SIEMENS

SINUMERIK

SINUMERIK 840D sl/828D Universal

Operating Manual

Valid for:

Controls
SINUMERIK 840D sl / 840DE sl / 828D

Software
CNC system software for 840D sl/ 840DE sl   V4.5 SP2
SINUMERIK Operate   V4.5 SP2
for PCU/PC

Legal information

Warning notice system

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
indicates that minor personal injury can result if proper precautions are not taken.

NOTICE
indicates that property damage can result if proper precautions are not taken.

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 product/system described in this documentation may be operated only by **personnel qualified** for the specific task in accordance with the relevant documentation, in particular its warning notices and safety instructions. Qualified personnel are those who, based on their training and experience, are capable of identifying risks and avoiding potential hazards when working with these products/systems.

Proper use of Siemens products

Note the following:

⚠ WARNING
Siemens products may only be used for the applications described in the catalog and in the relevant technical documentation. If products and components from other manufacturers are used, these must be recommended or approved by Siemens. Proper transport, storage, installation, assembly, commissioning, operation and maintenance are required to ensure that the products operate safely and without any problems. The permissible ambient conditions must be complied with. The information in the relevant documentation must be observed.

Trademarks

All names identified by ® are registered trademarks of 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.

Preface

SINUMERIK documentation

The SINUMERIK documentation is organized in the following categories:

- General documentation
- User documentation
- Manufacturer/service documentation

Additional information

You can find information on the following topics at www.siemens.com/motioncontrol/docu:

- Ordering documentation/overview of documentation
- Additional links to download documents
- Using documentation online (find and search in manuals/information)

Please send any questions about the technical documentation (e.g. suggestions for improvement, corrections) to the following address:

docu.motioncontrol@siemens.com

My Documentation Manager (MDM)

Under the following link you will find information to individually compile OEM-specific machine documentation based on the Siemens content:

www.siemens.com/mdm

Training

For information about the range of training courses, refer under:

- www.siemens.com/sitrain

  SITRAIN - Siemens training for products, systems and solutions in automation technology

- www.siemens.com/sinutrain

  SinuTrain - training software for SINUMERIK

FAQs

You can find Frequently Asked Questions in the Service&Support pages under Product Support. http://support.automation.siemens.com

SINUMERIK

You can find information on SINUMERIK under the following link:

www.siemens.com/sinumerik

Target group

This documentation is intended for users of universal machines running the SINUMERIK Operate software.

Benefits

The operating manual helps users familiarize themselves with the control elements and commands. Guided by the manual, users are capable of responding to problems and taking corrective action.

Standard scope

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

Furthermore, for the sake of clarity, 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

You will find telephone numbers for other countries for technical support in the Internet under http://www.siemens.com/automation/service&support

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:
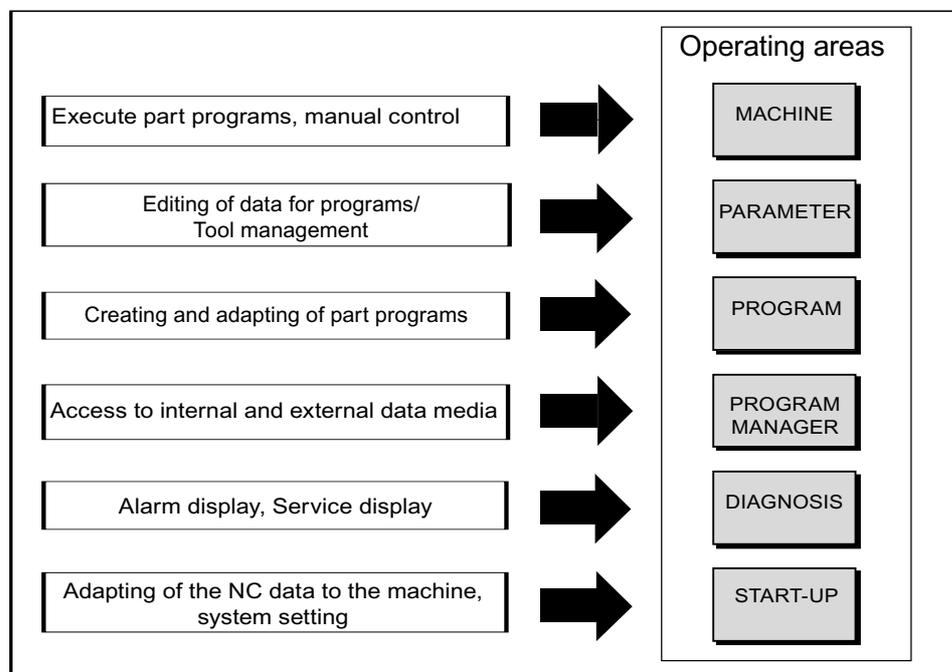
1.2 Operator panel fronts

1.2.1 Overview

Introduction

The display (screen) and operation (e.g. hardkeys and softkeys) of the SINUMERIK Operate user interface use the operator panel front.

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

Operator controls and indicators
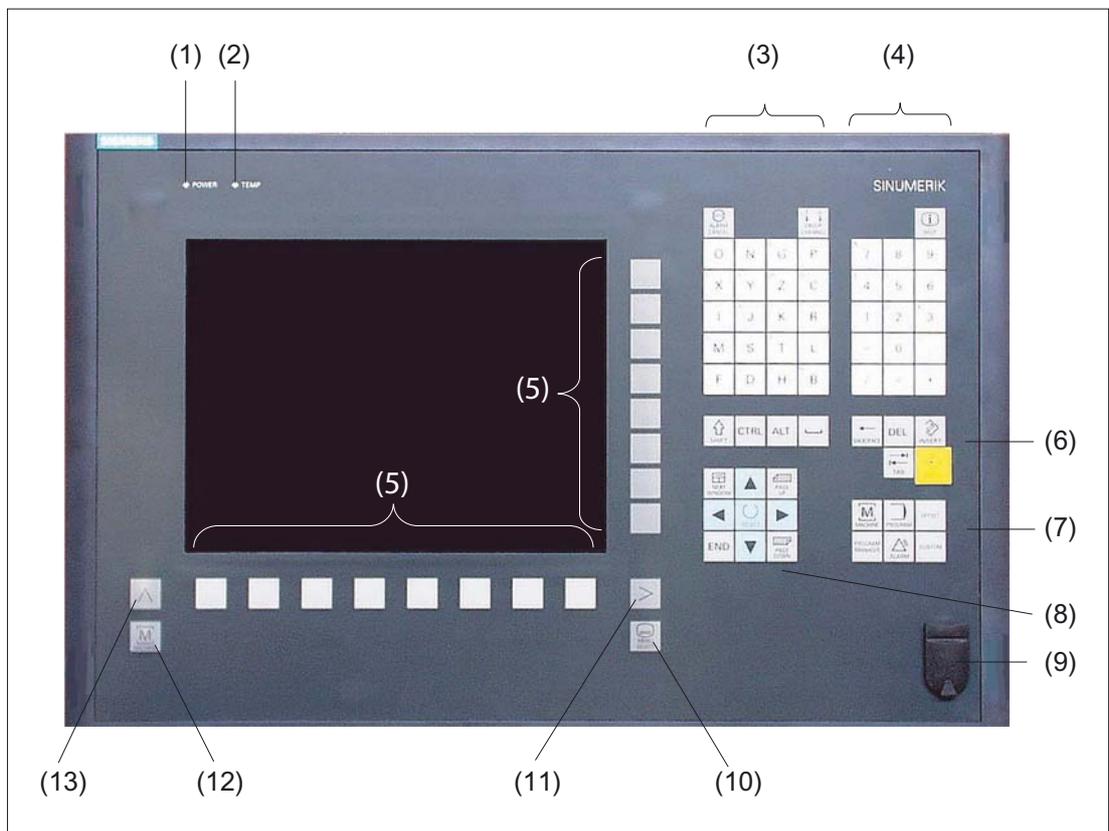
1	Status LED: POWER
2	Status LED: TEMP
	(illuminated LEDs indicate increased wear)
3	Alphabetic key group
4	Numerical key group
5	Softkeys

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

Figure 1-1   View of OP 010 operator panel front

References

A more precise description as well as a view of the other operator panel fronts that can be used may be found in the following reference:

Operator Components and Networking Manual; SINUMERIK 840D sl/840Di sl

1.2.2 Keys of the operator panel

The following keys and key combinations are available for operation of the controller and the machine tool.

Keys and key combinations

Key	Function

Key   Function

\<ALARM CANCEL\>

Cancels alarms and messages that are marked with this symbol.

\<CHANNEL\>

Advances for several channels.

\<HELP\>

Calls the context-sensitive online help for the selected window.

\<NEXT WINDOW\> *

- Toggles between the windows.
- For a multi-channel view or for a multi-channel functionality, switches within a channel gap between the upper and lower window.
- Selects the first entry in selection lists and in selection fields.
- Moves the cursor to the beginning of a text.

* on USB keyboards use the \<Home\> or \<Pos 1\> key

\<NEXT WINDOW\> + \<SHIFT\>

- Selects the first entry in selection lists and in selection fields.
- Moves the cursor to the beginning of a text.
- Selects a contiguous selection from the current cursor position up to the target position.
- Selects a contiguous selection from the current cursor position up to the beginning of a program block.

\<NEXT WINDOW\> + \<ALT\>

- Moves the cursor to the first object.
- Moves the cursor in the first column of a table row.
- Moves the cursor to the beginning of a program block.

\<NEXT WINDOW\> + \<CTRL\>

- Moves the cursor to the beginning of a program.
- Moves the cursor in the first row of the current column.

<NEXT WINDOW> + <CTRL> + <SHIFT>

- Moves the cursor to the beginning of a program.
- Moves the cursor in the first row of the current column.
- Selects a contiguous selection from the current cursor position up to the target position.
- Selects a contiguous selection from the current cursor position up to the beginning of the program.

<PAGE UP>

Scrolls upwards by one page in a window.
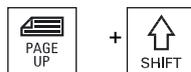
<PAGE UP> + <SHIFT>

In the program manager and in the program editor from the cursor position, selects directories or program blocks up to the beginning of the window.

<PAGE UP> + <CTRL>
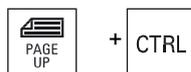
Positions the cursor to the topmost line of a window.

<PAGE DOWN>

Scrolls downwards by one page in a window.
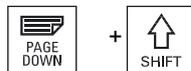
<PAGE DOWN> + <SHIFT>

In the program manager and in the program editor, from the cursor position, selects directories or program blocks up to the end of the window.

<PAGE DOWN> + <CTRL>
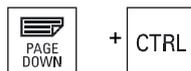
Positions the cursor to the lowest line of a window.

<Cursor right>

- Editing box

  Opens a directory or program (e.g. cycle) in the editor.
- Navigation

  Moves the cursor further to the right by one character.

<Cursor right> + <CTRL>

- Editing box

  Moves the cursor further to the right by one word.
- Navigation

  Moves the cursor in a table to the next cell to the right.

<Cursor left>

- Editing box

  Closes a directory or program (e.g. cycle) in the program editor. If you have made changes, then these are accepted.
- Navigation

  Moves the cursor further to the left by one character.

<Cursor left> + <CTRL>

- Editing box

  Moves the cursor further to the left by one word.
- Navigation

  Moves the cursor in a table to the next cell to the left.

<Cursor up>

- Editing box

  Moves the cursor into the next upper field.
- Navigation
  - Moves the cursor in a table to the next cell upwards.
  - Moves the cursor upwards in a menu screen.

<Cursor up> + <CTRL>

- Moves the cursor in a table to the beginning of the table.
- Moves the cursor to the beginning of a window.

<Cursor up> + <SHIFT>

In the program manager and in the program editor, selects a contiguous selection of directories and program blocks.

<Cursor down>

- Editing box

  Moves the cursor downwards.
- Navigation
  - Moves the cursor in a table to the next cell downwards.
  - Moves the cursor in a window downwards.

<Cursor down> + <CTRL>

- Navigation
  - Moves the cursor in a table to the end of the table.
  - Moves the cursor to the end of a window.
- Simulation
  Reduces the override.
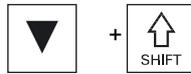
<Cursor down> + <SHIFT>

In the program manager and in the program editor, selects a contiguous selection of directories and program blocks.

<SELECT>

Switches between several specified options in selection drop-down list boxes and in selection boxes.

Activates checkboxes.

In the program editor and in the program manager, selects a program block or a program.

<SELECT> + <CTRL>

When selecting table rows, switches between selected and not selected.

<SELECT> + <SHIFT>

Selects in selection lists and in selection fields the previous entry or the last entry.

<END>

Moves the cursor to the last entry field in a window, to the end of a table or a program block.

Selects the last entry in selection lists and in selection fields.

<END> + <SHIFT>
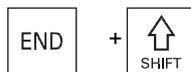
Moves the cursor to the last entry.

Selects a contiguous selection from the cursor position up to the end of a program block.
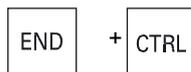
<END> + <CTRL>

Moves the cursor to the last entry in the last line of the actual column or to the end of a program.

<END> + <CTRL> + <SHIFT>

Moves the cursor to the last entry in the last line of the actual column or to the end of a program.

Selects a contiguous selection from the cursor position up to the end of a program block.

<BACKSPACE>

- Editing box

  Deletes a character selected to the left of the cursor.

- Navigation

  Deletes all of the selected characters to the left of the cursor.

<BACKSPACE> + <CTRL>
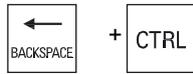
- Editing window
  Deletes a word selected to the left of the cursor.
- Navigation
  Deletes all of the selected characters to the left of the cursor.

<TAB>

- In the program editor, indents the cursor by one character.
- In the program manager, moves the cursor to the next entry to the right.

<TAB> + <SHIFT>

- In the program editor, indents the cursor by one character.
- In the program manager, moves the cursor to the next entry to the left.

<TAB> + <CTRL>

- In the program editor, indents the cursor by one character.
- In the program manager, moves the cursor to the next entry to the right.

<TAB> + <SHIFT> + <CTRL>
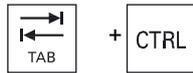
- In the program editor, indents the cursor by one character.
- In the program manager, moves the cursor to the next entry to the left.

<CTRL> + <A>
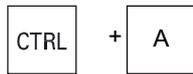
In the actual window, selects all entries (only in the program editor and program manager).

<CTRL> + <C>

Copies the selected content.

<CTRL> + <E>

Calls the "Ctrl Energy" function.

<CTRL> + <F>

Opens the search dialog in the machine data and setting data lists, when loading and saving in the MDI editor as well as in the program manager and in the system data.

<CTRL> + <G>

- Switches in the program editor for ShopMill or ShopTurn programs between the work plan and the graphic view.
- Switches in the the parameter screen between the help display and the graphic view.

CTRL	+	L	

<CTRL> + <L>

Scrolls the actual user interface through all installed languages one after the other.

CTRL	+	⇧ SHIFT	L

<CTRL> + <SHIFT> + <L>

Scrolls the actual user interface through all installed languages in the inverse sequence.

CTRL	+	P	

<CTRL> + <P>

Generates a screenshot from the actual user interface and saves it as file.

CTRL	+	S	

<CTRL> + <S>

Switches the single block in or out in the simulation.

CTRL	+	V	

<CTRL> + <V>

- Pastes text from the clipboard at the actual cursor position.
- Pastes text from the clipboard at the position of a selected text.

CTRL	+	X	

<CTRL> + <X>

Cuts out the selected text. The text is located in the clipboard.

CTRL	+	Y	

<CTRL> + <Y>

Reactivates changes that were undone (only in the program editor).

CTRL	+	Z	

<CTRL> + <Z>

Undoes the last action (only in the program editor).

CTRL	+	ALT	-	C	

<CTRL> + <ALT> + <C>

Creates a complete standard archive (.ARC) on an external data carrier (USB-FlashDrive) (for 840D sl / 828D).

Note:

Please refer to the machine manufacturer's specifications.

CTRL	+	ALT	-	S	

<CTRL> + <ALT> + <S>

Creates a complete standard archive (.ARC) on an external data carrier (USB-FlashDrive) (for 840D sl).

Creates a complete Easy Archive (.ARD) on an external data carrier (USB-FlashDrive) (for 828D).

Note:

Please refer to the machine manufacturer's specifications.

CTRL	+	ALT	-	D	

<CTRL> + <ALT> + <D>

Backs up the log files on the USB-FlashDrive. If a USB-FlashDrive is not inserted, then the files are backed-up in the manufacturer's area of the CF card.

<SHIFT> + <ALT> + <D>

Backs up the log files on the USB-FlashDrive. If a USB-FlashDrive is not inserted, then the files are backed-up in the manufacturer's area of the CF card.

<SHIFT> + <ALT> + <T>

Starts "HMI Trace".

<SHIFT> + <ALT> + <T>

Exits "HMI Trace".

<ALT> + <S>

Opens the editor to enter Asian characters.

<ALT> + <Cursor up>

Moves the block start or block end up in the editor.

<ALT> + <Cursor down>

Moves the block start or block end down in the editor.

<DEL>

- Editing box

  Deletes the first character to the right of the cursor.
- Navigation

  Deletes all characters.

<DEL> + <CTRL>

- Editing box

  Deletes the first word to the right of the cursor.
- Navigation

  Deletes all characters.

<Spacebar>

- Editing window
  Inserts a space
- Switches between several specified options in selection drop-down list boxes and in selection boxes.

<Plus>

- Opens a directory which contains the element.
- Increases the size of the graphic view for simulation and traces.

<Minus>

- Closes a directory which contains the element.
- Reduces the size of the graphic view for simulation and traces.

<Equals>

Opens the calculator in the entry fields.

<Asterisk>

Opens a directory with all of the subdirectories.

<Tilde>

Changes the sign of a number between plus and minus.

<INSERT>

- Opens an editing window in the insert mode. Pressing the key again, exits the window and the entries are undone.
- Opens a selection box and shows the selection possibilities.
- In the machining step program, enters an empty line for G code.
- Changes into the double editor or into the multi-channel view from the edit mode into the operating mode. You can return to the edit mode by pressing the key again.

<INSERT> + <SHIFT>
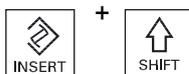
For G code programming, for a cycle call activates or deactivates the edit mode.

<INPUT>

- Completes input of a value in the entry field.
- Opens a directory or a program.
- Inserts an empty program block if the cursor is positioned at the end of a program block.
- Inserts a character to select a new line and the program block is split up into two parts.
- In the G code, inserts a new line after the program block.
- In the machining step program, inserts a new line for G code e
- Changes into the double editor or into the multi-channel view from the edit mode into the operating mode. You can return to the edit mode by pressing the key again.

<ALARM> - only OP 010 and OP 010C

Calls the "Diagnosis" operating area.

<PROGRAM> - only OP 010 and OP 010C

Calls the "Program Manager" operating area.

<OFFSET> - only OP 010 and OP 010C

Calls the "Parameter" operating area.

<PROGRAM MANAGER> - only OP 010 and OP 010C

Calls the "Program Manager" operating area.

Menu forward key

Advances in the extended horizontal softkey bar.

Menu back key

Returns to the higher-level menu.

<MACHINE>

Calls the "Machine" operating area.

<MENU SELECT>

Calls the main menu to select the operating area.

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
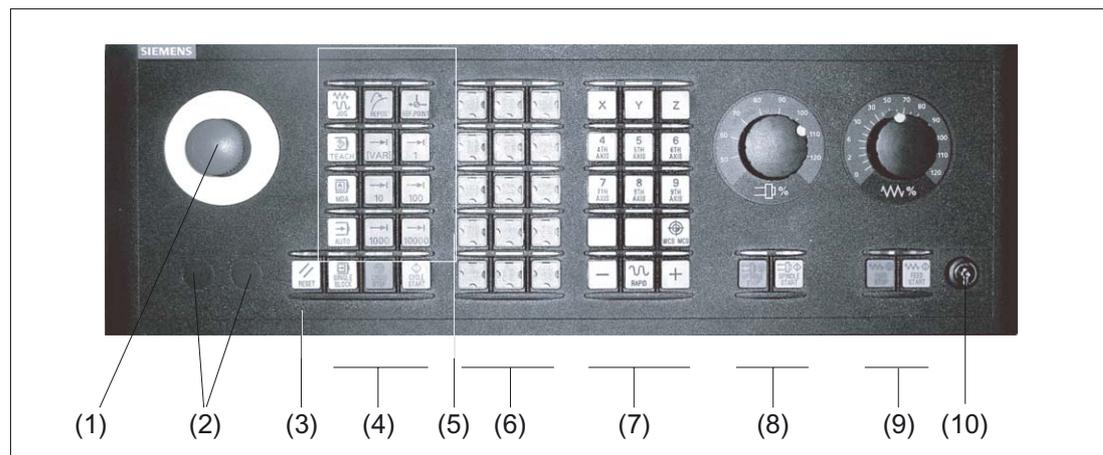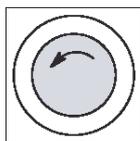
(1)		**EMERGENCY STOP button**
		Activate the button in situations where
		• life is at risk.
		• there is the danger of a machine or workpiece being damaged.
		All drives will be stopped with the greatest possible braking torque.

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.

