT_PROG_PATH and then specify the file name of the subprogram to be loaded with the EXTCALL command.

Supplementary conditions

The following supplementary conditions must be taken into account with EXTCALL calls:

- You can only call files with the MPF or SPF extension via EXTCALL from a network drive.
- The files and paths must comply with the NCK naming conventions (max. 25 characters for the name, 3 characters for the identifier).
- A program is found on a network drive with the EXTCALL command if
  - With SD42700 EXT_PROG_PATH: The search path refers to the network drive or a directory contained on the network drive. The program must be stored directly on that level, no subdirectories are searched.
  - Without SD42700: The correct location of the program is specified in the EXTCALL call itself by means of a fully qualified path that can also point to a subdirectory of the network drive.

Examples of EXTCALL calls

The setting data can be used to perform a targeted search for the program.

- Call of USB drive on TCU (USB storage device on interface X203), if SD42700 is empty: e.g. EXTCALL "//TCU/TCU1 /X203 ,1/TEST.SPF"

  - OR -

  Call of USB drive on TCU (USB storage device on interface X203), if SD42700 "//TCU/TCU1 /X203 ,1" contains: EXTCALL "TEST.SPF"
- Call of USB front connection (USB FlashDrive), if SD42700 is empty: e.g. EXTCALL "//ACTTCU/FRONT,1/TEST.SPF"

  - OR -

  Call of USB front connection (USB FlashDrive), if SD42700 "//ACTTCU/FRONT,1" contains: EXTCALL "TEST.SPF"
- Call of network drive, if SD42700 is empty: e.g. EXTCALL "//Computer name/Enabled drive/TEST.SPF"

  - OR -

  Call of network drive, if SD42700 "//Computer name/Enabled drive" contains: EXTCALL "TEST.SPF"
- Use of the HMI user memory (local drive):

The HMI user memory is subdivided into part programs (mpf.dir), subprograms (spf.dir) and workpieces (wks.dir) with the respective workpiece directories (.wpd).

– SD42700 is empty: EXTCALL "TEST.MPF"

  The same search sequence is used on the CompactFlash card as in the NCK part program memory.

– Specification of the complete path: e.g. EXTCALL "card/user/sinumerik//data/prog/spf.dir/test"

  A search is performed for the specified file.

Software options

You need the "Additional 256 MB HMI user memory on CF card of NCU" option in order to display the "Local drive" softkey.

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

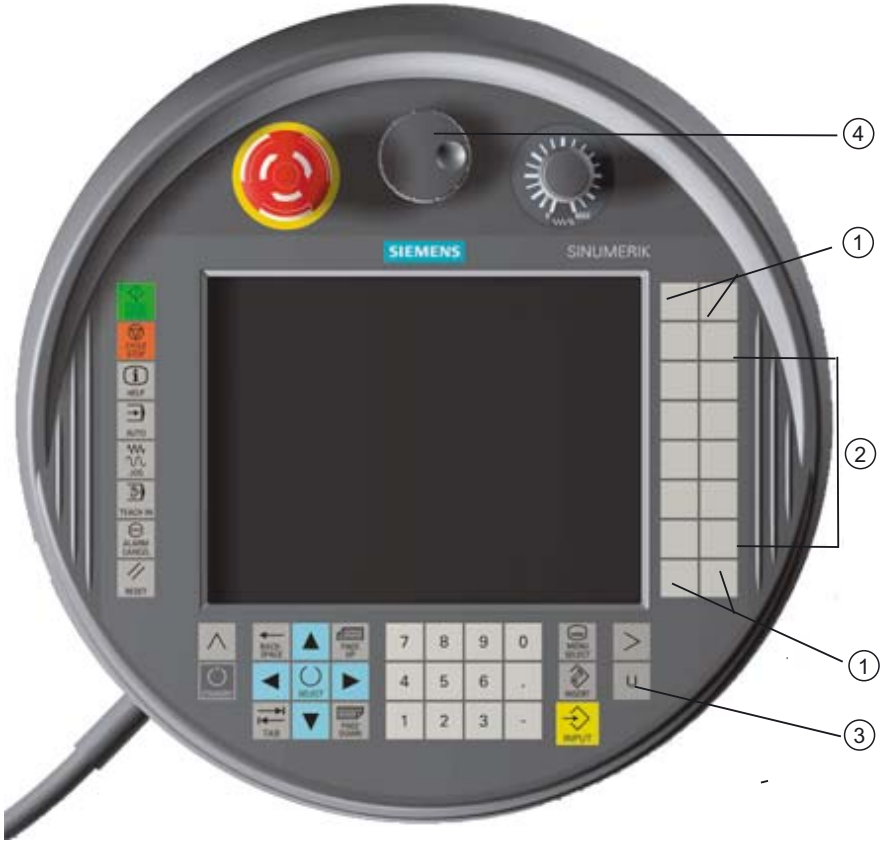
NOTICE
Execution from USB FlashDrive Direct execution from a USB FlashDrive is not recommended. There is no protection against contact problems, falling out, breakage through knocking or unintentional removal of the USB FlashDrive during operation. Disconnecting it during operation will result in immediate stopping of the machining and, thus, to the workpiece being damaged.