<MDI>

Select "MDI" 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**
	X	**Axis keys**
		Selects an axis.
	...	
	Z	
	+	**Direction keys**
		Select the traversing direction.
	...	
	−	
	RAPID	**<RAPID>**
		Traverse axis in rapid traverse while pressing the direction key.
	WCS MCS	**<WCS MCS>**
		Switches between the workpiece coordinate system (WCS) and machine coordinate system (MCS).
(8)		**Spindle control with override switch**
	SPINDLE STOP	**<SPINDLE STOP>**
		Stop spindle.
	SPINDLE START	**<SPINDLE START>**
		Spindle is enabled.
(9)		**Feed control with override switch**
	FEED STOP	**<FEED STOP>**
		Stops execution of the running program and shuts down axis drives.
	FEED START	**<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)**

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

1.4 Operator interface

1.4.1 Screen layout

Overview
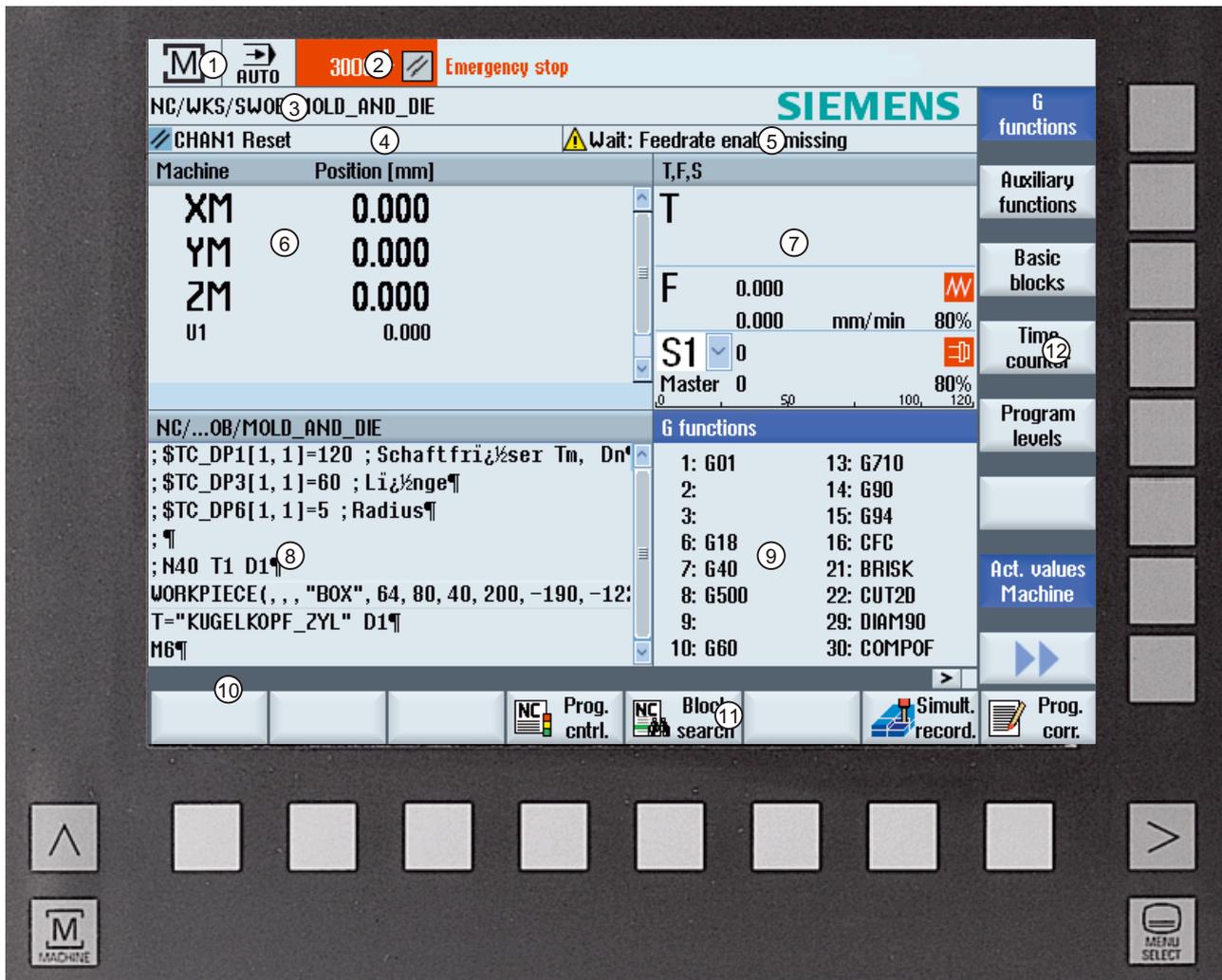
1	Active operating area and mode
2	Alarm/message line
3	Program name
4	Channel state and program control
5	Channel operational messages
6	Axis position display in actual value window

7   Display for

- Active tool T

- Current feedrate F

- Active spindle with current status (S)

- Spindle utilization rate in percent

- Name of the active tool carrier with display of a rotation in space and plane

- Name of the active kinematic transformation

8   Operating window with program block display

9   Display of active G functions, all G functions, auxiliary 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

Figure 1-3   User interface

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	
M	"Machine" operating area With touch operation, you can change the operating area here.
↑ ○	"Parameter" operating area

Display	Description
	"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	
10299 ↓ Kanal 1 Funktion Auto-R	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.
550125 Dies ist eine PLC-Meldung: Maschinentür geöffnet	NC or PLC message Message numbers and texts are shown in black lettering. An arrow indicates that several messages are active.
READY TO START	Messages from NC programs do not have numbers and appear in green lettering.

Second line

Display	Description
TEST_TEACHEN	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
CHAN1 RESET	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".
DRYPRT	Display of active program controls: PRT: no axis motion DRY: Dry run feedrate RG0: reduced rapid traverse M01: programmed stop 1 M101: programmed stop 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.

1.4.3 Actual value window

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

Work/Machine

The displayed coordinates are based on either the machine coordinate system or the workpiece coordinate system. The machine coordinate system (Machine), in contrast to the workpiece coordinate system (Work), does not take any work offsets into consideration.

You can use the "Machine 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 Work.

The ENS coordinate system corresponds to the Work coordinate system, reduced by certain components ($P_TRAFRAME, $P_PFRAME, $P_ISO4FRAME, $P_CYCFRAME), which are set by the system when machining and are then reset again. By using the ENS coordinate system, jumps into the actual value display are avoided that would otherwise be caused by the additional components.

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	
Work/Machine	Display of axes in selected coordinate system.
Position	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.
Collision monitoring (only 840D sl)	The collision avoidance is switched on for the JOG and MDI or AUTOMATIC modes and at least one active collision pair is present.
	The collision avoidance is switched off for the JOG and MDI or AUTOMATIC modes or no active collision pair is present.
Footer	Display of active work offsets and transformations.
	The T, F, S values are also displayed in the full-screen version.

See also

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 is displayed in the T, F, S window.

In addition to the "T, F, S" window name, the following information is also displayed:

Display	Meaning
BC (example)	Name of the tool carrier
Turning (example)	Name of the active kinematic transformation
	Active tool carrier rotated in the plane
	Active tool carrier swiveled in space

Tool data

Display	Meaning
T	
Tool name	Name of the current tool
Location	Location number of the current tool
D	Cutting edge of the current tool
	The tool is displayed with the associated tool type symbol corresponding to the actual coordinate system in the selected cutting edge position.
	If the tool is swiveled, then this is taken into account in the display of the cutting edge position.
	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 the current tool
R	Radius of the current tool
Z	Z value of the current tool
X	X value of the current tool

Feed data

Display	Meaning
F	
	Feed disable
	Actual feed value
	If several axes traverse, is displayed for:
	• "JOG" mode: Axis feed for the traversing axis
	• "MDA" and "AUTO" mode: Programmed axis feed
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
Symbol ⊟ (red) ↻ (green clockwise) ↺ (green counterclockwise) ⊠	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.

Note

Display of logical spindles

If the spindle converter is active, logical spindles are displayed in the workpiece coordinate system. When switching over to the machine coordinate system, the physical spindles are displayed.

Machine manufacturer

Please refer to the 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 G code blocks in the NC memory, not for external execution.
3. Press the <INSERT> key to exit the program and the edit mode again.

See also

Correcting a program (Page 98)

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.

The operating software is sub-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.

Press the <MACHINE> key to 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.
◀◀ Back	Use the "Return" softkey to close an open window.
✗ Cancel	Use the "Cancel" softkey to exit a window without accepting the entered values and return to the next highest window.
✓ Accept	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.
✓ OK	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 25)

Channel switchover (Page 57)

1.4.7 Entering or selecting parameters

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

Orange background	The input field is selected
Light orange background	The input field is in edit mode
Pink background	The 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

SELECT	1.	Keep pressing the <SELECT> key until the required setting or unit is selected.
		The <SELECT> key only works if there are several selection options available.
		- OR -
		Press the <INSERT> key.
INSERT		The selection options are displayed in a list.
	2.	Select the required setting using the <Cursor down> and <Cursor up> keys.
	3.	If required, enter a value in the associated input field.
	4.	Press the <INPUT> key to complete the parameter input.
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, you can also enter simple calculation expressions, without having to explicitly call the calculator. You can execute the four basic calculation types, work with expressions in brackets as well as generate square roots and squares.

Note

Generating square roots and squares

The extract roots and generate square functions is not available in the parameter screens of the cycles and functions in the "Program" operating area.

INSERT		Press the <INSERT> key. The insert mode is activated.
◄ ►		You can navigate within the input field using the <Cursor left> and <Cursor right> keys.
BACKSPACE DEL		Use the <BACKSPACE> and <DEL> key to delete individual characters.
SHIFT	+ <*>	Enter the multiplication characters using the <SHIFT> + <*> keys.
SHIFT	+ </>	Enter the division character using the <SHIFT> + </> keys.
SHIFT		Enter bracket expressions using the <SHIFT> + <(> and <SHIFT> + <)> keys.
SHIFT	+ <)>	
R	+ <number>	Enter "r" or "R" as well as the number x from which you would like to extract the root.
S	+ <number>	Enter "s" or "S" as well as the number x for which you would like to generate the square.
INPUT		Close the value entry using the <INPUT> key and the result is transferred into the field.

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.

1.4.8 Pocket calculator

You can use the 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.
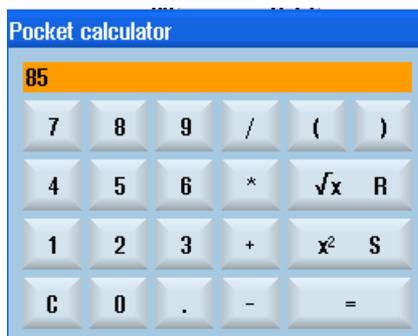
Procedure

	1.	Position the cursor on the desired entry field.
=	2.	Press the **<=>** key.
		The calculator is displayed.
	3.	Input the arithmetic statement.
		You can use arithmetic symbols, numbers, and commas.
=	4.	Press the equals symbol on the calculator.
		- OR -
Calculate		Press the "Calculate" softkey.
		- OR -
INPUT		Press the **<INPUT>** key.
		The new value is calculated and displayed in the entry field of the calculator.
Accept	5.	Press the "Accept" softkey.
		The calculated value is accepted and displayed in the entry 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 38)

1.4.9 Context menu

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

- Cut

  Cut Ctrl+X

- Copy

  Copy Ctrl+C

- Paste

  Paste Ctrl+V

Program editor

Additional functions are available in the editor

- Undo the last change

  Undo Ctrl+Z

- Redo the changes that were undone

  Redo Ctrl+Y

Up to 10 changes can be undone.

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

Setup	1.	Select the "Start-up" operating area.
Change language	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.
OK	4.	Press the "OK" softkey.
		- OR -
INPUT		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 Entering Asian characters

You have the possibility of entering Asian characters.

Note
Call the input editor with <Alt + S>

The input editor can only be called there where it is permissible to enter 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
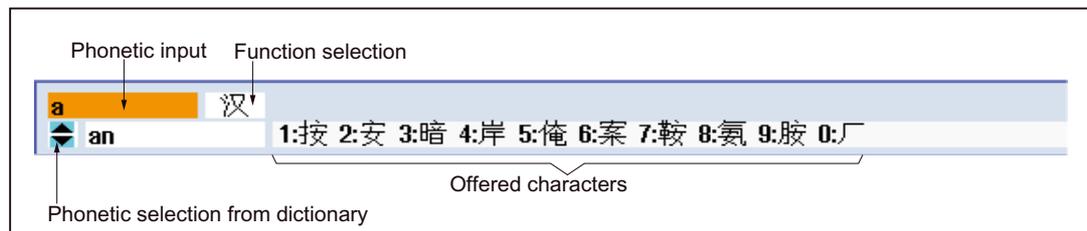
Functions

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

Precondition

The control has been set to Chinese or Korean.

Procedure

Editing characters

A	1. Open the screen form and position the cursor on the entry field and press the <Alt +S> keys.
+	The editor is displayed.
S	
▼	2. Enter the desired phonetic notation.
	3. Click the <Cursor down> key to access the dictionary.
	4. By keeping the <Cursor down> key pressed, displays all the entered phonetic notations and the associated selection characters.
← BACKSPACE	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

○ SELECT	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).
TAB	You can toggle the input cursor between the compound phonetic notations field and the phonetic input field by pressing the <TAB> key.
← BACKSPACE	Combined characters are cancelled via the <BACKSPACE> key.
○ SELECT	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

● Zero offsets

● Setting data

● Program creation / program editing

Note

Configuring access levels for softkeys

You have the option of providing softkeys with protection levels or completely hiding them.

References

For additional information, please refer to the following documentation:

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

Softkeys
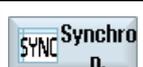
Machine operating area	Protection level
SYNC Synchron.	End user (protection level 3)

Parameters operating area	Protection level
Tool management lists Details	Keyswitch 3 (protection level 4)

Diagnostics operating area	Protection level
Logbook	Keyswitch 3 (protection level 4)
Change	User (protection level 3)
New entry	User (protection level 3)
Startup complete	Manufacturer (protection level 1)
Machine installed	User (protection level 3)
Add HW comp.	Service (protection level 2)

Start-up operating area		Protection levels
System data		End user (protection level 3)
Setup archive		Keyswitch 3 (protection level 4)
General MD	Control unit MD	Keyswitch 3 (protection level 4)
Lic- enses		Keyswitch 3 (protection level 4)
Set MD active (cf)		Keyswitch 3 (protection level 4)
NCK		Service (protection level 2)
Change password		End user (protection level 3)
Delete password		End user (protection level 3)

1.4.14 Online help in SINUMERIK Operate

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

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.
Current topic		
Fullscreen	3.	Press the "Full screen" softkey to use the entire user interface for the display of the online help.
Fullscreen		Press the "Full screen" softkey again to return to the subscreen.
Follow reference	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.
Back to reference	5.	Press the "Back to reference" softkey to jump back to the previous help.

Calling a topic in the table of contents

Table of contents	1.	Press the "Table of contents" softkey.
		Depending on which technology you are using, the Operating Manuals "Operator control Milling", "Operator control Turning" or "Operator control Universal" as well as the "Programming" 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 manual and the section.
⇨ INPUT		
▼	4.	Navigate to the desired topic with the "Cursor down" key.
Follow reference	5.	Press the <Follow reference> softkey or the <INPUT> key to display the help page for the selected topic.
⇨ INPUT		
Current topic	6.	Press the "Current topic" softkey to return to the original help.

Searching for a topic

Search	1.	Press the "Search" softkey. The "Search in Help for:" window is opened.
	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.
✓ **OK**	3.	Enter the desired keyword in the "Text" field and press the "OK" softkey. If you enter the search term on the operator panel, replace an umlaut (accented character) by 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.
Table of contents	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

ⓘ HELP	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.
ⓘ HELP	2.	If you are in the "Start-up" 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

ⓘ HELP	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.
Display all G functions	2.	Press the "Display all G functions" softkey.
Search	3.	With the aid of the search function, select, for example, the desired G code command.
Transfer to editor	4.	Press the "Transfer to editor" softkey.
		The selected G function is taken into the program at the cursor position.
Exit Help	5.	Press the "Exit help" softkey again to close the help.

Setting up the machine

<div align="right">

2

</div>

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 axis with incremental path measuring system must be referenced after the controller has been switched on – however, an absolute path measuring system does not have to be referenced.

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

X	3. Select the axis to be traversed.
Z	

−	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 requirement for the availability of other Safety Integrated functions.

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

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

Option

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

Procedure

M **Machine**	1. Select the "Machine" operating area.
REF.POINT	2. Press the **<REF POINT>** key.

X	3.	Select the axis to be traversed.
Z		
-	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 ◕.
User enable	5.	Press the "User enable" softkey.
		The "User Agreement" window opens.
		It shows a list of all machine axes with their current position and SI position.
	5.	Position the cursor in the "Acknowledgement" field for the axis in question.
SELECT	6.	Activate the acknowledgement with the <SELECT> key.
		The selected axis is marked with an "x" meaning "safely referenced" in the "Acknowledgement" column.
SELECT		By pressing the <SELECT> key again, 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:

● Approach reference point, i.e. the machine axis is referenced

● 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" operating mode

The "REF POINT" operating mode 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" operating mode

The "REPOS" operating mode 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.

"MDI" mode (Manual Data Input)

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

Selecting "MDI"

Press the <MDI> key.

"AUTO" mode

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

Select "AUTO"

Press the <AUTO> key.

"TEACH IN" operating mode

"TEACH IN" is available in the "AUTO" and "MDI" operating 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 Modes groups and channels

Every channel behaves like an independent NC. A maximum of one part program can be processed per channel.

- Control with 1channel

  One mode group exists.

- Control with several channels

  Channels can be grouped to form several "mode groups."

Example

Control with 4 channels, where machining is carried out in 2 channels and 2 other channels are used to control the transport of the new workpieces.

Mode group 1 channel 1 (machining)

Channel 2 (transport)

Mode group 2 channel 3 (machining)

Channel 4 (transport)

Mode groups (MGs)

Technologically-related channels can be combined to form a mode group.

Axes and spindles of the same mode group can be controlled by one or more channels.

An operating mode group is in one of "Automatic", "JOG" or "MDI" operating modes, i.e., several channels of an operating mode group can never assume different operating modes.

2.3.3 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.

When a channel menu is available, all of the channels are displayed on softkeys and can be switched over.

Changing the channel

<table>
<tr>
<td>

```
 1...n
 ↓  ↓
CHANNEL
```

</td>
<td>

Press the <CHANNEL> key.

The channel changes over to the next channel.

- OR -

If the channel menu is available, a softkey bar is displayed. The active channel is highlighted.

Another channel can be selected by pressing one of the other softkeys.

</td>
</tr>
</table>

References

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

Channel switchover via touch operation

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

See also

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 into account any zero offsets, tool offsets and coordinate rotation.

Procedure

![M Machine]	1.	Select the "Machine" operating area.
![JOG] ![AUTO]	2.	Press the <JOG> or <AUTO> key.
Act. vls. MCS	3.	Press the "Act.vls. MCS" softkey.
Act. vls. MCS		The machine coordinate system is selected. The title of the actual value window changes in the MCS.

Machine manufacturer

The softkey to changeover the coordinate system can be hidden. Please refer to the machine manufacturer's specifications.

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

Machine / JOG / AUTO	1.	Select <JOG> or <AUTO> mode in the "Machine" operating area.
> / Settings	2.	Press the menu forward key and the "Settings" softkey. A new vertical softkey bar appears.
Changeover inch	3.	Press the "Switch to inch" softkey. A prompt asks you whether you really want to switch over the unit of measurement.
OK	4.	Press the "OK" softkey. The softkey label changes to "Switch to metric". The unit of measurement applies to the entire machine.
Changeover metric	5.	Press the "Switch to metric" softkey to set the unit of measurement of the machine to metric again.

2.4.3 Setting the zero offset

You can enter a new position value in the actual value display for individual axes when a settable zero 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 zero offset (e.g. G54).

Relative actual value

Further, you also have the possibility of entering position values in the relative coordinate system.

Note

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

Resetting the relative actual value

Delete REL   Press the "Delete REL" softkey.

The actual values are deleted.

The softkeys to set the zero point in the relative coordinate system are only available if the corresponding machine data is set.

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

Precondition

The controller is in the workpiece coordinate system.

The actual value is set in the reset state.

Note

Setting the ZO 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

M **Machine** **JOG**	1.	Select the "JOG" mode in the "Machine" operating area.
Set WO	2.	Press the "Set ZO" softkey.
▶▶ **REL act. vals** **Set REL**		- OR - Press the ">>", "REL act. vals" and "Set REL" softkeys to set position values in the relative coordinate system.
INPUT	3.	Enter the new required 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 -
X=0 ... **Z=0**		Press softkeys "X=0","Y=0" or "Z=0" to set the relevant position to zero.
X=Y=Z=0		- OR - Press softkey "X=Y=Z=0" to set all axis positions to zero simultaneously.
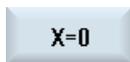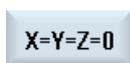
Resetting the actual value

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

Note

Irreversible active zero offset

The current active zero offset is irreversibly deleted by this action.

Setting up the machine

2.5 Zero offsets

2.5 Zero 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 (Machine). The program for machining the workpiece, however, is based on the workpiece zero (W) of the workpiece coordinate system (Work). 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 zero 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 (Machine).

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.
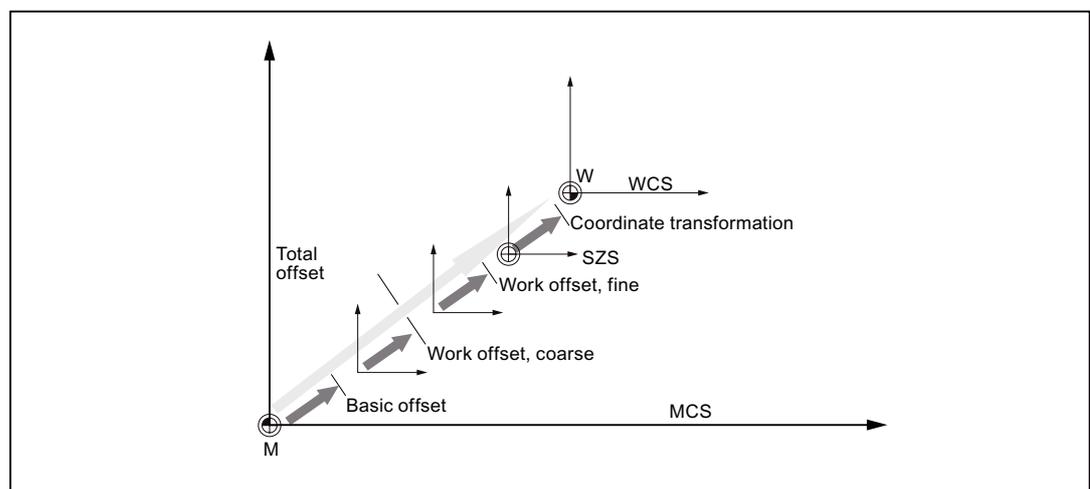
Figure 2-1   Zero offsets

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

Base offset

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

Coarse and fine offsets

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

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

Note

Deselect fine offset (only 840D sl)

You have the option of deselecting the fine offset using machine data MD18600 $MN_MM_FRAME_FINE_TRANS

2.5.1 Display active zero offset

The following zero offsets are displayed in the "Zero Offset - Active" window:

- Zero offsets, for which active offsets are included, or for which values are entered.
- Settable zero offsets
- Total zero offset

This window is generally used only for monitoring.

The availability of the offsets depends on the setting.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure
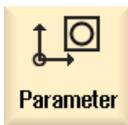
	1.	Select the "Parameter" operating area.
	2.	Press the "Zero offset" softkey.
		The "Zero Offset - Active" window is opened.

Note

Further details on zero offsets

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

2.5.2 Displaying the zero offset "overview"

The active offsets or system offsets are displayed for all set-up axes in the "Zero Offset - Overview" window.

In addition to the offset (course and fine), the rotation, scaling and mirroring defined using this are also displayed.

This window is generally used only for monitoring.

Display of active zero offsets

Zero offsets	
DRF	Displays the handwheel axis offset.
Basic reference	Displays the additional zero offsets programmed with $P_SETFRAME.
	Access to the system offsets is protected via a keyswitch.
External ZO frame	Displays the additional zero offsets programmed with $P_EXTFRAME.
Total base ZO	Displays all effective basis offsets.
G500	Displays the zero offsets activated with G54 - G599.
	Under certain circumstances, you can change the data using "Set ZO", i.e. you can correct a zero point that has been set.
Tool reference	Displays the additional zero offsets programmed with $P_TOOLFRAME.
Workpiece reference	Displays the additional zero offsets programmed with $P_WPFRAME.
Programmed ZO	Displays the additional zero offsets programmed with $P_PFRAME.
Cycle reference	Displays the additional zero offsets programmed with $P_CYCFRAME.
Total ZO	Displays the active zero offset, resulting from the total of all zero offsets.

Procedure

	1.	Select the "Parameter" operating area.

	2	Press the "Zero offset" and "Overview" softkeys. The "Zero Offsets - Overview" window opens.

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 "Zero offset - Base" window.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure
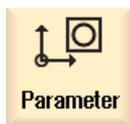
	1.	Select the "Parameter" operating area.

	2.	Press the "Zero offset" softkey.

	3.	Press the "Base" softkey. The "Zero Offset - Base" window is opened.

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

Note

Activate base offsets

The offsets specified here are immediately active.

2.5.4   Displaying and editing settable zero offset

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

Rotation, scaling and mirroring are displayed.

Procedure

[Parameter icon]	1.	Select the "Parameter" operating area.
[Work offset icon]	2.	Press the "Zero offset" softkey.
[G54...G599 icon]	3.	Press the "G54…G599" softkey.
		The "Zero Offset - G54..G599" window is opened.

Note

The labeling of the softkeys for the settable zero offsets varies, i.e. the settable zero offsets configured on the machine are displayed (examples: G54…G57, G54…G505, G54…G599).

Please observe the machine manufacturer's specifications.

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

Note

Activate settable zero offsets

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

2.5.5 Displaying and editing details of the zero offsets

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

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

- Coarse and fine offsets
- Rotation
- Scaling
- Mirroring

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

Parameter	1.	Select the "Parameter" operating area.
Work offset	2.	Press the "Zero offset" softkey.
Active **G54...G599**	3.	Press the "Active", "Base" or "G54…G599" softkey. The corresponding window appears.
	4.	Place the cursor on the desired zero offset to view its details.
Details	5.	Press the "Details" softkey.
		A window opens, depending on the selected zero offset, e.g. "Zero Offset - Details: G54 to G599".
	6.	You can edit the values directly in the table. - OR -
Clear Offset		Press the "Clear offset" softkey to reset all entered values.

WO +	Press the "ZO +" or "ZO -" 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.
...	
WO -	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.

« Back	Press the "Back" softkey to close the window.

2.5.6 Deleting a zero offset

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

Procedure

Parameter	1.	Select the "Parameter" operating area.
Work offset	2.	Press the "Zero offset" softkey.
Active	3.	Press the "Active", "Base" or "G54…G599" softkey.
G54...G599		
Details	4.	Press the "Details" softkey.
	5.	Position the cursor on the zero offset you would like to delete.
Clear Offset	6.	Press the "Clear offset" softkey.

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.

Requirements

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.

Procedure
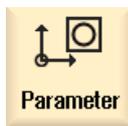
Parameter	1.	Select the "Parameter" operating area.
SD Setting data	2.	Press the "Setting data" softkey.
Working area limit.		The "Working Area Limitation" window appears.
	3.	Place the cursor in the required field and enter the new values via the numeric keyboard. The upper or lower limit of the protection zone changes according to your inputs.
	4.	Click the "active" checkbox to activate the protection zone.
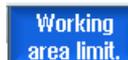
Note

You will find all of the setting data in the "Start-up" operating area under "Machine data" via the menu forward key.

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
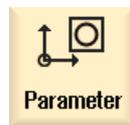
	1.	Select the "Parameter" operating area.
	2.	Press the "Setting data" and "Spindle data" softkeys. The "Spindles" window opens.
	3.	If you want to change the spindle speed, place the cursor on the "Maximum", "Minimum", or "Spindle speed limitation at G96" and enter a new value.

2.7 Displaying setting data lists

You can display lists with configured setting data.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure
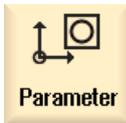
Parameter	1. Select the "Parameter" operating area.
SD Setting data **Data lists**	2. Press the "Setting data" and "Data lists" softkeys. The "Setting Data Lists" window opens.
Select data list	3. Press the "Select data list" softkey and in the "View" list, select the required list with setting data.

2.8 Handwheel assignment

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

Software option

You require the "Extended operator functions" option for the handwheel offset (only for 828D).

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

- Geometry axes

  When traversing, the geometry axes taken into account the actual machine status (e.g. rotations, transformations). All channel machine axes, which are currently assigned to the geometry axis, are in this case simultaneously traversed.

- Channel machine axes

  Channel machine axes are assigned to the particular channel. They can only be individually traversed, i.e. the actual machine state has no influence.

  The also applies to channel machine axes, that are declared as geometry axes.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

M Machine	1.	Select the "Machine" operating area.
JOG / MDA		Press the <JOG>, <AUTO> or <MDI> key.
> / Hand-wheel	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.	Position the cursor in the field next to the handwheel with which you wish to assign the axis (e.g. No. 1).
X	4.	Press the corresponding softkey to select the desired axis (e.g. "X").
INSERT / INPUT		- OR 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).
Hand-wheel	5.	Press the "Handwheel" softkey again.
<< Back		- OR - Press the "Back" softkey. The "Handwheel" window closes.

Deactivate handwheel

1. Position the cursor on the handwheel whose assignment you wish to cancel (e.g. No. 1).

2. Press the softkey for the assigned axis again (e.g. "X").

- OR -

Open the "Axis" selection box using the <INSERT> key, navigate to the empty field, and press the <INPUT> key.

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.

Software option

You require the "Extended operator functions" option to load and save MDA programs (for 828D).

2.9.1 Loading an MDA program from the Program Manager

Procedure

Machine	1.	Select the "Machine" operating area.
MDA	2.	Press the <MDI> key.
		The MDI editor opens.
Load MDI	3.	Press the "Load MDI" softkey.
		A changeover is made into the Program Manager.
		The "Load in MDI" window opens. It shows you a view of the program manager.
Search	4.	Position the cursor to the corresponding storage location, press the "Search" softkey and enter the required search term in the search dialog if you wish to search for a specific file.
		Note: The place holders "*" (replaces any character string) and "?" (replaces any character) make it easier for you to perform a search.
	5.	Select the program that you would like to edit or execute in the MDI window.
OK	6.	Press the "OK" softkey.
		The window closes and the program is ready for operation.

2.9.2 Saving an MDA program

Procedure

	1.	Select the "Machine" operating area.
	2.	Press the <MDI> key.
		The MDI editor opens.
	3.	Create the MDI program by entering the G-code commands using the operator's keyboard.
	4.	Press the "Store MDI" softkey.
		The "Save from MDI: 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 MDI program you created, and place the cursor on the directory in which the program is to be stored.
		- OR -
		Position the cursor to the required storage location, press the "Search" softkey and enter the required search term in the search dialog if you wish to search for a specific directory or subdirectory.
		Note: The place holders "*" (replaces any character string) and "?" (replaces any character) make it easier for you to perform a search.
	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

[Machine icon]	1.	Select the "Machine" operating area.
[MDA icon]	2.	Press the <MDA> key. The MDA editor opens.
	3.	Input the desired G-code commands using the operator's keyboard.
[CYCLE START icon]	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 109)

2.9.4   Deleting an MDA program

Precondition

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

Procedure

[Delete blocks icon]	Press the "Delete blocks" softkey.
	The program blocks displayed in the program window are deleted.

Working in manual mode

<div style="text-align: right; font-size: xx-large;">3</div>

3.1 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.

3.2 Selecting a tool and spindle

3.2.1 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 using the "Select tool" softkey.
D	Cutting edge number of the tool (1 - 9)
Spindle	Spindle selection, identification with spindle number
Spindle M function	▓
	↻
	↺
	↻↺
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.

3.2.2 Selecting a tool

Procedure

Machine	1.	Select the "JOG" operating mode.
JOG		
T,S,M	2.	Press the "T, S, M" softkey.
	3.	Enter the name or the number of the tool T in the input field.
		- OR -
Select tool		Press the "Select tool" softkey to open the tool list, position the cursor on the desired tool and press the "In Manual" softkey.
In manual		The tool is transferred to the "T, S, M... window" and displayed in the field of tool parameter "T".
SELECT	4.	Select tool edge D or enter the number directly in field "D".
CYCLE START	5.	Press the <CYCLE START> key.
		The tool is loaded into the spindle.

3.2.3 Starting and stopping a spindle manually

Procedure

Machine	1.	Select the "JOG" operating mode.
JOG		
T,S,M	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.

3.2.4 Position spindle

Procedure

1. Select the "JOG" operating 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.

3.3 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.

3.3.1 Traverse axes by a defined increment

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

Procedure

⬛ M. Machine	1.	Select the "Machine" operating area.
〰 JOG	2.	Press the <JOG> key.
→⎮ 1 →⎮ 10000	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 microinches. Example: Press the "100" button for a desired increment of 100 µm (= 0.1 mm).
X Z	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 controller 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 refer to the machine manufacturer's specifications.

3.3.2 Traversing axes by a variable increment

Procedure

Machine (M)	1.	Select the "Machine" operating area.
JOG		Press the <JOG> key.
Settings	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).
[VAR]	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.
–		

3.4 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.

3.5 Manual retraction

After an interruption of a tapping operation (G33/G331/G332) or a general drilling operation (tools 200 to 299) due to power loss or a RESET at the machine control panel, you have the possibility to retract the tool in the JOG mode in the tool direction without damaging the tool or the workpiece.

The retraction function is especially useful when the coordinate system is swiveled, i.e. the infeed axis is not in the vertical position.

Note

Tapping

In the case of tapping, the form fit between the tap and the workpiece is taken into account and the spindle moved according to the thread.

The Z axis as well as the spindle can be used for retraction at the thread.

The "Retraction" function must be set up by the machine manufacturer.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

	1.	<RESET> interrupts the power supply to the machine and any running part program.
	2.	After a power supply interruption, switch on the controller.
	3.	Select the JOG operating mode.
	4.	Press the Menu forward key.
	5.	Press the "Retract" softkey. The "Retract Tool" window opens. The softkey is available only when an active tool and retraction data are present.
	6.	Select the "WCS" coordinate system on the machine control panel.

	7.	Select the required axis in the "Retraction axis" selection box.

SELECT

Retract tool

Retraction axis   Z

+Z
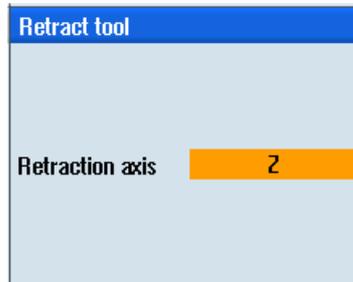
8.   Use the traversing keys (e.g. Z +) to traverse the tool from the workpiece according to the retraction axis selected in the "Retract Tool" window.

Retract

9.   To exit the window, press the "Retract" softkey again when the tool is at the desired position.

3.6 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.

Machining the workpiece

<div style="text-align: right; font-size: 2em;">4</div>

4.1 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 zero offsets have been entered.
- The necessary safety interlocks implemented by the machine manufacturer are activated.

General sequence

Program manager	1.	Use the Program manager to select the desired program.
NC NC **USB** USB		Select the desired program under "NC", "Local drive", "USB" or set-up network drives.
Select	3.	Press the "Select" softkey. The program is selected for execution and automatically switched to the "Machine" operating area.
CYCLE START	4.	Press the <CYCLE START> key. The program is started and executed.

Note

Starting the program in any operating area

If the controller 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

4.2 Selecting a program

Procedure

| | 1. | Select the "Program manager" operating area. |
| | | The directory overview is opened. |

Program manager

| | 2. | Place the cursor on the directory containing the program that you want to select. |
| | 3. | Press the <INPUT> key |

INPUT

- OR -

Press the <Cursor right> key.

The directory contents are displayed.

| | 4. | Place the cursor on the desired program. |
| | 5. | Press the "Select" softkey. |

Select

The program is selected.

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

4.3   Program running-in

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.

Note

Settings for the automatic mode

Rapid traverse reduction and dry run feed rate are available to run-in or to test a program.

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)

Precondition

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

Procedure

NC Prog. cntrl.	1.	Press the "Prog. ctrl." softkey and select the desired variant in the "SBL" field.
SINGLE BLOCK	2.	Press the <SINGLE BLOCK> key.
CYCLE START	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.
CYCLE START	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 93)

4.4 Displaying the current program block

4.4.1 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 G code blocks in the NC memory, not for external execution.
	3.	Press the <INSERT> key to exit the program and the edit mode again.

4.4.2 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.
Basic blocks	2.	Press the "Basic blocks" softkey. The "Basic Blocks" window opens.
SINGLE BLOCK	3.	Press the <SINGLE BLOCK> key if you wish to execute the program block-by-block.
CYCLE START	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.
Basic blocks	5.	Press the "Basic blocks" softkey once again to hide the window again.

4.4.3 Display program level

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

Several program run throughs

If you have programmed several program run throughs, i.e. subprograms are run through several times one after the other by specifying the additional parameter P, then during processing, the program runs still to be executed are displayed in the "Program Levels" window.

Program example

N10 subprogram P25

If, in at least one program level, a program is run through several times, a horizontal scroll bar is displayed that allows the run through counter P to be viewed in the righthand window section. The scroll bar disappears if multiple run-through is no longer applicable.

Display of program level

The following information will be displayed:

- Level number
- Program name
- Block number, or line number
- Remain program run throughs (only for several program run throughs)

Precondition

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

Procedure

Program levels	Press the "Program levels" softkey. The "Program levels" window appears.

4.5 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 also available for execute from external; however, when making program changes, the NC channel must be brought into the reset state.

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.
Prog. corr.	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.
NC Execute	4.	Press the "NC Execute" softkey. The system switches back to the "Machine" operating area and selects "AUTO" mode.
CYCLE START	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.

4.6   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

NOTICE
Risk of collision
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.

Precondition

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.

Procedure

REPOS	1.	Press the <REPOS> key.
X Z	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.

4.7 Starting execution at a specific point

4.7.1 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

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

Software option

You require the "Extended operator functions" option for the "Search pointer" function (only for 828D).

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

Preconditions

1. You have selected the desired program.

2. The controller is in the reset state.

3. The desired search mode is selected.

NOTICE
Risk of collision
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 -

Press the "Back" softkey.

You have now exited the block search function.

See also

4.7.2 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

4.7.3 Simple search target definition

Requirement

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

Procedure

NC Block search	1.	Press the "Block search" softkey.
	2.	Place the cursor on a particular program block.
		- OR -
Search for text		Press the "Find text" softkey, select the search direction, enter the search text and confirm with "OK".
✓ OK		
Start search	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.
Start search	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.

4.7.4 Defining an interruption point as search target

Requirement

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

Software option

You require the "Extended operator functions" option (only for 828D).

Procedure

NC Block search	1.	Press the "Block search" softkey.
Interrupt point	2.	Press the "Interrupt point" softkey. The interruption point is loaded.
Higher level / **Lower level**	3.	If the "Higher level" and "Lower level" softkeys are available, use these to change the program level.
Start search	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.
CYCLE START	5.	Press the <CYCLE START> key twice. The execution will continue from the interruption point.

4.7.5   Entering the search target via search pointer

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

Software option

You require the "Extended operator functions" option for the "Search pointer" function (only for 828D).

Requirement

The program is selected and the controller is in the reset state.

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

NC Block search	1.	Press the "Block search" softkey.
Search pointer	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.
Start search	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.

4.7.6 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

4.7.7 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. **Note**: If machine data 11450.1=1 is set, the rotary axes of the active swivel data record are pre-positioned after the block search.
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.
With calculation - skip extcall	This is used to speed-up a search with calculation when using EXTCALL programs: EXTCALL programs are not taken into account. **Notice**: Important information, e.g. modal functions, which are located in the EXTCALL program, are not taken into account. In this case, after the search target has been found, the program is not able to be executed. Information such as this should be programmed in the main program.
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.).
With program test	Multi-channel block search with calculation (SERUPRO). All blocks are calculated during the block search. Absolutely no axis motion is executed, however, all auxiliary functions are output. The NC starts the selected program in the program test mode. If the NC reaches the specified target block in the actual channel, it stops at the beginning of the target block and deselects program test mode again. After continuing the program with NC start (after REPOS motion) the auxiliary functions of the target block are output. For single-channel systems, the coordination is supported with events running in parallel, e.g. synchronized actions. **Note** The search speed depends on MD settings.

Note

Search mode for the ShopMill and ShopTurn programs

The search variant for the ShopMill/ShopTurn machining step programs can be specified via MD 51024. This applies only to the ShopMill/ShopTurn single-channel view.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

References

For additional information, please refer to the following documentation:

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

Procedure

Machine	1.	Select the "Machine" operating area.
AUTO	2.	Press the <AUTO> key.
Block search / **Blk sear. mode**	3.	Press the "Block search" and "Block search mode" softkeys. The "Search Mode" window appears.

4.8 Controlling the program run

4.8.1 Program control

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

Abbreviation/program control	Mode of operation
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.
RG0 Reduced rapid traverse	In the rapid traverse mode, the traversing speed of the axes is reduced to the percentage value entered in RG0. Note: You define the reduced rapid traverse in the settings for automatic operation.
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. Note: The display can be changed. Please refer to the machine manufacturer's specifications.
DRF Handwheel offset	Enables an additional incremental zero offset while processing in automatic mode with an electronic handwheel. This function can be used to compensate for tool wear within a programmed block. Note: You require the "Extended operator functions" option to use the handwheel offset (for 828D).
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.
GCC	When executing a jobshop program, it is converted into a G-code program.

Activating program control

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

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.

Procedure

	1.	Select the "Machine" operating area.
	2.	Press the <AUTO> or <MDI> key.
	3.	Press the "Prog. ctrl." softkey. The "Program Control" window appears.