HT 8

8

8.1 HT 8 overview

The mobile SINUMERIK HT 8 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.

References

For more information about connection and startup of the HT 8, see the following references:

CNC Commissioning: SINUMERIK 840D sl, Operator Components and Networking

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 HT 8 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.

Note

PLC interface signals that are triggered via the softkeys of the machine control panel menus are edge triggered.

Enabling button

The HT 8 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

To traverse the axes of your machine using the traversing keys of the HT 8, 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.

Changing the channel

- 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).
- The "1… n CHANNEL" softkey is available in the machine control panel menu that can be reached via the user menu key "U".

Operating-area switchover

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

Handwheel

The HT 8 is available with a hand wheel.

References

For information about connecting the hand wheel, refer to:

CNC Commissioning: SINUMERIK 840D sl, Operator Components and Networking

See also

Channel switchover (Page 45)

8.2 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 six axes on the touch panel by default.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

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 refer to the machine manufacturer's specifications.

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

8.3 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.

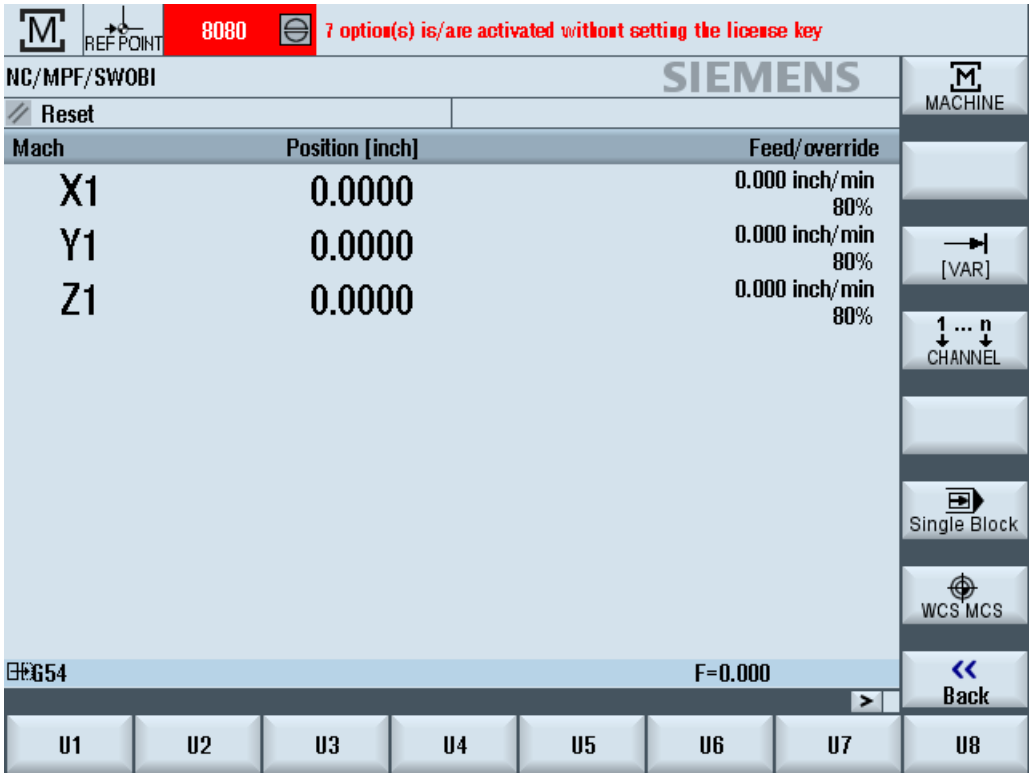
See chapter "Controls on the machine control panel" for a description of the individual keys.