4.8.2 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 refer to the machine manufacturer's specifications.

Software option

In order to have more than two skip levels, for 828D you require the "Extended operator functions" option.

Skip levels, activate

Select the corresponding checkbox to activate the desired skip level.

Note

The "Program Control - Skip Blocks" window is only available when more than one skip level is set up.

Procedure

	1.	Select the "Machine" operating area.
	2.	Press the <AUTO> or <MDA> key.
	3	Press the "Prog. ctrl." and "Skip blocks" softkeys. The "Program Control" window appears and shows a list of skip levels.
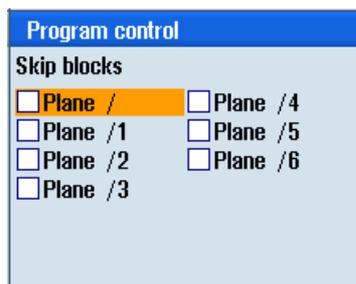
4.9 Overstore

With overstore, you have the option of executing technological parameters (for example, auxiliary functions, axis feed, spindle speed, programmable instructions, etc.) before the program is actually started. The program instructions act as if they are located in a normal part program. These program instructions are, however, only valid for one program run. The part program is not permanently changed. When next started, the program will be executed as originally programmed.

After a block search, the machine can be brought into another state with overstore (e.g. M function, tool, feed, speed, axis positions etc.), in which the normal part program can be successfully continued.

Software option

You require the "Extended operator functions" option for the overstore function (for 828D).

Precondition

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

Procedure

Machine	1.	Open the program in the "AUTO" mode.
AUTO		
Over- store	2.	Press the "Overstore" softkey. The "Overstore" window opens.
	3.	Enter the required data and NC block.
CYCLE START	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.
Back	5.	Press the "Back" softkey. The "Overstore" window closes.
CYCLE START	6.	Press the <CYCLE START> key again. The program selected before overstoring continues to run.

Note

Block-by-block execution

The <SINGLE BLOCK> key is also active in the overstore mode. If several blocks are entered in the overstore buffer, then these are executed block-by-block after each NC start

---

Deleting blocks

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

4.10 Editing a program

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

Note

Maximum block length

The maximum block length is 512 characters.

Calling the editor

- The editor is started via the "Program correction" function in the "Machine" operating area and with the <INSERT> key.

- 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, if 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 121)

Opening and closing a program (Page 230)

Correcting a program (Page 98)

4.10.1 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.

Various search options are available that enable selective searching.

Search options

- Whole words

  Activate this option and enter a search term if you want to search for texts/terms that are present as words in precisely this form.

  If, for example, you enter the search term "Finishing tool", only single "Finishing tool" terms are displayed.

- Exact expression

  Activate this option if you want to search for place holders in program lines (e.g. "*" or "?") that you want to replace.

Note

Search with place holders

When searching for specific program locations, you have the option of using place holders:

- "*": Replaces any character string
- "?": Replaces any character

Requirement

The desired program is opened in the editor.

Procedure

Search ▶	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.

- OR -

Activate the "Exact expression" checkbox if, for example, you want to search for place holders ("*", "?") in program lines.

4. Position 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
Go to start	The cursor is set to the first character in the program.
Go to end	The cursor is set to the last character in the program.

4.10.2 Replacing program text

You can find and replace text in one step.

Requirement

The desired program is opened in the editor.

Procedure

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

2. Press the "Find and 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. Position 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 "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.

Note

Replacing texts

- Read-only lines (;*RO*)

  If hits are found, the texts are not replaced.

- Contour lines (;*GP*)

  If hits are found, the texts are replaced as long as the lines are not read-only.

- Hidden lines (;*HD*)

  If hidden lines are displayed in the editor and hits are found, the texts are replaced as long as the lines are not read-only. Hidden lines that are not displayed, are not replaced.

See also

Editor settings (Page 121)

4.10.3 Copying/pasting/deleting program blocks

Requirement

The program is opened in the editor.

Procedure

Mark	1.	Press the "Mark" softkey.
		- OR -
SELECT		Press the <SELECT> key.
	2.	Select the desired program blocks with the cursor or mouse.
Copy	3.	Press the "Copy" softkey in order to copy the selection to the buffer memory.
Paste	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.
		- OR -
Cut		Press the "Cut" softkey to delete the selected program blocks.
		Note: When editing a program, you cannot copy or cut more than 1024 lines. While a program that is not on the NC is opened (progress display less than 100%), you cannot copy or cut more than 10 lines or insert more than 1024 characters.

Note

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

Note

Copy/cut current line

To copy and cut the current line where the cursor is positioned, it is not necessary to mark or select it. You have the option of making the "Cut" softkey only operable for marked program sections via editor settings.

See also

Opening additional programs (Page 120)

4.10.4 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.
Renumbering	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.
✓ OK	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.

4.10.5 Creating a program block

In order to structure programs in order to achieve a higher degree of transparency, you have the option of combining several G-code blocks to form program blocks.

You then have the option of opening and closing these blocks depending on your requirement.

Settings for a program block

Display	Meaning
Text	Block designation
Spindle	• S1 • S2 Spindle assignment. Defines at which spindle a program block is to be executed.
Addit. run-in code	• Yes For the case that the block is not executed, as the specified spindle should not be processed, then it is possible to temporarily activate a so-called "Addit. run-in code". • No
Automat. Retraction	• Yes Block start and block end are moved to the tool change point, i.e. the tool is brought into a safe range. • No

Procedure

Program manager	1.	Select the "Program Manager" operating area.
NC / Local drive	2.	Select the storage location and create a program or open a program. The program editor is opened.
	3.	Select the required program blocks that you wish to combine to form a block.
Build block	4.	Press the "Build group" softkey. The "Build group" window is opened.

5. Enter a designation for the block, assign the spindle, if required, select the Addit. run-in code and the automatic retraction and then press the "OK" softkey.

Opening and closing blocks

6. Press the ">>" and "View" softkeys.

7. Press the "Open blocks" softkey if you wish to display the program with all blocks.

8. Press the "Close blocks" softkey, if you wish to display the program again in a structured form.

4.10.6 Opening additional programs

You have the option of viewing and editing two programs simultaneously in the editor.

For instance, you can copy program blocks or machining steps of a program and paste them into another program.

Opening several programs

You have the option of opening up to ten program blocks.

1. In the program manager, select the programs that you wish to open and view in the multiple editor and then press on the "Open" softkey.

   The editor is opened and the first two programs are displayed.

2. Press the <NEXT WINDOW> key to change to the next opened program.

3. Press the "Close" softkey to close the actual program.

Note

Pasting program blocks

JobShop machining steps cannot be copied into a G code program.

Requirement

You have opened a program in the editor.

Procedure

▶▶	1.	Press the ">>" and "Open additional program" softkeys.
Open 2nd program		The "Select Additional Program" window is opened.
	2.	Select the program or programs that you wish to display in addition to the already opened program.
✓ **OK**	3.	Press the "OK" softkey.
		The editor opens and displays both programs next to each another.

See also

Copying/pasting/deleting program blocks (Page 117)

4.10.7 Editor settings

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

Defaults

Setting	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 displayed under "Number automatically".
Increment	Defines the increment used for the block numbers. The field is only editable when "Yes" is displayed under "Number automatically".
Display hidden lines	• Yes: Hidden lines marked with "*HD" (hidden) will be displayed. • No: Lines marked with ";*HD*" will not be displayed. **Note**: Only visible program lines are taken into account with the "Search" and "Search and Replace" functions.
Display block end a symbol	The "CFLF" (line feed) symbol ¶ is displayed at the block end.

Setting	Meaning
Scroll horizontally	Yes: 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" and "No" softkeys.
Visible programs	• 1 - 10 Select how many programs can be displayed next to one another in the editor. • Auto Specifies that the number of programs entered in a job list or up to 10 selected programs will be displayed next to each other.
Width of the program with focus	Here, you enter the width of the program that has the input focus in the editor as a percentage of the window width.
Cut only after selecting	• Yes: The cutting of program sections is possible only when program lines have been selected, i.e. the "Cut" softkey only then is active. • No: The "Cut" softkey is available by default.

Note

All entries that you make here are effective immediately.

Procedure

Program	1.	Select the "Program" operating area.
Edit		You have activated the editor.
▶▶ / Settings	2.	Press the ">>" and "Settings" softkeys. The "Settings" window opens.
OK	3.	Make the desired changes here and press the "OK" softkey to confirm your settings.

See also

Replacing program text (Page 115)

4.11 Display and edit user variables

4.11.1 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.

4.11.2 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 controller 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 refer to the machine manufacturer's specifications.

Procedure
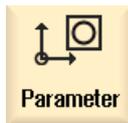
Parameter	1.	Select the "Parameter" operating area.
R User variable	2.	Press the "User variable" softkey.
R variables	3.	Press the "R variables" softkey. The "R Parameters" window appears.

Delete R variables

▶▶ Delete ▶	1.	Press the ">>" and "Delete" softkeys. The "Delete R parameters" window appears.
✓ OK	2.	Enter the R parameter(s) whose channel-specific values you would like to delete and press the "OK" softkey. A value of 0 is assigned to the selected R parameters or to all R parameters.

4.11.3   Displaying global user data (GUD)

Global user variables

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	Meaning
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

Procedure
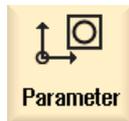
	1.	Select the "Parameter" operating area.
	2.	Press the "User variable" softkey.
	3.	Press the "Global GUD" softkeys.

The "Global User Variables" 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 wish to display SGUD, MGUD, UGUD as well as GUD4 to GUD 6 of the global user variables.

- 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 variables.

Note

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

4.11.4   Displaying channel GUDs

Channel-specific user variables

Like the GUDs, channel-specific user variables 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

Procedure

Parameter	1.	Select the "Parameter" operating area.
R User variable	2.	Press the "User variable" softkey.
Channel GUD **GUD selection**	3.	Press the "Channel GUD" and "GUD selection" softkeys.
		A new vertical softkey bar appears.
SGUD **GUD6**	4.	Press the "SGUD" ... "GUD6" softkeys if you want to display the SGUD, MGUD, UGUD as well as GUD4 to GUD 6 of the channel-specific user variables.
		- OR -
Continue **GUD9**		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 variables.

4.11.5 Displaying local user data (LUD)

Local user variables

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

The controller 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)

Procedure

	1.	Select the "Parameter" operating area.
	2.	Press the "User variable" softkey.
	3.	Press the "Local LUD" softkey.

4.11.6 Displaying program user data (PUD)

Program-global user variables

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

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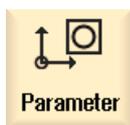
Procedure

	1.	Select the "Parameter" operating area.
	2.	Press the "User variable" softkey.
	3.	Press the "Program PUD" softkey.

4.11.7 Searching for user variables

You can search for R parameters and user variables.

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 variables.
	3.	Press the "Search" softkey. The "Search for R Parameters" or "Search for User Variables" window opens.
	4.	Enter the desired search term and press "OK". The cursor is automatically positioned on the R parameter or user variables you are searching for, if they exist.

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

Procedure

	1.	Select the "Start-up" 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 <Cursor right> key.

The selected file is opened in the editor and can be edited there.

6. Define the desired user variable.

7. Press the "Exit" softkey to close the editor.

Activating user variables

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.

4.12 Display G and auxiliary functions

4.12.1 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

[Machine]	1.	Select the "Machine" operating area.
[JOG] ... [AUTO]	2.	Press the <JOG>, <MDA> or <AUTO> key.
[G functions]	3.	Press the "G functions" softkey. The "G Functions" window is opened.
[G functions]	4.	Press the "G functions" softkey again to hide the window.

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:

SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual

4.12.2 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 surface 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

[Machine icon]	1.	Select the "Machine" operating area.
[JOG icon]	2.	Press the <JOG>, <MDA> or <AUTO> key.
...		
[AUTO icon]		
[>> and All G functions softkeys]	3.	Press the ">>" and "All G functions" softkeys. The "G Functions" window is opened.

4.12.3 G functions for mold making

In the window "G functions", important information for machining free-form surfaces can be displayed using the "High Speed Settings" function (CYCLE832).

Software option

You require the "Advanced Surface" software option in order to use this function.

High-speed cutting information

In addition to the information that is provided in the "All G functions" window, the following programmed values of the following specific information is also displayed:

- CTOL
- OTOL
- STOLF

The tolerances for G0 are only displayed if they are active.

Particularly important G groups are highlighted.

You have the option to configure which G functions are highlighted.

References

- Additional information is available in the following references:

  Function Manual, Basic Functions; Chapter, "Contour/orientation tolerance"

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

  Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

Procedure

	1.	Select the "Machine" operating area
	2.	Press the <JOG>, <MDI> or <AUTO> key.
	3.	Press the ">>" and "All G functions" softkeys. The "G Functions" window is opened.

4.12.4 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>, <MDA> or <AUTO> key.
	3.	Press the "H functions" softkey. The "Auxiliary Functions" window opens.
	4.	Press the "H functions" softkey again to hide the window again.

4.13 Displaying status of synchronized actions

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

You get a list with all currently active synchronized actions.

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

References

Programming Guide Job Planning (PGA) Chapter: Motion-synchronous actions

Status of synchronized actions

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

● Waiting

● Active

● Blocked

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

Synchronization types

Synchronization types	Meaning
ID=n	Modal synchronized actions in the automatic mode up to the end of program, local to program; n = 1... 254
IDS=n	Static synchronized actions, modally effective in every operating type, also beyond the end of program; n = 1... 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.

Display of synchronized actions

Using softkeys, you have the option of restricting the display to activated synchronized actions.

Procedure

[Machine]	1.	Select the "Machine" operating area.
[AUTO] [JOG]	2.	Press the <AUTO>, <MDA> or <JOG> key.
[>] [SYNC Synchron.]	3.	Press the menu forward key and the "Synchron." softkey. The "Synchronized Actions" window appears. You obtain a display of all activated synchronized actions.
[ID]	4.	Press the "ID" softkey if you wish to hide the modal synchronized actions in the automatic mode.
[IDS]		- AND / OR - Press the "IDS" softkey if you wish to hide static synchronized actions.
[Blockwise]		- AND / OR - Press the "Blockwise" softkey if you wish to hide the non-modal synchronized actions in the automatic mode.
[ID] ... [Blockwise]	5.	Press the "ID", "IDS" or "Blockwise" softkeys to re-display the corresponding synchronized actions.

4.14 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 refer to the machine manufacturer's specifications.

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

  track how much of the current program has been completed as a percentage by using the 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.

4.15   Setting for automatic mode

Before machining a workpiece, you can test the program in order to identify programming errors early on. Use the dry run feedrate for this purpose.

In addition, you have the option of additionally limiting the traversing speed for rapid traverse so that when running-in a new program with rapid traverse, no undesirable high traversing speeds occur.

Dry run feedrate

The feedrate defined here replaces the programmed feedrate during execution if you have selected "DRY run feedrate" under program control.

Reduced rapid traverse

This value entered here reduces the rapid traverse to the entered percentage value if you selected "RG0 reduced rapid traverse" under program control.

Displaying measurement results

Using an MMC command, you can display measurement results in a part program:

You can define the following settings:

- When it reaches the command, the control automatically jumps to the "Machine" operating area and the window with the measurement results is displayed

- The window with the measurement results is opened by pressing the "Measurement result" softkey.

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.
	5.	Enter the desired percentage in the "Reduced rapid traverse RG0" field. RG0 has not effect if you do not change the specified amount of 100%.
	6.	Enter "Automatic" in the "Display measurement result" box if the measurement result window should be automatically opened, or "Manual", if the measurement result window should be opened by pressing the "Measurement result" softkey.

References

Programming Manual Measuring Cycles / 840D sl/828D

Note

The feedrate can be changed while the operation is running.

4.16 Mold making view

4.16.1 Mold making view

For large mold making programs, as provided by CAD systems, you have the option, using a fast view, to display the machining paths. This allows you to obtain a fast overview of the program and possibly correct it.

Checking the program

For instance, you can check whether

- The programmed workpiece has the correct form (shape)
- Whether there are large traversing errors
- If yes, which block must be corrected
- How the approach and retraction are performed

Simultaneous view of the program and mold making view

In the editor, next to the program block display, switch-in the graphic view.

At the left in the editor, if you set the cursor to an NC block with position data, then this NC block is selected in the graphic view.

If you select a point to the right in the graphic view - then the NC block is selected in the left-hand part of the editor - i.e. in the other direction. This is how you jump directly to a position in the program in order to edit a program block for example.
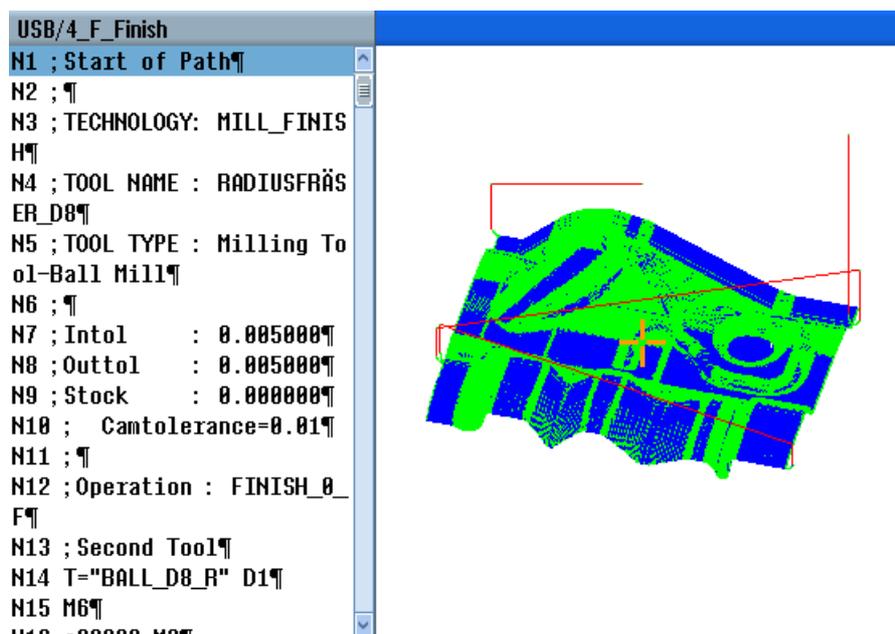
NC blocks that can be interpreted

Following NC blocks are supported for the mold building view.

- Types

  - Lines

    G0, G1 with X Y Z

  - Circles

    G2, G3 with center point I, J, K or radius CR, depending on the working plane G17, G18, G19, CIP with circular point I1, J1, K1 or radius CR

  - Polynomial

    POLY with X, Y, Z or PO[X] PO[Y] PO[Z]

  - B-splines

    BSPLINE with degree SD (SD < 6) node PL weights PW

  - Absolute data AC and incremental data IC are possible

  - For G2, G3 and different radii at the start and end, an Archimedes spiral is used

- Orientation

  - Rotary axis programming with ORIAXES or ORIVECT using ABC for G0, G1, G2, G3, CIP, POLY

  - Rotary axis programming with ORIAXES or ORIVECT using PO[A] POS[b] PO[C] for POLY

  - Orientation vector programming with ORIVECT using A3, B3, C3 for G0, G1, G2, G3, CIP

  - Orientation curve with ORICURVE using XH, YH, ZH, for G0, G1, G2, G3, CIP, POLY, BSPLINE

  - Orientation curve with ORICURVE using PO[XH] PO[YH] PO[ZH] for POLY

  - Rotary axes can be specified using DC

- G codes

  - Working planes (for circle definition G2, G3): G17 G18 G19

  - Incremental or absolute data: G90 G91

Following NC blocks are **not** supported for the mold making view.

- Helix programming

- Rational polynomials

- Other G codes or language commands

All blocks that cannot be interpreted are simply ignored

Changing and adapting the mold making view

Just the same as for simulation and simultaneous recording, you have the option of changing and adapting the simulation graphical representation in order to achieve the optimum view.

- Increasing or reducing the size of the graphic
- Moving the graphic
- Rotating the graphic
- Changing the section

4.16.2 Starting the mold making view

Procedure

Program manager	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 display in the mold making view.
Open	3.	Press the "Open" softkey. The program is opened in the editor.
▶▶ **Mold mak view**	4.	Press the ">>" and "Mold making view" softkeys. The editor splits up into two areas. The G code blocks are displayed in the left half of the editor. The workpiece graphic is displayed in the right half of the editor. All of the points and paths programmed in the part program are shown in the graphical representation.
Graphic	5.	Press the "Graphic" softkey in order to hide the graphic and to view the program as usual in the editor
		- OR -
NC blocks		Press the "NC blocks" softkey in order to hide the G code blocks and therefore only to display the graphic.

4.16.3 Specifically jump to the program block

If you notice anything peculiar in the graphic or identify an error, then from this location, you can directly jump to the program block involved to possibly edit the program.

Preconditions

- The required program is opened in the mold making view.
- The "Graphic" softkey is active.

Procedure

⏩ **Select point**	1.	Press the ">>" and "Select point" softkeys. Cross-hairs for selecting a point is shown in the diagram.
▶ ◀ ▼ ▲	2.	Using the cursor keys, move the cross-hairs to the appropriate position in the graphic.
Select NC block	3.	Press the "Select NC block" softkey. The cursor jumps to the associated program block in the editor.

4.16.4 Searching for program blocks

Using the "Search" function, you can search for specific program blocks as well as edit programs; you can do this in one step by replacing the text that you are searching for by a new text.

Precondition

- The required program is opened in the mold making view.
- The "NC blocks" softkey is active.

Procedure

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

See also

Searching in programs (Page 114)

Replacing program text (Page 115)

4.16.5 Changing the view

4.16.5.1 Enlarging or reducing the graphical representation

Precondition

- The mold making view has been started.
- The "Graphic" softkey is active.

Procedure

 + ... -	1.	Press the <+> and <-> keys if you wish to enlarge or reduce the graphic display. The graphic display enlarged or reduced from the center.
Details Zoom +		- OR - Press the "Details" and "Zoom +" softkeys if you wish to increase the size of the segment.
Details Zoom -		- OR - Press the "Details" and "Zoom -" softkeys if you wish to decrease the size of the segment.
Details Autozoom		- OR - Press the "Details" and "Auto zoom" softkeys if you wish 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

Selected section

The selected sections and size changes are kept as long as the program is selected.

4.16.5.2   Modifying the viewport

Use the magnifying glass if you would like to move, increase or reduce the size of the section of the mold making view, e.g. to view details or display the complete workpiece.

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

Precondition

- The mold making view has been started.
- The "Graphic" softkey is active.

Procedure

Details	1.	Press the "Details" softkey.
Zoom	2.	Press the "Zoom" softkey. A magnifying glass in the shape of a rectangular frame appears.
Zoom +	3.	Press the "Magnify +" or <+> softkey to enlarge the frame.
		- OR - Press the "Magnify -" or <-> softkey to reduce the frame.
Zoom -		
▲ ◄ ► ▼		- OR - Press one of the cursor keys to move the frame up, down, left or right.
✓ Accept	4.	Press the "Accept" softkey to accept the section.

Simulating machining

5

5.1 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.

Graphic display

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

For simulation at milling machines, the workpiece is located, fixed in space. Only the tool moves, independent of the machine type.

Definition of blank

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

The blank is clamped with reference to the coordinate system, which is valid at the time that the blank was defined. This means that before defining the blank in G code programs, the required output conditions must be established, e.g. by selecting a suitable zero offset.

Programming a blank (example)
```
G54 G17 G90
CYCLE800(0,"TISCH", 100000,57,0,0,0,0,0,0,0,0,-1,100,1)
WORKPIECE(,,,"Box",112,0,-50,-80,00,155,100)
T="NC-SPOTDRILL_D16
```

Note

Blank offset for a changed zero offset

The blank is always created in the zero offset which is presently active.

If you select another zero offset, then the coordinate system is converted, however, the display of the blank is not changed.

Display of the traversing paths

The traversing paths are displayed in color. Rapid traverse is red and the feedrate is green.

Depth display

The depth infeed is color-coded. The depth display indicates the actual depth at which machining is currently taking place. The following applies to the color-coded depth display: "The deeper, the darker".

Machine references

The simulation is implemented as workpiece simulation. This means that it is not assumed that the zero offset has already been precisely scratched or is known.

In spite of this, unavoidable machine references are in the programming, such as for example, the tool change point in the machine, the retraction position when swiveling and the table components of a swivel kinematic. Depending on the actual zero offset - in the worst case - these machine references can mean that collisions are shown in the simulation that would not occur for a realistic zero offset - or vice versa, collisions are not shown, which could occur for a realistic zero offset.

Programmable frames

All frames and zero offsets are taken into account in the simulation.

Note

Manually swiveled axes

Note that swivel movement in simulation and during simultaneous recording is also displayed when the axes are swiveled manually at the start.

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.

Note

Tool display in the simulation and for simultaneous recording

In order that workpiece simulation is also possible for tools that have either not been measured or have been incompletely entered, certain assumptions are made regarding the tool geometry.

For instance, the length of a miller or drill is set to a value proportional to the tool radius so that cutting can be simulated.

Note

Thread turns not displayed

For thread and drill thread milling, the thread turns are not displayed in the simulation and for simultaneous recording.

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.

- Simultaneous recording before machining of the workpiece

  Before machining the workpiece on the machine, you can graphically display how the program will be executed during the program test and dry run feedrate. The machine axes do not move if you have selected "no axis motion".

- 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
- Side views

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.

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

Determining the program runtime

The program runtime is determined when executing the simulation. The program runtime is temporarily displayed in the editor at the end of the program.

Properties of simultaneous recording and simulation

Traversing paths

For the simulation, the displayed traversing paths are saved in a ring buffer. If this buffer is full, then the oldest traversing path is deleted with each new traversing path.

Optimum display

If simultaneous machining is stopped or has been completed, then the display is again converted into a high-resolution screen. In some cases this is not possible. In this case, the following message is output: "High-resolution image cannot be generated".

Working zone limitation

No working zone limits and software limit switches are effective in the tool simulation.

Start position for simulation and simultaneous recording

During simulation, the start position is converted via the zero offset to the workpiece coordinate system.

The simultaneous recording starts at the position at which the machine is currently located.

Restrictions

- Traori: 5-axis motion is linearly interpolated. More complex motion cannot be displayed.
- Referencing: G74 from a program run does not function.
- Alarm 15110 "REORG block not possible" is not displayed.
- Compile cycles are only partly supported.
- No PLC support.
- Axis containers are not supported.

Supplementary conditions

- All of the existing data records (Toolcarrier / TRAORI, TRACYL) are evaluated and must be correctly commissioned for correct simulation.
- Transformations with swiveled linear axis (TRAORI 64 - 69) as well as OEM transformations (TRAORI 4096 - 4098) are not supported.
- Changes to the toolcarrier or transformation data only become effective after Power On.
- Transformation change and swivel data record change are supported. However, a real kinematic change is not supported, where a swivel head is physically changed.
- The simulation of mold making programs with extremely short block change times can take longer than machining, as the computation time distribution for this application is dimensioned in favor of the machining and to the detriment of simulation.

Examples

Several examples for machine types that are supported:
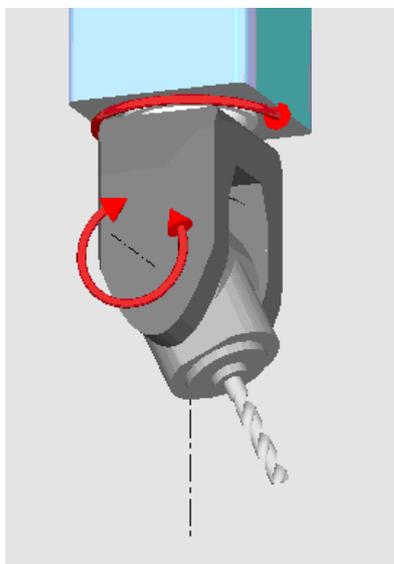
Swivel head 90°/90°
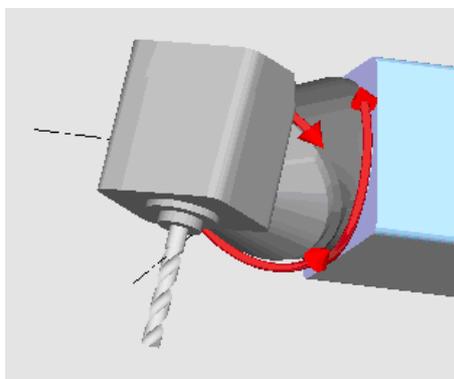
Swivel head 90°/45°
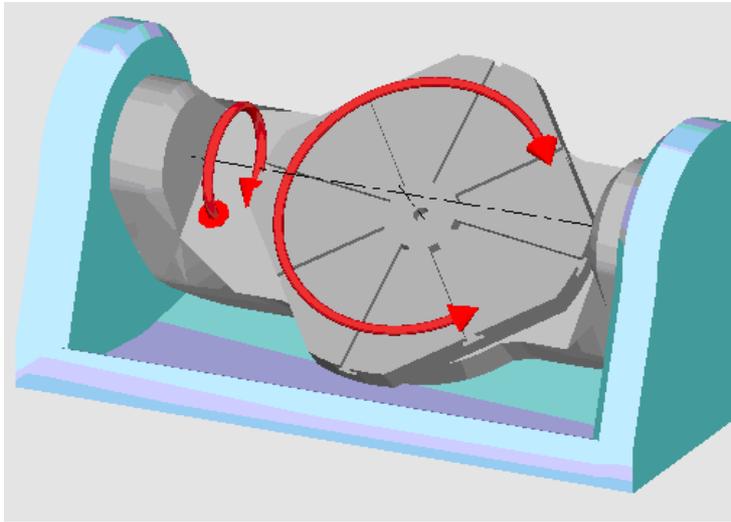
Swivel table 90°/90°
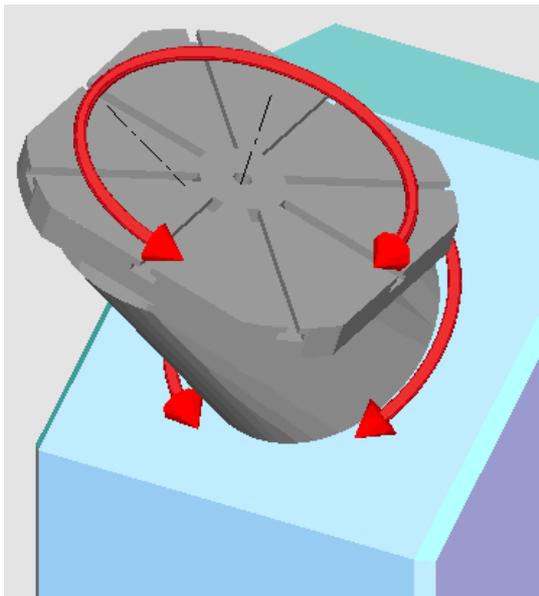
Swivel table 90°/45°
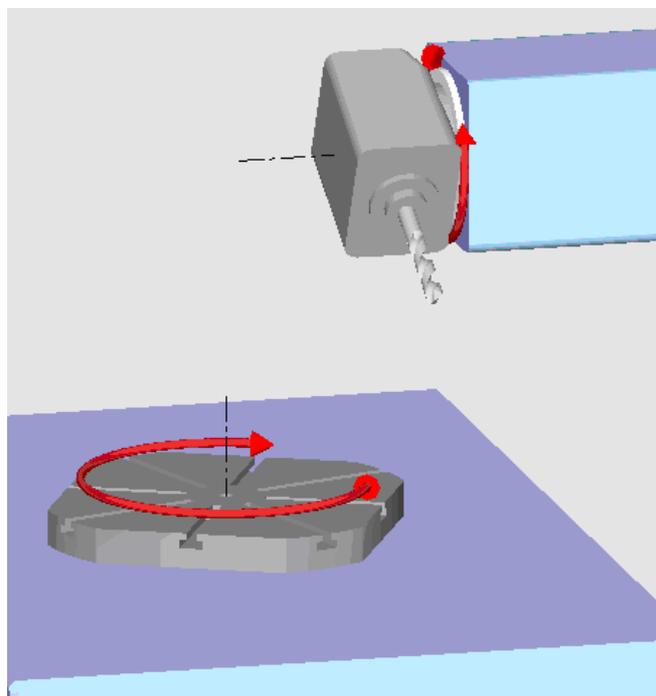
Swivel combination 90°/90°
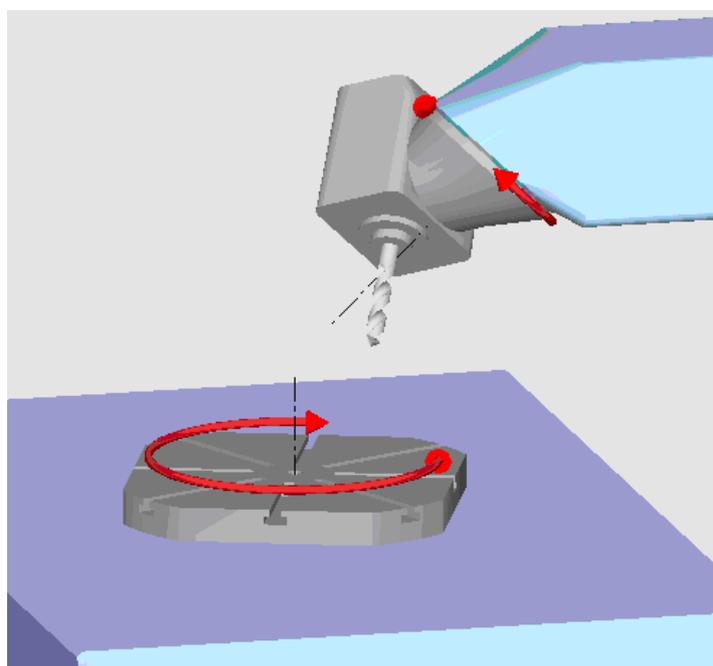
Swivel combination 45°/90°

5.2   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 162)

Simulating the program block by block (Page 163)

Procedure

Program manager	1.	Select the "Program Manager" operating area.
	2.	Select the storage location and position the cursor on the program to be simulated.
INPUT	3.	Press the <INPUT> or <Cursor right> key.
▶		- OR -
		Double-click the program. The selected program is opened in the "Program" operating area.
Simu-lation	4.	Press the "Simulation" softkey. 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.
	6.	Press the "Start" softkey to restart or continue the simulation.

Note

Operating area switchover

The simulation is exited if you switch into another operating area. If you restart the simulation, then this starts again at the beginning of the program.

5.3 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 and select the program test to disable axis motion.

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

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 "Sim. rec." softkey again to stop the recording.

5.4 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.

Software option

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.
	4.	Press the "Sim. rec." softkey again to stop the recording.

Note

- If you switch-on simultaneous recording after the unmachined part information has already been processed in the program, only traversing paths and tool are displayed.

- If you switch-off simultaneous recording during machining and then switch-on the function again at a later time, then the traversing paths generated in the intermediate time will not be displayed.

5.5 Different views of the 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
- Side views

5.5.1 Plan view

	1.	Start the simulation.
Top view	2.	Press the "Top view" softkey. The workpiece is shown from above in the top view.

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 164)

Panning a graphical representation (Page 165)

Modifying the viewport (Page 166)

5.5.2 3D view

	1.	Start the simulation.
Further views	2.	Press the "Other views" and "3D view" softkeys.
3D view		

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

5.5.3 Side view

	1.	Starting the simulation.
Further views	2.	Press the "Other views" softkey.
From front	3.	Press the "From front" softkey if you wish to view the workpiece from the front.
		- OR -
From rear		Press the "From rear" softkey if you wish to view the workpiece from the rear.
		- OR -
From left		Press the "From left" softkey if you wish to view the workpiece from the left.
		- OR -
From right		Press the "From right" softkey if you wish to view the workpiece from the right.

Changing the display

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

5.6 Editing the simulation display

5.6.1 Blank display

You have the option of replacing the blank defined in the program or to define a blank for programs in which a blank definition cannot be inserted.

Note

The unmachined part can only be entered if simulation or simultaneous recording is in the reset state.

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.
Blank		
	3.	Enter the desired values for the dimensions.
✓ **Accept**	4.	Press the "Accept" softkey to confirm your entries. The newly defined workpiece is displayed.

5.6.2 Showing and hiding the tool path

The path display follows the programmed tool path of the selected program. The path is continuously updated as a function of the tool movement. The tool paths can be shown or hidden as required.

Procedure

	1.	The simulation or the simultaneous recording is started.
▶▶	2.	Press the ">>" softkey. The tool paths are displayed in the active view.
Show tool path	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.
Delete tool path	4.	Press the '"Delete tool path" softkey. All of the tool paths recorded up until now are deleted.

5.7 Program control during simulation

5.7.1 Changing the feedrate

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

You can track the changes in the status line.

Note

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

Procedure

	1.	Simulation is started.
Program control	2.	Press the "Program control" softkey.
Override + **Override –**	3.	Press the "Override +" or "Override -" softkey to increase or decrease the feedrate by 5%.
100% override		- OR - Press the "Override 100%" softkey to set the feedrate to the maximum value.
◀◀		- OR - Press the "<<" softkey to return to the main screen and perform the simulation with changed feedrate.

Note

Toggling between "Override +" and "Override -"

CTRL ▼ ▲	Press the <CTRL> and <Cursor down> or <Cursor up> keys to toggle between the "Override +" and "Override -" softkeys.

5.7.2 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.
Program control / **Single block**	2.	Press the "Program control" and "Single block" softkeys.
Back / **SBL**	3.	Press the "Back" and "Start SBL" softkeys. The pending block of the program is simulated and then stops.
SBL	4.	Press "Start SBL" as many times as you want to simulate a single program block.
Program control / **Single block**	5.	Press the "Program control" and the "Single block" softkeys to exist the single block mode.

Note

Enabling/disabling single block

CTRL + S	Press the <CTRL> and <S> keys simultaneously to enable and disable the single block mode.

5.8 Modifying and adapting the simulation graphics

5.8.1 Enlarging or reducing the graphical representation

Precondition

The simulation or the simultaneous recording is started.

Procedure

+ ... -	1.	Press the <+> and <-> keys if you wish to enlarge or reduce the graphic display. The graphic display enlarged or reduced from the center.
Details Zoom +		- OR - Press the "Details" and "Zoom +" softkeys if you wish to increase the size of the segment.
Details Zoom -		- OR - Press the "Details" and "Zoom -" softkeys if you wish to decrease the size of the segment.
Details Autozoom		- OR - Press the "Details" and "Auto zoom" softkeys if you wish 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
Selected section

The selected sections and size changes are kept as long as the program is selected.

5.8.2 Panning a graphical representation

Precondition

The simulation or the simultaneous recording is started.

Procedure

◀ ▲ ▼ ▶	1.	Press a cursor key if you wish to move the graphic up, down, left, or right.

5.8.3 Rotating the graphical representation

In the 3D view you can rotate the position of the workpiece to view it from all sides.

Precondition

Simulation has been started and the 3D view is selected.

Procedure

Details	1.	Press the "Details" softkey.
Rotate view	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 -
↰		
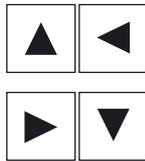
	Keep the <Shift> key pressed and then turn the workpiece in the desired direction using the appropriate cursor keys.

5.8.4 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.

Precondition

The simulation or the simultaneous recording is started.

Procedure

Details	1.	Press the "Details" softkey.
Zoom	2.	Press the "Magnifying glass" softkey. A magnifying glass in the shape of a rectangular frame appears.
Zoom +	3.	Press the "Magnify +" or <+> softkey to enlarge the frame.
		- OR -
Zoom -		Press the "Magnify -" or <-> softkey to reduce the frame.
		- OR -
		Press one of the cursor keys to move the frame up, down, left or right.
Accept	4.	Press the "Accept" softkey to accept the section.

5.8.5   Defining cutting planes

In the 3D view, you have the option of "cutting" the workpiece and therefore displaying certain views in order to show hidden contours.

Precondition

The simulation or the simultaneous recording is started.

Procedure

Details	1.	Press the "Details" softkey.
Cut	2.	Press the "Cut" softkey.
Cut active		The workpiece is displayed in the cut state.
X +	3.	Press the corresponding softkey to shift the cutting plane in the required direction.
…		
Z –		
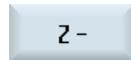
5.9   Displaying simulation alarms

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

Precondition

Simulation is running and an alarm is active.

Procedure

Program control **Alarm**	1.	Press the "Program control" and "Alarm" softkeys. The "Simulation Alarms" window is opened and a list of all pending alarms is displayed.
Acknowl. alarm		Press the "Acknowledge alarm" softkey to reset the simulation alarms indicated by the Reset or Cancel symbol. The simulation can be continued. - OR -
Simulation Power On		Press the "Simulation Power On" softkey to reset a simulation alarm indicated by the Power On symbol.

Multi-channel view (only 840D sl) 6

6.1 Multi-channel view

The multi-channel view allows you to simultaneously view several channels in the following operating areas:

- "Machine" operating area
- "Program" operating area

6.2 Multi-channel view in the "Machine" operating area

With a multi-channel machine, you have the option of simultaneously monitoring and influencing the execution of several programs.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Displaying the channels in the "Machine" operating area

In the "Machine" operating area, you can display 2 - 4 channels simultaneously.

Using the appropriate settings, you can define the sequence in which channels are displayed. Here, you can also select if you wish to hide a channel.

Note

The "REF POINT" operating mode is shown only in the single-channel view.

Multi-channel view

2 - 4 channels are simultaneously displayed in channel columns on the user interface.

- Two windows are displayed one above the other for each channel.
- The actual value display is always in the upper window.
- The same window is displayed for both channels in the lower window.
- You can select the display in the lower window using the vertical softkey bar.

  The following exceptions apply when making a selection using the vertical softkeys:

  - The "Actual values MCS" softkey switches over the coordinate systems of both channels.
  - The "Zoom actual value" and "All G functions" softkeys switch into the single-channel view.

Single-channel view

If, for your multi-channel machine, you always only wish to monitor one channel, then you can set a permanent single-channel view.

Horizontal softkeys

- Block search

  When selecting the block search, the multi-channel view is kept. The block display is displayed as search window.

- Program control

  The "Program Control" window is displayed for the channels configured in the multi-channel view. The data entered here applies for these channels together.