Note

PLC interface signals that are triggered via the softkeys of the machine control panel menus are edge triggered.

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 can expand the user softkey bar to display eight additional softkeys via the menu forward key.

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
"1… n CHANNEL" softkey	Change the channel
"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.

8.4 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.

Procedure

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 -

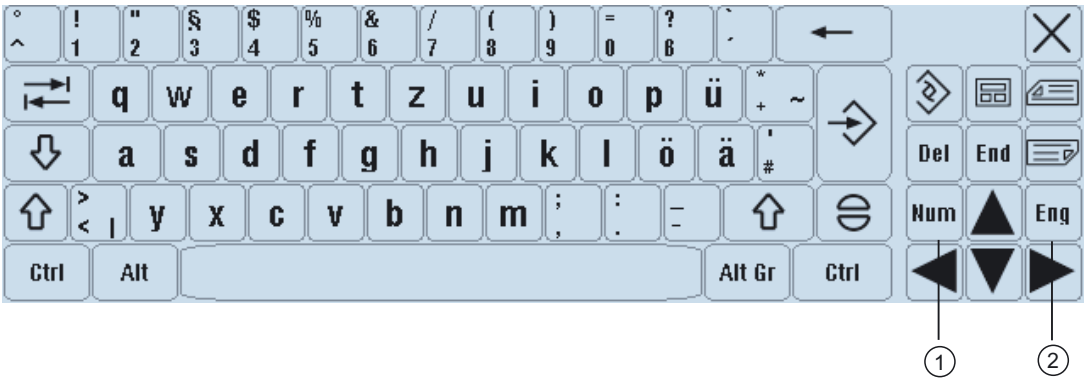
Position the cursor on an another operator element.

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 pressing the empty bar next to the "Close window" icon with your finger or a stylus and moving it back and forth.

Special keys on the virtual keyboard



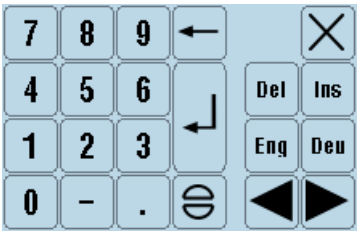
1 Num:

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.

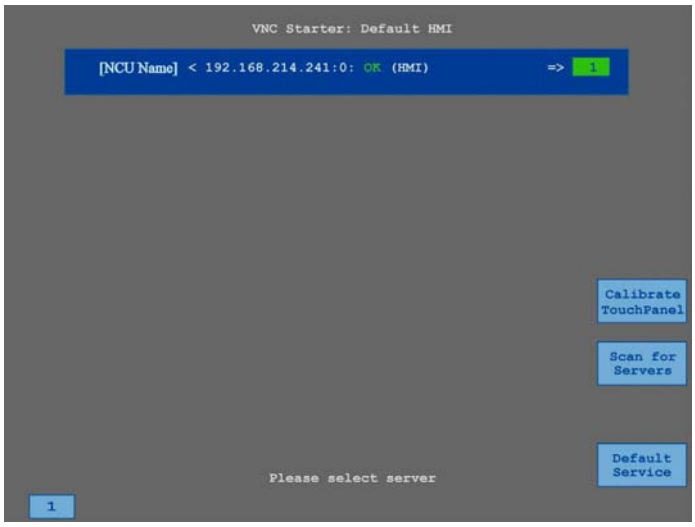
8.5 Calibrating the touch panel

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

Note

Recalibration

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



Procedure

...

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 9

9.1 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
- Deletion criterion

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

Procedure

1. Select the "Diagnostics" operating area.

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

3. Position the cursor on an alarm.

4. Press the key that is specified as acknowledgement symbol to delete the alarm.

   - OR -

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

   - OR -

   Press the "Acknowledge alarm" softkey to delete a PLC alarm of the SQ type (alarm number as of 800000).

   The softkeys are activated when the cursor is on the corresponding alarm.

Acknowledgement symbols

Symbol	Meaning
	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 refer to the machine manufacturer's specifications.

9.2 Displaying PLC and NC variables

The "PLC/NC Status" window supports the observation and modification of PLC memory locations and NC system variables.

You can also modify PLC memory locations.

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

$AA_IM[1]

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 "Diagnostics" operating area.

2. Press the "Variab. view" softkey.

   The "Variables" window appears.

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 required format with "SELECT".

6. 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.