- If you press an additional horizontal softkey in the "Machine" operating area (e.g. "Overstore", "Synchronized actions"), then you change into a temporary single-channel view. If you close the window again, then you return to the multi-channel view.

Switching between single- and multi-channel view

![MACHINE key]	Press the <MACHINE> key in order to briefly switch between the single- and multi-channel view in the machine area.

![NEXT WINDOW key]	Press the <NEXT WINDOW> key in order to switch between the upper and lower window within a channel column.

Editing a program in the block display

![INSERT key]	You can perform simple editing operations as usual with the <INSERT> key in the actual block display.

If there is not sufficient space, you switch over into the single-channel view.

Running-in a program

You select individual channels to run-in the program at the machine.

Requirement

- Several channels have been set-up.
- The setting "2 channels", "3 channels" or "4 channels" is selected.

Displaying/hiding a multi-channel view

	1.	Select the "Machine" operating area
...	2.	Select the "JOG", "MDA" or "AUTO" mode.
	3.	Press the menu forward key and the "Settings" softkey.
	4.	Press the "Multi-channel view" softkey.
	5.	In the window "Settings for Multi-Channel View" in the selection box "View", select the required entry (e.g. "2 channels") and define the channels as well as the sequence in which they are to be displayed.
		In the basic screen for the "AUTO", "MDA" and JOG" operating modes, the upper window of the left-hand and right-hand channel columns are occupied by the actual value window.
	6.	Press the "T,F,S" softkey if you wish to view the "T,F,S" window.
		The "T,F,S" window is displayed in the lower window of the left-hand and right-hand channel column.
		Note:
		The "T,F,S" softkey is present only for smaller operator panels, i.e. up to OP012.

6.3 Multi-channel view for large operator panels

On the OP015 and OP019 operator panels as well as on the PC, you have the option of displaying up to four channels next to each one. This simplifies the creation and run-in for multi-channel programs.

Constraints

- OP015 with a resolution of 1024x768 pixels: up to three channels visible
- OP019 with a resolution of 1280x1024 pixels: up to four channels visible
- The operation of a OP019 requires a PCU50.5

3- or 4-channel view in the "Machine" operating area

Use the multi-channel view settings to select the channels and specify the view.

Channel view	Display in the "Machine" operating area
3-channel view	The following windows are displayed one above the other for each channel: • Actual Value window • T,F,S window • Block Display window Selecting functions • The T,F,S window is overlaid by pressing one of the vertical softkeys.
4-channel view	The following windows are displayed one above the other for each channel: • Actual Value window • G functions (the "G functions" softkey is omitted). "All G functions" is accessed with the Menu forward key. • T,S,F window • Block Display window Selecting functions • The window showing the G codes is overlaid if you press one of the vertical softkeys.

Toggling between the channels

1...n ↓ ↓ CHANNEL	Press the <CHANNEL> key to toggle between the channels.
NEXT WINDOW	Press the <NEXT WINDOW> key to toggle within a channel column between the three or four windows arranged one above the other.

Note

2-channel display

Unlike the smaller operator panels, the T,F,S window is visible for a 2-channel view in the "Machine" operating area.

Program operating area

You can display as many as ten programs next to each other in the editor.

Displaying a program

You can define the width of the program in the Editor window using the settings in the editor. This means that you can distribute programs evenly - or you can widen the column with the active program .

Channel status

When required, channel messages are displayed in the status display.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

6.4 Setting the multi-channel view

Setting	Meaning
View	Here, you specify how many channels are displayed. • 1 channel • 2 channels • 3 channels • 4 channels
Channel selection and sequence (for "2 - 4 channels" view)	You specify which channels in which sequence are displayed in the multi-channel view.
Visible (for "2 - 4 channels" view)	Here, you specify which channels are displayed in the multi-channel view. You can quickly hide channels from the view.

Example

Your machine has 6 channels.

You configure channels 1 - 4 for the multi-channel view and define the display sequence (e.g. 1,3,4,2).

In the multi-channel view, for a channel switchover, you can only switch between the channels configured for the multi-channel view; all others are not taken into consideration. Using the <CHANNEL> key, advance the channel in the "Machine" operating area - you obtain the following views: Channels "1" and "3", channels "3" and "4", channels "4" and "2". Channels "5" and "6" are not displayed in the multi-channel view.

In the single-channel view, toggle between all of the channels (1...6) without taking into account the configured sequence for the multi-channel view.

Using the channel menu, you can always select all channels, also those not configured for multi-channel view. If you switch to another channel, which is not configured for the multi-channel view, then the system automatically switches into the single-channel view. There is no automatic switchback into the multi-channel view, even if a channel is again selected, which has been configured for multi-channel view.

Procedure

M Machine	1.	Select the "Machine" operating area.
JOG / AUTO	2.	Select the "JOG", "MDA" or "AUTO" mode.
> / Settings	3.	Press the menu forward key and the "Settings" softkey.
Mlti-chan. view	4.	Press the "Multi-channel view" softkey. The "Settings for Multi-Channel View" window is opened.
	5.	Set the multi-channel or single-channel view and define which channels are to be seen in the "Machine" operating area - and in the editor - in which sequence.

Collision avoidance (only 840D sl) 7

7.1 Collision monitoring in the machine operator area

With the aid of collision avoidance, you can avoid collisions and therefore major damage during the machining of a workpiece or when creating programs.

Software option

You require the "Collision avoidance (machine, working area)" software option in order to use this function.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Collision avoidance is based on a machine model. The kinematics of the machine are described as a kinematic chain. For machine parts, which are to be protected, protection areas are attached to these chains. The geometry of the protection areas is defined using protection area elements. The control then knows how they move in the machine coordinate system depending on the position of the machine axes. You then subsequently define the collision pairs, i.e. two protection areas, which are monitored with respect to one another.

The "Collision avoidance" function regularly calculates the clearance from these protection areas. When two protection areas approach one another, and a specific safety clearance is reached, an alarm is displayed and before the corresponding traversing block, the program is stopped and/or the traversing motion is stopped.

References

More detailed explanations on the collision avoidance can be found in the following reference:

SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual;

Function Manual, Special Functions (FB3):

- Section: "Kinematic chain (K7)"
- Section: "Geometric machine modeling (K8)"
- Section: "Collision avoidance (K9)"
- Section: "NC/PLC interface signals (Z3)" > "Collision avoidance (K9)"

Precondition

- Collision avoidance is setup and an active machine model is available.
- In the "Collision avoidance" setting, for the AUTO mode, collision avoidance is selected.

Procedure

Machine	1.	Select the "Machine" operating area
AUTO	2.	Press the <AUTO> key.
Simult. record.	3.	Press the "Sim. rec." softkey.
Further views	4.	Press the "Other views" and "Machine area" softkeys.
Machine space		An active machine model is displayed for simultaneous recording.

7.2 Switching the collision avoidance on and off

Using "Settings", you have the option of separately activating or deactivating the collision monitoring for the Machine operating area (operating modes, AUTO, JOG and MDI) separately for the machine and tools.

Using machine data, you define from which protection level the collision monitoring for the machine or the tool can be activated or deactivated in the operating modes JOG/MDI or AUTO.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Setting	Effect
JOG/MDI operating mode Collision avoidance	They switch the collision monitoring for the operating modes JOG/MDI completely on or off.
AUTO mode Collision avoidance	They switch the collision monitoring for the operating mode AUTO completely on or off.

Setting	Effect
JOG/MDI Machine	If the collision monitoring for the JOG/MDI operating modes is activated, then as a minimum, the machine protection areas are monitored. The parameter cannot be changed.
AUTO Machine	If the collision monitoring for the AUTO operating mode is activated, then as a minimum, the machine protection areas are monitored. The parameter cannot be changed.
JOG/MDI Tools	They switch the collision monitoring of the tool protection areas for the operating modes JOG/MDI on or off.
AUTO Tools	They switch the collision monitoring of the tool protection areas for the operating mode AUTO on or off.

Procedure

	1. Select the "Machine" operating area.
	2. Select the "JOG", "MDI" or "AUTO" mode.
	3. Press the menu forward key and the "Settings" softkey.
	4. Press the "Collision avoidance" softkey. The "Collision Avoidance" window opens.
	5. In the "Collision avoidance" line for the required operating modes (e.g. for JOG/MDI), select the entry "On" to activate the collision avoidance or "Off" to deactivate collision avoidance.
	6. Deactivate the "Tools" checkbox if you only want to monitor the machine protection areas.

Tool management

<div style="text-align: right; font-size: 3em;">8</div>

8.1 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**

  You will find the magazine and magazine location-related parameters and functions for the tools / magazine locations here.

- **Tool data OEM**

  This list can be freely defined by the OEM.

  Grinding-specific tool data is provided here if you are working with

  grinding tools.

Sorting the lists

You can change the sorting within the lists according to:

- The magazine
- The name (tool identifier, alphabetic)
- The tool type
- The T number (tool identifier, numerical)
- The D number

Filtering the lists

You can filter the lists according to the following criteria:

- Only display the first cutting edge
- Only tools that are ready to use
- Only tools that have reached the pre-alarm limit
- Only locked tools
- Only tools with active code

Search functions

You have the option of searching through the lists according to the following objects:

- Tool
- Magazine location
- Empty location

8.2 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.

8.3 Tool types

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

Tool types

Figure 8-1   Example of Favorites list

Figure 8-2   Available tools in the "New Tool - Milling Cutter" window

Typ	Identifier	Tool position
200 -	Twist drill	
205 -	Solid drill	
210 -	Boring bar	
220 -	Center drill	
230 -	Countersink	
231 -	Counterbore	
240 -	Tap	
241 -	Fine tap	
242 -	Tap, Whitworth	
250 -	Reamer	

New tool – drill

Figure 8-3   Available tools in the "New Tool - Drill" window

Type	Identifier	Tool position
400 -	Surf. grinding wheel	
410 -	Facing wheel	
490 -	Dresser	

New tool – favorites
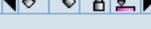
Figure 8-4   Tools listed in the "New Tool - Grinding Tools" window

Type	Identifier	Tool position
700 -	Slotting saw	
710 -	3D probe	
711 -	Edge finder	
712 -	Mono probe	
713 -	L probe	
714 -	Star probe	
725 -	Calibrating tool	
730 -	Stop	
900 -	Auxiliary tools	
	Multitool	

New tool – special tools

Figure 8-5   Available tools in the "New Tool - Special Tools" window

8.4 Tool dimensioning

This section provides an overview of the dimensioning of tools.