7. Press the "OK" 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.

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

9.3 Displaying an alarm log

A list of all the alarms and messages that have occurred so far are listed in the "Alarm Log" window.

Up to 500 administered, incoming and outgoing events are displayed in chronological order.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

1. Select the "Diagnostics" operating area.

2. Press the "Alarm log" softkey.

   The "Alarm Log" window opens.

   All the raised and cleared events that have occurred since HMI sl was started are listed.

3. Press the "Display new" softkey to update the list of displayed alarms/messages.

4. Press the "Save Log" softkey.

   The log that is currently displayed is stored as text file alarmlog.txt in the system data in directory card/user/sinumerik/hmi/log/alarm_log.

9.4 Displaying messages

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.

9.5 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.

Procedure

1. Select the "Diagnosis" operating area.

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

3. Select the component for which you would like more information.

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

9.6 Creating screenshots

You can create screenshots of the current user interface directly on the TCU.

Each screenshot is saved as a file and stored in a folder.

You can use the default storage path or specify a path yourself.

Procedure

Ctrl + P	Press the <Ctrl+P> key combination. A screenshot of the current user interface is created in .png format. The file names assigned by the system run in ascending order from "SCR_SAVE_0001.png" to "SCR_SAVE_9999". You can create up to 9,999 screenshots.

Specifying the storage location

The storage location of the screenshots is defined in the "systemconfiguration.ini" file. The file is on the CF card in the following directory: Siemens/sinumerik/hmi/cfg.

To change the storage location, copy the "systemconfiguration.ini" file to the "oem" or "user" directory and change the path.

The default setting is:

```
[miscellaneous]
prt_screen_save_directory=/user/sinumerik/hmi/log/screenshot
```

Opening a file

1. Select the "Startup" operating area.

2. Press the "System data" softkey and open the HMI Data/Logs/Screenshots folder.

   You can copy the files to a Windows PC and open them there using, e.g. the "Office Picture Manager" program.

Appendix

A

A.1 Feedback on the documentation

This document will be continuously improved with regard to its quality and ease of use. Please help us with this task by sending your comments and suggestions for improvement via e-mail or fax to:

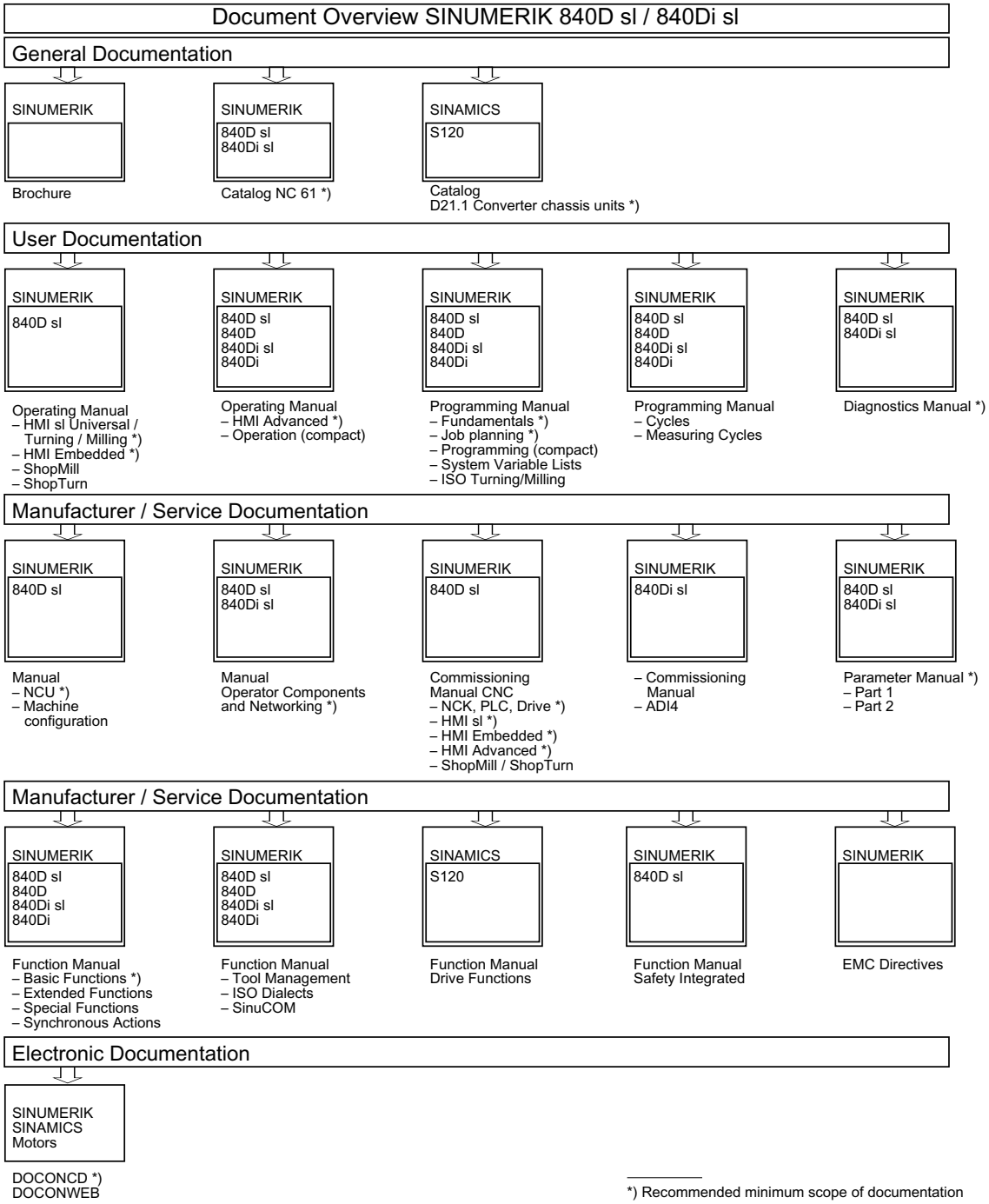
E-mail: mailto:docu.motioncontrol@siemens.com