Tool types
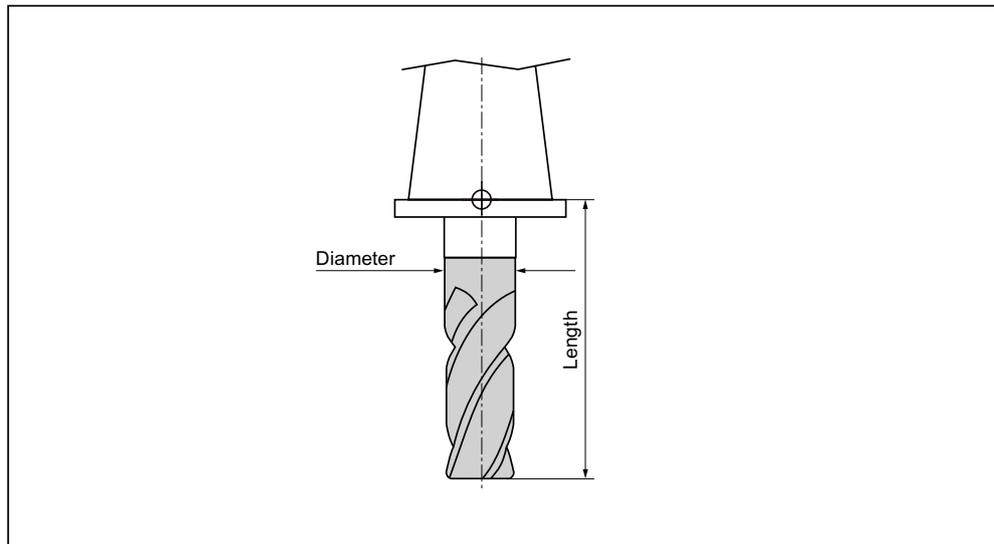
Figure 8-6   End mill (Type 120)
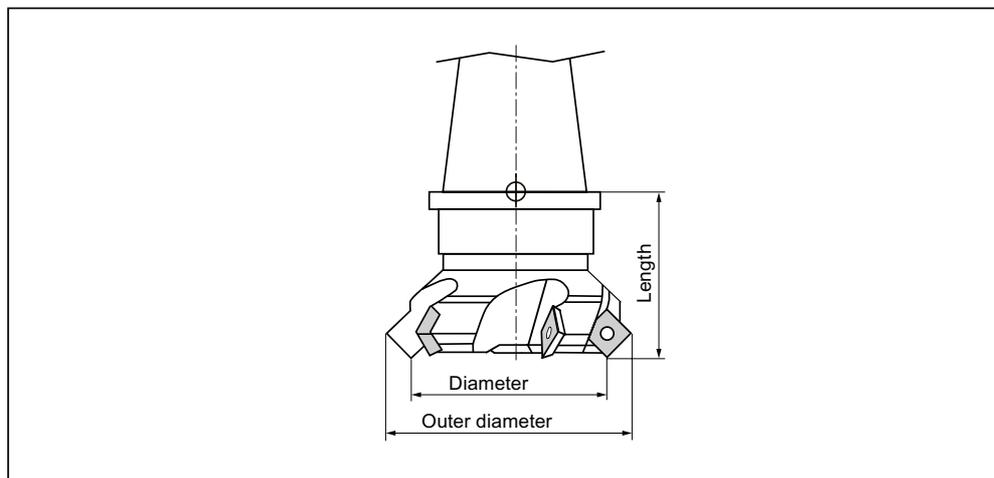
Figure 8-7   Face mill (Type 140)
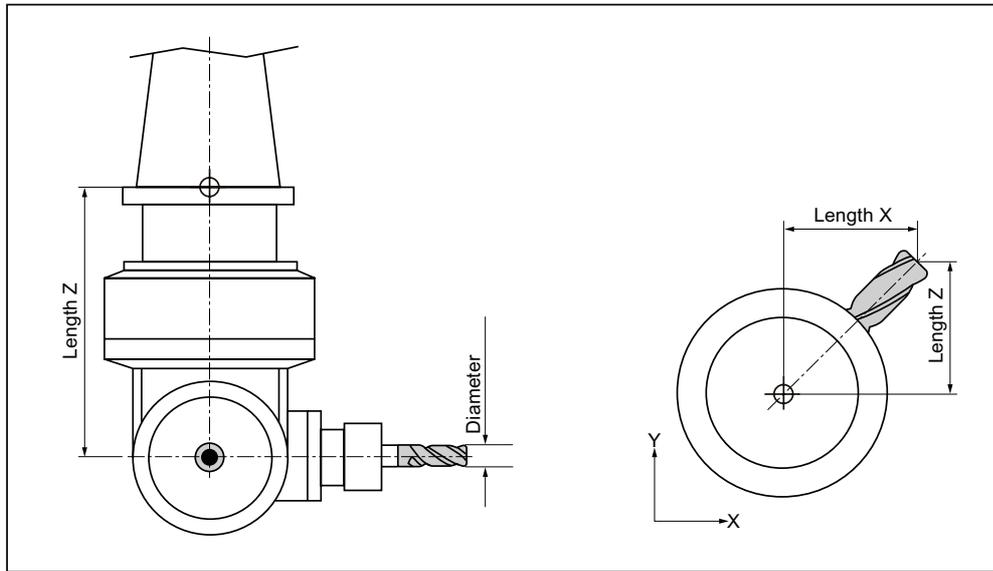
Figure 8-8   Angle head cutter (Type 130)
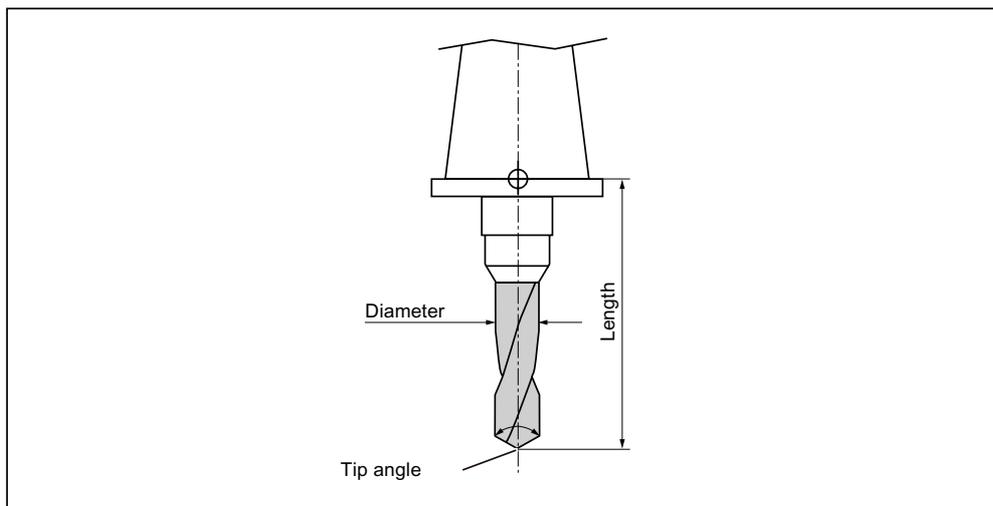
Figure 8-9   Drill (Type 200)
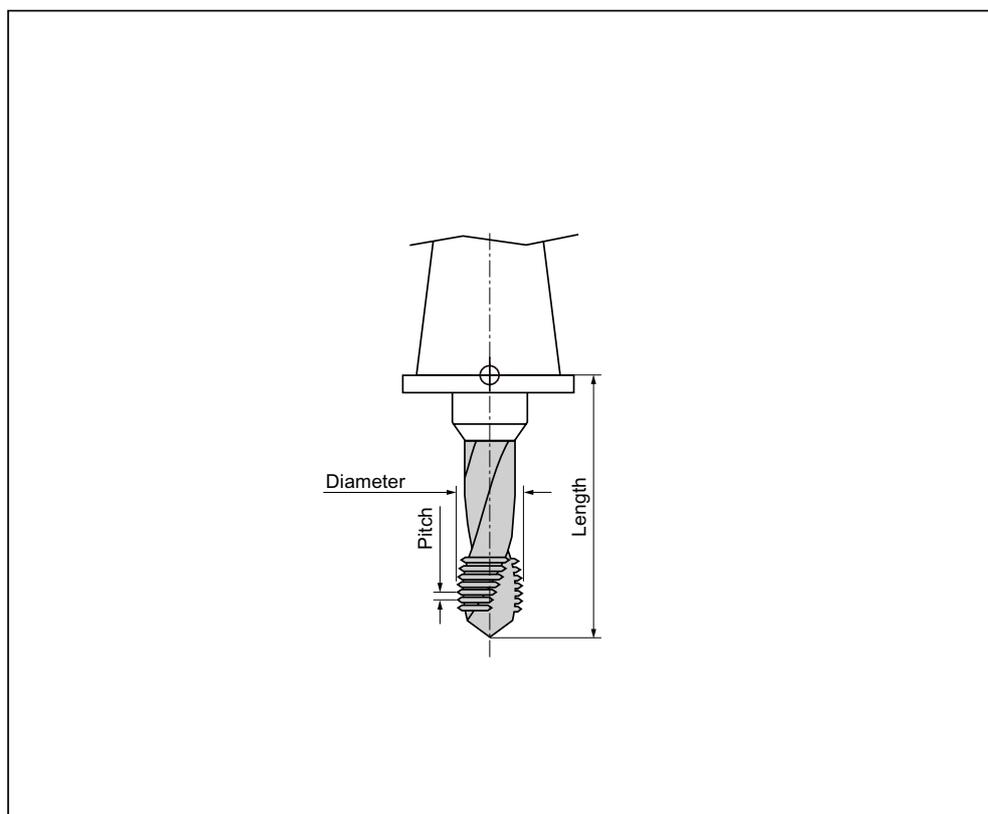
Figure 8-10   Tap (Type 240)
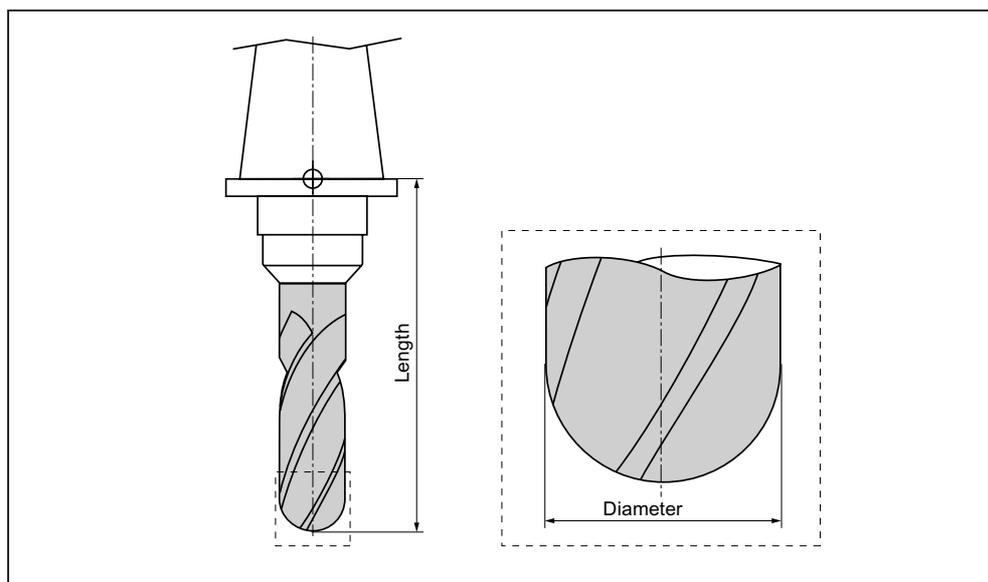
Figure 8-11   3D tool with an example of a cylindrical die-sinking cutter (Type 110)
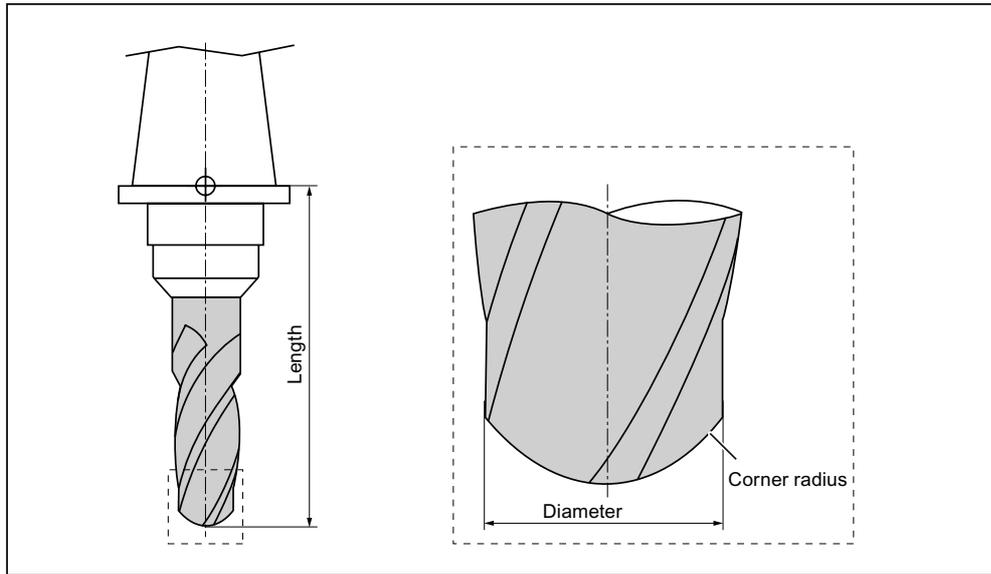
Figure 8-12   3D tool type with an example of a ballhead cutter (Type 111)
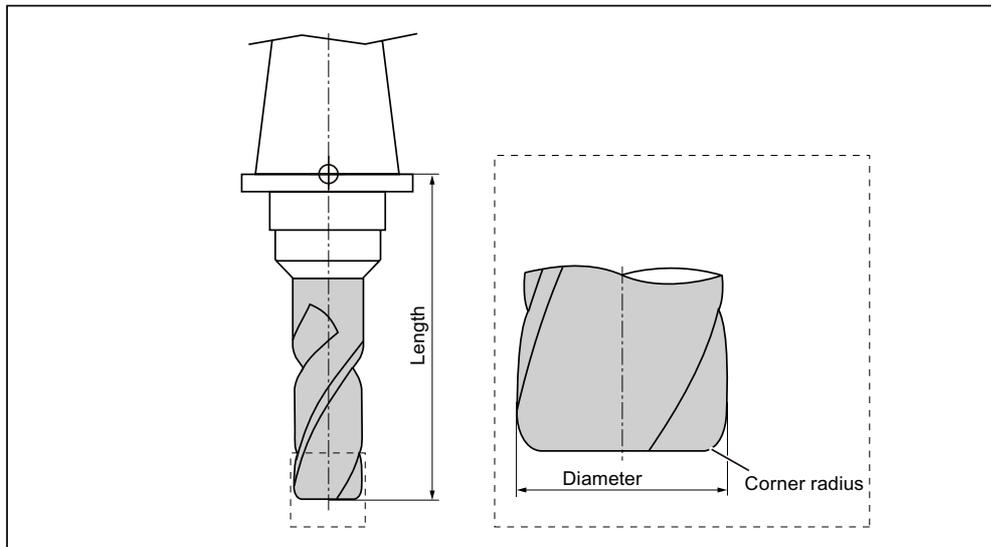
Figure 8-13   3D tool with an example of an end mill with corner rounding (Type 121)
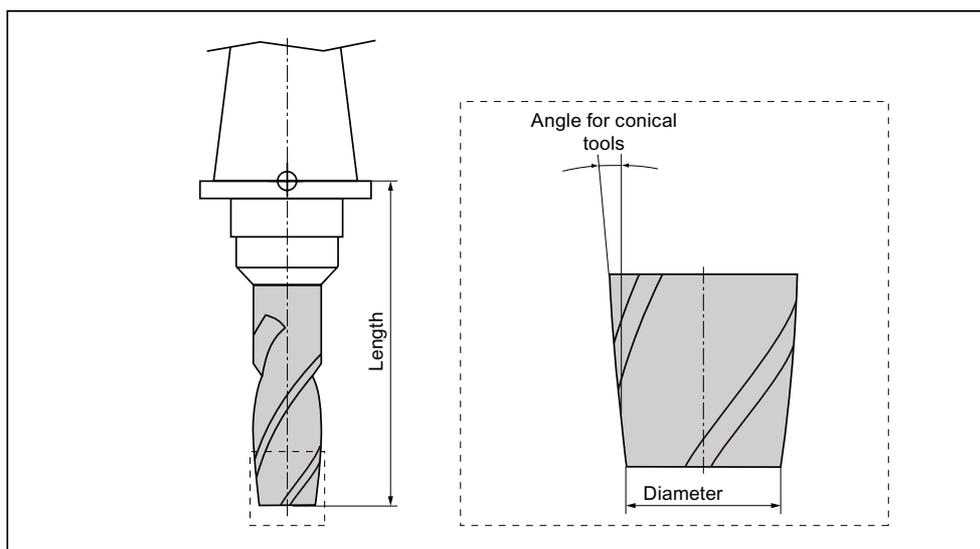
Figure 8-14   3D tool type with an example of a bevel cutter (Type 155)
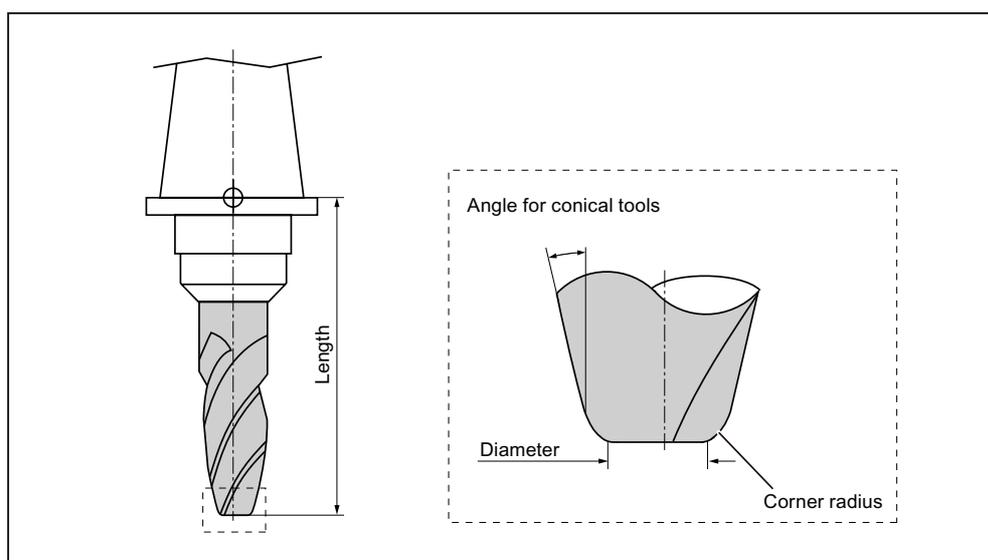
Figure 8-15   3D tool with an example of a bevel cutter with corner rounding (Type 156)
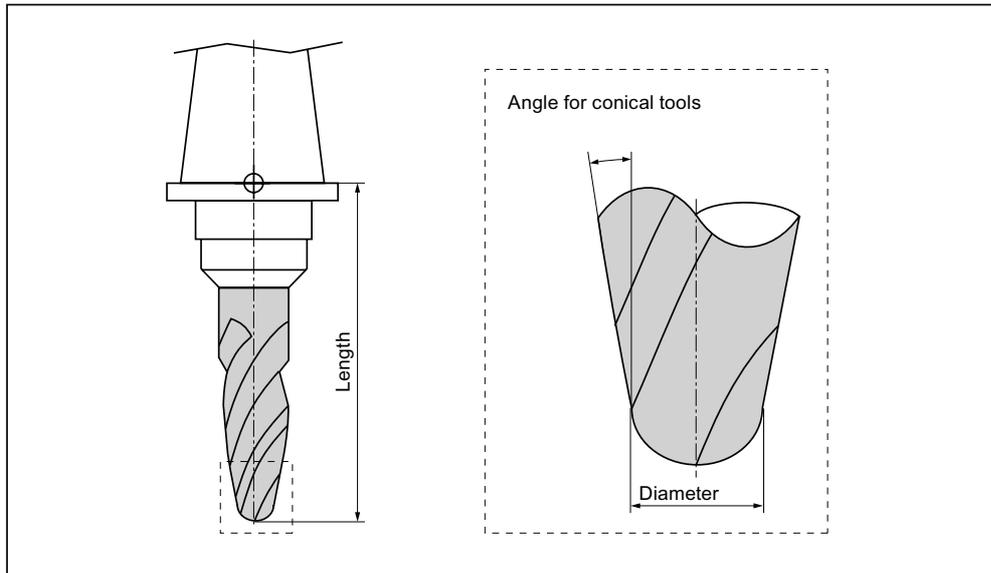
Figure 8-16   3D tool with an example of a tapered die-sinking cutter (Type 157)
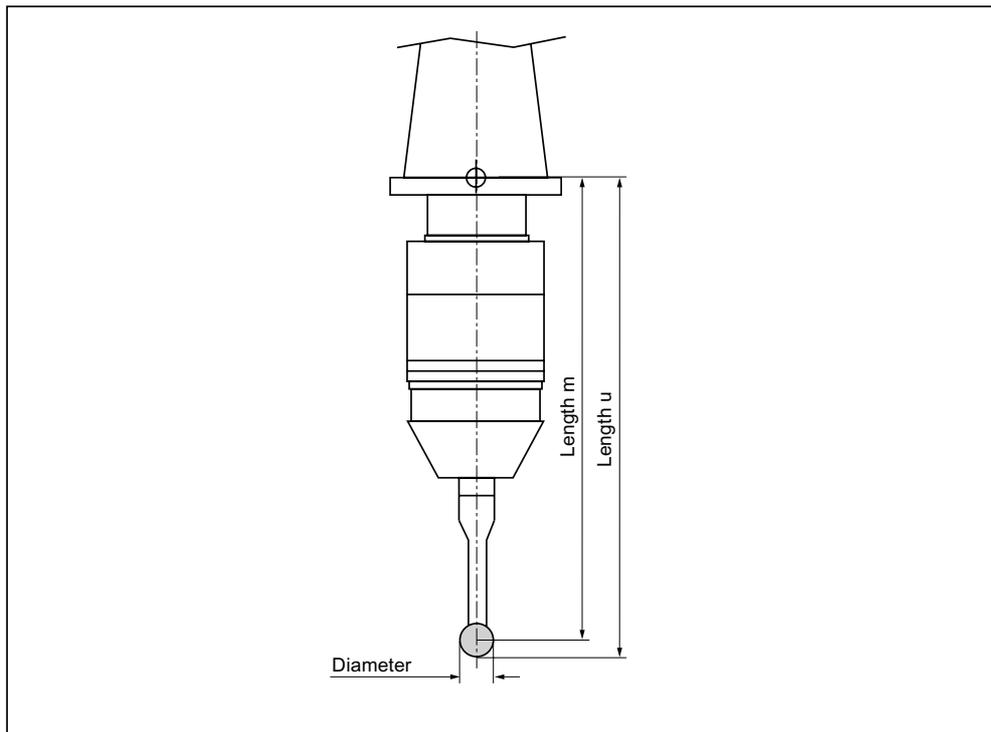
Figure 8-17   Electronic workpiece probe

Machine manufacturer

The tool length of the workpiece probe 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

An electronic workpiece probe must be calibrated before use.

8.5 Tool list

8.5.1 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	Magazine/location number
	• The magazine location numbers
	The magazine number is specified first, followed by the location number in the magazine.
BS	If there is only one magazine, only the location number is displayed.
	• Load position in the load magazine
⏚	The following icons can also be displayed for other magazine types (e.g. for a chain):
⟩ ⟨	• Spindle location as an icon
* If activated in magazine selection	• Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons.
Type	Tool type
	Specific tool offset data is displayed depending on the tool type (represented as an icon).

Column heading	Meaning
SELECT	You have the option of changing the tool type using the <SELECT> key.
Tool name	The tool is identified by the name and the replacement tool number. You may enter the names as text or numbers. **Note:** The maximum length of tool names is 31 ASCII characters. The number of characters is reduced for Asian characters or Unicode characters. The following special characters are not permitted: | # ".
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 - centering tool and Type 230 countersink Pitch for Type 240 - tap
N	Number of teeth for Type 100 - milling tool, Type 110 - ball end mill for cylindrical die-sinking cutter, Type 111 - ball end mill or tapered die-sinking cutter, Type 120 - end mill, Type 121 - end mill with corner rounding, Type 130 - angle head cutter, Type 131 - angle head cutter with corner rounding, 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.
	Direction of spindle rotation Spindle is not switched on CW spindle rotation CCW spindle rotation
	Coolant 1 and 2 (e.g. internal and external cooling) can be switched on and switched off. The coolant infeed at the machine does not necessarily have to be set-up.
M1 - M4	Other tool-specific functions such as additional coolant infeed, monitoring functions for speed, tool breakage, etc.

Further parameters

If you have created unique cutting edge numbers, they will be displayed in the first column.

Column heading	Meaning
D no.	Unique cutting edge number
SN	Cutting edge number
EC 	Setting-up offsets Display of the available setup offsets

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

Machine manufacturer

Please refer to the machine manufacturer's specifications.

References

Information about the configuration and setup of the tool list can be found in the following references:

SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual

Icons in the tool list

Icon/ Marking		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

Parameter	1.	Select the "Parameter" operating area.
Tool list	2.	Press the "Tool list" softkey. The "Tool List" window opens.

8.5.2 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, grinding or special tool via the corresponding softkeys.

Note

Grinding tools

Only grinding tools are available depending on the particular machine configuration.

Procedure

Tool list	1..	The tool list is opened.
	2.	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 position the cursor on an existing tool in the area of the NC tool memory. Data from the displayed tool will not be overwritten.
New tool	3.	Press the "New tool" softkey.
Favorites		The "New tool - favorites" window opens.
		- OR -
Cutters 100-199		If you want to create a tool that is not in the "Favorites" list, press the softkey "Cutters 100-199", "Drill 200-299", Grinders 400-499" or " Spec.tool 700-900".
...		The "New tool - milling cutter", "New tool - drill", "New tool - grinding tool" or "New tool - special tools" window opens.
Spec.tool 700-900		
	4.	Select the tool by placing the cursor on the corresponding icon.
OK	5.	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 "Select loading point" 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

Commissioning Manual SINUMERIK Operate / SINUMERIK 840D sl

8.5.3 Additional data

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

Tools with additional geometry data

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
585 Calibration tool	Geometry length (length X, length Y, length Z)
	Wear length (Δ length X, Δ length Y, Δ length Z)
710 3D-probe milling	Geometry length (length X, length Y, length Z)
	Wear length (Δ length X, Δ length Y, Δ length Z)
712 Mono-probe	Geometry length (length X, length Y, length Z)
	Wear length (Δ length X, Δ length Y, Δ length Z)
713 L-probe	Geometry length (length X, length Y, length Z)
	Wear length (Δ length X, Δ length Y, Δ length Z)
	Boom length (length)
714 Star-type probe	Geometry length (length X, length Y, length Z)
	Wear length (Δ length X, Δ length Y, Δ length Z)
	Outer diameter (∅)

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

Tool list	1.	The tool list is opened.
	2.	In the list, select an appropriate tool, e.g. an angle head cutter.
Further data	3.	Press the "Additional data" softkey.
		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.

8.5.4   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

Tool list	1.	The tool list is opened.
	2.	Position the cursor on the tool for which you would like to store more cutting edges.
Edges	3.	Press the "Edges" softkey in the "Tool list".
New cuttgEdge	4.	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.
	5.	Enter the offset data for the 2nd cutting edge.
	6.	Repeat this process if you wish to create more tool edge offset data.
Delete cut. edge	7.	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.

8.5.5 Deleting a tool

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

Procedure

Tool list	1.	The tool list is opened.
	2.	Place the cursor on the tool that you would like to delete.
Delete tool	3.	Press the "Delete tool" softkey. A safety prompt is displayed.
OK	4.	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.

8.5.6 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 NC memory.

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 then automatically saves the tool data in the NC memory.

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

Tool list	1.	The tool list is opened.
	2.	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).
Load	3.	Press the "Load" softkey.

The "Load to… " window opens.

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

OK	4.	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.

Loading empty magazine location directly with tool

Load **OK**	1.	Position the cursor at an empty magazine location where you want to load a tool and press the "Load" softkey. The "Load with ..." window opens. Select the desired tool in the " ... Tool" field and press the "OK" softkey.

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

Unload	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.
OK	3.	Confirm your selection with "OK".
		- OR -
Cancel		Undo your selection with "Cancel".

8.5.7 Selecting a magazine

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

Procedure

Tool list	1.	The tool list is opened.
Magazine selection	2.	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 -
Go to		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

Commissioning Manual SINUMERIK Operate / SINUMERIK 840D sl

8.5.8   Code carrier connection (only 840D sl)

You have the option of configuring a connection to MCIS TDI Ident Connection.

This means that the following functions are available in SINUMERIK Operate:

* Creating a new tool from code carrier
* Unloading tools on code carrier

Software option

In order to use the functions, you require the option "MC Information System TDI Ident Connection".

References

Further details on tool management with code carrier and the configuration of the user interface in SINUMERIK Operate can be found in the following reference:

* Function Manual MCIS TDI Ident Connection
* Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

With a code carrier connection, in the list of favorites, there is also a tool available.

Type	Identifier	Tool position
	Tool from code carrier	
120 -	End mill	
140 -	Facing tool	
200 -	Twist drill	
220 -	Center drill	
240 -	Tap	
710 -	3D milling probe	
711 -	Edge finder	
110 -	Ball nose end mill	
111 -	Conical ball end	
121 -	End mill corner rounding	
155 -	Bevelled cutter	
156 -	Bevelled cutter corner	
157 -	Tap. die-sink. cutter	
	Multitool	

Figure 8-18   New tool from code carrier in the list of favorites

Creating a new tool from code carrier

1. The tool list is opened.

2. Place the cursor in the tool list at the position where the new tool should be created.

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

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

3. Press the "New tool" softkey.

   The "New Tool - Favorites" window is opened.

4. Position the cursor on the entry "Tool from code carrier" and press the "OK" softkey.

   The tool data is read from the code carrier, and is displayed in the "New tool" window with the tool type, tool name and possibly with certain parameters.

OK	5.	Press the "OK" softkey.
		The tool is added to the tool list with the specified 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.

Unloading tool on code carrier

Tool list	1.	The tool list is opened.
Unload On code carr.	2.	Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" and "On code carrier" softkeys.
		The tool is unloaded and the data of the tool are written to the code carrier.

According to the corresponding setting, the unloaded tool on the code carrier is deleted from the MC memory after reading out the code carrier.

Deleting tool on code carrier

Tool list	1.	The tool list is opened.
	2.	Position the cursor on the tool on code carrier that you want to delete.
Delete tool On code carr.	3.	Press the "Delete tool" and "On code carrier" softkeys.
		The tool is unloaded and the data of the tool are written to the code carrier. The tool is then deleted from the NC memory.

The deletion of the tool can be set differently, i.e. the "On code carrier" softkey is not available.

References

A description of the configuration options can be found in the following reference:

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

8.6 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.

Monitoring types

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

Note

Combination of monitoring types

You have the option to activate the monitoring of a tool by type or any combination of monitoring types.

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

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

Tool parameters

Column heading	Meaning
Location	Magazine/location number
	• The magazine location numbers 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.
BS	• Load position in the load magazine
	The following icons can also be displayed for other magazine types (e.g. for a chain):
⎇	• Spindle location as an icon
⊃ ⊂	• Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons.
* If activated in magazine selection	

Column heading	Meaning
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.
	Note: The maximum length of tool names is 31 ASCII characters. The number of characters is reduced for Asian characters or Unicode characters. The following special characters are not permitted: | # ".
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.
Tool life Workpiece count Wear * *Parameter depends on selection in TC	Tool life Number of workpieces Tool wear
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.
G	The tool is disabled when the checkbox is selected.