Fax: +49 9131 - 98 63315
Please use the fax form on the back of this page.

To	Sender
SIEMENS AG A&D MC MS1 P.O. Box 3180	Name:
	Address of your Company/Dept.
91050 ERLANGEN, GERMANY	Address:
	Zip code: City:
	Phone: /
Fax: +49 9131 - 98 63315 (Documentation)	Telefax: /

Suggestions and/or corrections

A.2 Overview


Document Overview SINUMERIK 840D sl / 840Di sl
General Documentation
SINUMERIK
Brochure
SINUMERIK
840D sl
840Di sl
Catalog NC 61 *)
SINAMICS
S120
Catalog
D21.1 Converter chassis units *)
User Documentation
SINUMERIK
840D sl
Operating Manual
– HMI sl Universal / Turning / Milling *)
– HMI Embedded *)
– ShopMill
– ShopTurn
SINUMERIK
840D sl
840D
840Di sl
840Di
Operating Manual
– HMI Advanced *)
– Operation (compact)
SINUMERIK
840D sl
840D
840Di sl
840Di
Programming Manual
– Fundamentals *)
– Job planning *)
– Programming (compact)
– System Variable Lists
– ISO Turning/Milling
SINUMERIK
840D sl
840D
840Di sl
840Di
Programming Manual
– Cycles
– Measuring Cycles
SINUMERIK
840D sl
840Di sl
Diagnostics Manual *)
Manufacturer / Service Documentation
SINUMERIK
840D sl
Manual
– NCU *)
– Machine configuration
SINUMERIK
840D sl
840Di sl
Manual
Operator Components and Networking *)
SINUMERIK
840D sl
Commissioning Manual CNC
– NCK, PLC, Drive *)
– HMI sl *)
– HMI Embedded *)
– HMI Advanced *)
– ShopMill / ShopTurn
SINUMERIK
840Di sl
– Commissioning Manual
– ADI4
SINUMERIK
840D sl
840Di sl
Parameter Manual *)
– Part 1
– Part 2
Manufacturer / Service Documentation
SINUMERIK
840D sl
840D
840Di sl
840Di
Function Manual
– Basic Functions *)
– Extended Functions
– Special Functions
– Synchronous Actions
SINUMERIK
840D sl
840D
840Di sl
840Di
Function Manual
– Tool Management
– ISO Dialects
– SinuCOM
SINAMICS
S120
Function Manual
Drive Functions
SINUMERIK
840D sl
Function Manual
Safety Integrated
SINUMERIK
EMC Directives
Electronic Documentation
SINUMERIK
SINAMICS
Motors
DOCONCD *)
DOCONWEB
*) Recommended minimum scope of documentation

Index

Q

R

S

T

U

V

W