Further parameters

If you have created unique cutting edge numbers, they will be displayed in the first column.

Column heading	Meaning
D no.	Unique cutting edge number
SN	Cutting edge number
SC 🔵	Setting-up offsets Display of the available setup offsets

Icons in the wear list

Icon/ Marking		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

Parameter	1.	Select the "Parameter" operating area.
Tool wear	2.	Press the "Tool wear" softkey.

See also

Changing a tool type (Page 219)

8.6.1 Reactivating a tool

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

Preconditions

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

Procedure

Tool wear	1.	The tool wear list is opened.
	2.	Position the cursor on the disabled tool which you would like to reuse.
Reacti-vate	3.	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

Commissioning Manual SINUMERIK Operate / SINUMERIK 840D sl

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.

8.7 Tool data OEM

You have the option of configuring the list according to your requirements.

Depending on the machine configuration, grinding-specific parameters are displayed in the list with OEM tool data.

Grinding tool-specific parameters

Column heading	Meaning
Min. radius	Limit value for the radius of the grinding wheel to monitor the geometry.
Actual radius	Displays the sum of the geometry value, wear value and, if set, the base dimension.
Min. width	Limit value for the width of the grinding wheel to monitor the geometry.
Actual width	The width of the grinding wheel measured, for example, after the dressing.
Max. speed	Maximum speed
Max. peripheral velocity	Maximum peripheral velocity
Wheel angle	Angle of inclined wheel
Monitoring	Monitoring geometry and speed
Tool base dimension	Defines whether the base dimension is used when calculating the SUG and when monitoring the minimum wheel radius.
Spindle number	Number of programmed spindle (e.g. grinding wheel peripheral velocity) and spindle to be monitored (e.g. wheel radius and width).
Param. rad.calc.	Selects the parameter to calculate the radius • Length X • Length Y • Length Z • Radius
Chaining rule	This parameter defines which tool parameters of tool cutting edge 2 (D2) and tool cutting edge 1 (D1) have to be chained to one another. When the value of a chained parameter is modified, then the other cutting edge is automatically used when chaining the parameter.

References

You can find more information on grinding tools in the following manual:

Function Manual, Extended Functions W4: Grinding-specific tool offset and monitoring functions / SINUMERIK 840D sl

Refer to the following document for more information on configuring OEM tool data:

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

Procedure
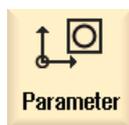
<table>
<tr><td></td><td>1.</td><td>Select the "Parameter" operating area.</td></tr>
<tr><td>OEM Tool</td><td>2.</td><td>Press the "OEM tool" softkey.</td></tr>
<tr><td></td><td>3.</td><td>Position the cursor on a grinding tool.</td></tr>
</table>

8.8 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	Magazine/location number
	• The magazine location numbers 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.
BS	• Load position in the load magazine The following icons can also be displayed for other magazine types (e.g. for a chain):
⊥	• Spindle location as an icon
> <	• Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons
* If activated in magazine selection	
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. **Note**: The maximum length of tool names is 31 ASCII characters. The number of characters is reduced for Asian characters or Unicode characters. The following special characters are not permitted: | # ".

Column heading	Meaning
ST	Replacement tool number (for replacement tool strategy).
D	Cutting edge number
G	Disabling of the magazine location.
Mag.loc. type	Display of magazine location type.
Tool.loc. type	Display of tool location type.
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.

Further parameters

If you have created unique cutting edge numbers, they will be displayed in the first column.

Column heading	Meaning
D no.	Unique cutting edge number
SN	Cutting edge number

Magazine list icons

Icon/ Marking		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.

See also

Changing a tool type (Page 219)

8.8.1 Position magazine

You can position magazine locations directly on the loading point.

Procedure

	1.	The magazine list is opened.
	2.	Place the cursor on the magazine location that you want to position onto the load point.
	3.	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.

8.8.2 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.

Buffer

You have the option of relocating the tool to buffer locations.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

Maga-zine	1.	The magazine list is opened.
	2.	Position the cursor on the tool that you wish to relocate to a different magazine location.
Relocate	3.	Press the "Relocate" softkey.
		The "... relocate from location ... to location ..." window is displayed. The "Location" field is pre-assigned with the number of the first empty magazine location.
OK	4.	Press the "OK" softkey to relocate the tool to the recommended magazine location.
		- OR -
OK		Enter the required magazine, enter the location number and press the "OK" softkey.
		- OR -
		Enter the number "9998" or the number "9999" into the "... magazine" field in order to select the buffer as well as the required buffer location in the "Location" field.
		- OR -
Spindle OK		Press the "Spindle" softkey to load a tool into the spindle and press the "OK" softkey.
		The tool is relocated to the specified magazine location, in the spindle or in the buffer.

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.

8.8.3 Unload all tools

You have the option of unloading all tools from the magazine list. A single request successively unloads the tools from the list.

Requirement

The following requirements must be satisfied so that the "Unload all" softkey is displayed and available:

- Magazine management is set up
- There is no tool in the buffer / in the spindle

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

Maga-zine	1. The magazine list is opened.
Unload all	2. Press the "Unload all" softkey. A prompt is displayed as to whether you really want to unload all tools.
OK	3. Press the "OK" softkey to continue with the unloading of the tools. The tools are unloaded from the magazine in ascending magazine location number order.
	4. Press the "Cancel" softkey if you wish to cancel the unloading operation.

Multiple load points

For a magazine, if more than one loading point was set-up, using the "Select loading point" softkey, you have the option of opening a window in which you can assign a loading point to a magazine.

8.9 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 easily in the lists.

Procedure
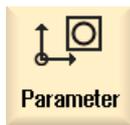
Parameter	1.	Select the "Parameter" operating area.
Tool list ... **Magazine**	2.	Press the "Tool list", "Tool wear" or "Magazine" softkey.
▶▶ **Sort ▶**	3.	Press the ">>" and "Sort" softkeys.
Acc. to magazine		The lists are displayed sorted numerically according to magazine location. Tool types are used to sort tools with the same magazine location. Identical types (e.g. milling cutters), in turn, are sorted according to their radius value.
Acc. to type	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 -
Surname		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 -
Acc. to T number		Press the "Acc. to T number" softkey to display the tool names sorted numerically.
		- OR -
By D number		Press the "Acc. to D number" softkey to display the tools sorted by D number.
		The list is sorted according to the specified criteria.
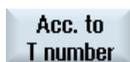
Machine manufacturer

Please refer to the machine manufacturer's specifications.

8.10 Filtering the tool management lists

The filter function allows you to filter-out tools with specific properties in the tool management lists.

For instance, you have the option of displaying tools during machining that have already reached the prewarning limit in order to prepare the corresponding tools to be loaded.

Filter criteria

- Only display the first cutting edge
- Only tools that are ready to use
- Only tools that have reached the prewarning limit
- Only tools with active code
- Only locked tools
- Only tools with remaining quantity of ... to ...
- Only tools with residual tool life of ... to ...
- Only tools with unloading marking
- Only tools with loading marking

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Note

Multiple selection

You have the option of selecting several criteria. You will receive an appropriate message if conflicting filter options are selected.

You can configure OR logic operations for the various filter criteria.

References

A description of the configuration options is provided in
SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual
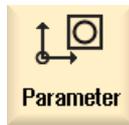
Procedure

	1.	Select the "Parameter" operating area.
...	2.	Press the "Tool list", "Tool wear" or "Magazine" softkey.
...	3.	Press the ">>" and "Filter" softkeys. The "Filter" window opens.
	4.	Activate the required filter criterion and press the "OK" softkey. The tools that correspond to the selection criteria are displayed in the list. The active filter is displayed in the window header.

8.11 Specific search in the tool management lists

There is a search function in all tool management lists, where you can search for the following objects:

- **Tools**
    - You enter a tool name. You can narrow down your search by entering a replacement tool number.

      You have the option of only entering a part of the name as search term.

    - Enter the D number and, if necessary, activate the "Active D number" checkbox.

- **Magazine locations or magazines**

  If only one magazine is configured, then the search is only made for the magazine location.

  If several magazines are configured, then it is possible to search a specific magazine location in a specific magazine or just to search in a specific magazine.

- **Empty locations**

  If the lists with the location type are used, then the empty location search is made using the location type and location size.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

Parameter	1.	Select the "Parameter" operating area.
Tool list	2.	Press the "Tool list", "Tool wear" or "Magazine" softkey.
...		
Maga- zine		
▶▶	3.	Press the ">>" and "Search" softkeys.
...		
Search		
Tool	4.	Press the "Tool" softkey if you wish to search for a specific tool.
		- OR -
Magazine location		Press the "Magazine location" softkey if you wish to search for a specific magazine location or a specific magazine.
		- OR -
Empty location		Press the "Empty location" softkey if you wish to search for a specific empty location.

8.12 Displaying tool details

All of the parameters of the selected tool are listed in the "Tool Details - All Parameters" window.

The parameters are displayed, sorted according to the following criteria

- Tool data
- Grinding data (if grinding tools have been configured)
- Cutting edge data
- Monitoring data

Protection level

You require access level, key switch 3 (protection level 4) in order to edit the parameters in the detail window.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

Tool list ... **Maga- zine**	1.	The tool list, the wear list, the OEM tool list or the magazine is opened.
	2.	Position the cursor to the desired tool.
▶▶	3.	If you are in the tool list or in the magazine, press the ">>" and "Details" softkey.
Details		
		- OR -
Details		If you are in the wear list or OEM tool list, press the "Details" softkey.
Tool data		The "Tool Details" window is displayed. All of the available tool data is displayed in the list.
Grinding data	4.	Press the "Grinding data" softkey if you want to display the grinding data.
Cutting edge data	5.	Press the softkey "Cutting edge data" if you wish to display the cutting edge data.

	6.	Press the softkey "Monitoring data" if you wish to display the monitoring data.
	7.	Press the "Additional details" softkey.
		The "Tool Details - All Parameters" window is opened.
		All of the tool parameters are displayed in the list.

8.13 Changing a tool type

Procedure

	1.	The tool list, the wear list, the OEM tool list or the magazine is opened.
...		
	2.	Position the cursor in the column "Type" of the tool that you wish to change.
	3.	Press the <SELECT> key.
		The "Tool types - Favorites" window opens.
	4.	Select the desired tool type in the list of favorites or select using the softkeys "Cutters 100-199", "Drill 200-299", "Grinders 400-499" or " Spec.tool 700-900".
		Note: A grinding tool can only be changed into another grinding tool type.
	5.	Press the "OK" softkey.
		The new tool type is accepted and the corresponding icon is displayed in the "Type" column.

8.14 Settings for tool lists

The "Settings" window provides the following options to set the view in the tool lists:

- Display only one magazine in the magazine sorting

    – You limit the display to one magazine. The magazine is displayed with the assigned buffer locations and the not-loaded tools.

    – You use a configuration to specify whether the "Magazine selection" softkey jumps to the next magazine or whether the "Magazine Selection" dialog for changing switches to any magazine.

- Display only spindles in the buffer

    In order to display during operation only the spindle location, the remaining locations of the buffer will be hidden.

- Activating the adapter-transformed view

    – The geometry lengths and the operation offsets are displayed transformed in the tool list.

    – The wear lengths and the sum offsets are displayed transformed in the tool wear list.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

References

Further information about configuring the settings are shown in the following reference:

SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual

Procedure
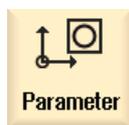
	1.	Select the "Parameter" operating area.
	2.	Press the "Tool list", "Tool wear" or "Magazine" softkey.
...		
	3.	Press the "Continue" and "Settings" softkeys.
	4.	Activate the checkbox for the desired setting.

Managing programs

<div style="text-align: right; font-size: 3em;">9</div>

9.1 Overview

You can access 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

Possible interruption when executing 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.

Multiple clamping with ShopMill

With ShopMill, you can implement multiple clamping of identical or different workpieces with optimization of the tool series.

   Software options

Multiple clamping is only possible with ShopMill programs. You require the "ShopTurn/ShopMill" option for this.

Storage for programs

Possible storage locations are:

- NC
- Local drive
- Network drives
- USB drives
- V24
- FTP drives

 Software options

To display the "Local drive" softkey, you require the "Additional HMI user memory on CF card of the NCU" option (not for SINUMERIK Operate on PCU50 or PC/PG).

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
- FTP drive

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).

Displaying PDF and HTML documents

You can display HTML documents as well as PDFs on drives of the program manager (e.g. in the local drive or USB) and via the data tree of the system data.

However, a preview of the documents is only possible for PDFs.

Note

FTP drive

It is not possible to preview documents on the FTP drive.

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 9-1   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 be a maximum of 24 characters long

  Permissible characters include all upper-case letters (without accents), numbers, and underscores.

- Type

  Directory: WPD

  Program: MPF

  Subprogram: SPF

  Initialization programs: INI

  Job lists: JOB

  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)

Active programs

Selected, i.e. active programs are identified using a green symbol.

CHAN1	Name	Type	Length	Date	Time
⊞ 🗀 Part programs		DIR		11/30/09	3:49:09 PM
⊞ 🗀 Subprograms		DIR		12/02/09	11:24:33 AM
⊟ 🗁 Workpieces		DIR		12/02/09	2:53:07 PM
⊞ 🗀 DREHEN1		WPD		12/02/09	8:40:58 AM
⊞ 🗀 GGG		WPD		12/01/09	12:03:39 PM
⊞ 🗀 JOBSHOP_MEHRK		WPD		12/03/09	9:18:27 AM
⊞ 🗀 MEHR		WPD		11/30/09	3:49:23 PM
⊞ 🗀 MEHRKANAL		WPD		12/02/09	12:47:20 PM
⊞ 🗀 SIM_CHESS_KING		WPD		11/30/09	3:49:14 PM
⊞ 🗀 SIM_CHESS_LADY_26		WPD		11/30/09	3:49:14 PM
⊞ 🗀 SIM_CHESS_TOWER		WPD		11/30/09	3:49:15 PM
⊞ 🗀 SIM_ZYK_T_26		WPD		11/30/09	3:49:17 PM
⊟ 🗁 SWOB		WPD		12/03/09	8:39:49 AM
└ 📄 UT		MPF	205	12/03/09	3:22:48 PM
⊞ 🗀 TEMP		WPD		11/30/09	3:49:33 PM

Figure 9-2   Active program shown in green

9.1.1   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.

9.1.2 Local drive

Workpieces, main and subprograms that are saved in the user memory of the CF card or on the local hard disk are displayed.

For archiving, you have the option of mapping the structure of the NC memory system or to create a separate archiving system.

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

Software options

To display the "Local drive" softkey, you require the "Additional HMI user memory on CF card of the NCU" option (not for SINUMERIK Operate on PCU50 or PC/PG).

Procedure

Program manager	1.	Select the "Program manager" operating area.
Local drive	2.	Press the "Local drive" softkey.

On the local drive, you have the option of mapping the directory structure of the NC memory. This also simplifies the search sequence.

Creating Directories

Local drive	1.	The local drive is selected.
CHAN1	2.	Position the cursor on the main directory.
New / Directory	3.	Press the "New" and "Directory" softkeys. The "New Directory" window opens.
OK	4.	In the "Name" entry field, enter "mpf.dir", "spf.dir" and "wks.dir" and press the "OK" softkey. The directories "Part programs", "Subprograms" and "Workpieces" are created below the main directory.

9.1.3 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
Interruption of operation
Direct execution from the USB FlashDrive is not recommended, because machining can be undesirably interrupted, therefore resulting in workpiece damage.

Partitioned USB-FlashDrive (only 840D sl and TCU)

If the USB-FlashDrive has several partitions, these are displayed in a tree structure as a subtree (01,02,...).

For EXTCALL calls, enter the partition (e.g. USB:/02/... or //ACTTCU/FRONT/02/... or //ACTTCU/FRONT,2/... or //TCU/TCU1/FRONT/02/...)

You can also configure any partition (e.g. //ACTTCU/FRONT,3).

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.

9.1.4 FTP drive

The FTP drive offers you the following options - to transfer data, e.g. part programs, between your control system and an external FTP server.

You have the option of archiving any files in the FTP server by creating new directories and subdirectories.

Note

Selecting a program / execution

It is not possible to select a program directly on the FTP drive, and change to execution in the "Machine" operating area.

Precondition

User name and password have been set up in the FTP server.

Procedure

Program manager	1.	Select the "Program manager" operating area.
FTP	2.	Press the "FTP" softkey. When selecting the FTP drive for the first time, a login window is displayed.
OK	3.	Enter the user name and password and press the "OK" softkey to log into the FTP server. The content of the FTP server with its folders is displayed.
Log off	4.	Press the "Log out" softkey after the required data processing has been completed. The connection to the FTP server is disconnected. In order to reselect the FTP drive, you must log on again.

9.2   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.

Note

Channel changeover in the editor

When opening the program, the editor is opened for the currently selected channel. This channel is used to simulate the program.

If you change over a channel in the editor, this does not influence the editor. Only when closing the editor do you change into the other channel.

Procedure

Program manager	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.
Open	3.	Press the "Open" softkey.
		- OR -
INPUT		Press the <INPUT> key.
		- OR -
▶		Press the <Cursor right> 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.

9.3 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 MD11280 $MN_WPD_INI_MODE.

MD11280 $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>.

MD11280 $MN_WPD_INI_MODE=1:

All files of type 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

Program manager	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.
Select	3.	Press the "Select" softkey.
		The controller switches automatically into the "Machine" operating area.
		- OR -

NC Ex-ecute	If the selected program is already opened in the "Program" operating area,
	Press the "Execute NC" softkey.
◇ CYCLE START	Press the <CYCLE START> key.
	Execution of the workpiece is started.

Note

Only workpieces/programs that are located in the NCK memory, local drive or USB drive can be selected for execution.

9.4 Creating a directory/program/job list/program list

9.4.1 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 49 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

Program manager	1.	Select the "Program manager" operating area.
Local drive / **USB**	2.	Select your chosen storage medium, i.e. a local or USB drive.
New / **Directory**	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 opens.
OK	4.	Enter the desired directory name and press the "OK" softkey.

9.4.2 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

Workpiece directories

You have the option of nesting tool directories. You must note that the length of the call line is restricted. You will be informed if the maximum number of characters is reached when entering the workpiece name

---

Procedure

Program manager	1.	Select the "Program manager" operating area.
NC / **USB**	2.	Select the desired storage location and position the cursor on the folder, in which you would like to create a workpiece.
New	3.	Press the "New" softkey. The "New Workpiece" window appears.

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

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

The name can be a maximum of 24 characters long.

You can use any letters (except accented), 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.

See also

Storing any new file (Page 236)

9.4.3 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.

         Program names may be a maximum of 24 characters in length.

         You can use all letters (with the exception of special characters, language-specific special characters, Asian or Cyrillic characters), numbers and underscores (_). The program type is appropriately specified.

9.4.4   Storing any new file

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

Note

File extensions

In the NC memory, the extension must have 3 characters, and DIR or WPD are not permitted.

In the NC memory, 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 be a maximum of 24 characters long. You can use any letters (except accented), digits or the underscore symbol (_).
	5.	Press the "OK" softkey.

9.4.5 Creating a job list

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

Comment

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

Program manager	1.	Select the "Program manager" operating area.
NC **NC**	2.	Press the "NC" softkey, and in directory "Workpieces" place the cursor on the program for which you wish to create a job list.
New ▶	3.	Press the "New" and "Any" softkeys. The "Any New Program" window opens.
✓ **OK**	4.	Select entry "Job list JOB" from the "Type" selection field and enter a name and press the "OK" softkey.

9.4.6 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

Program manager	1.	Select the "Program manager" operating area.
> PLC **Prog.-list**	2.	Press the menu forward key and the "Program list" softkey. The "Prog.-list" window opens.

	3.	Place the cursor in the desired line (program number).
Select program	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.
OK	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.
Delete	6.	To remove a program from the list, place the cursor on the appropriate line and press the "Delete" softkey.
		- OR -
Delete all		To delete all programs from the program list, press the "Delete all" softkey.

9.5 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

Setup	1.	Select the "Start-up" operating area.
System data	2.	Press the "System data" softkey.
Copy	3.	Position the cursor on the file that you wish to store as a template and press the "Copy" softkey.
Paste	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.

9.6 Searching directories and files

You have the possibility of searching in the Program Manager for certain directories and files.

Note

Search with place holders

The following place holders simplify the search:
- "*": replaces any character string
- "?": replaces any character

Search strategy

The search is made in all of the selected directories and their subdirectories.

If the cursor is positioned on a file, then a search is made from the higher-level directory.

Note

Searching in opened directories

Open the closed directories for a successful search.

Procedure

[Program manager]	1.	Select the "Program Manager" operating area.
[>>] [Search]	2.	Select the storage location in which you wish to perform the search and then press the ">>" and "Search" softkeys. The "Find file" window appears.
	3.	Enter the desired search term in the "Text" field. Note: When searching for a file, enter the complete name with extension (e.g. DRILLING.MPF).
	4.	When required, activate the checkbox "Observe upper and lower case".
[OK]	5.	Press the "OK" softkey to start the search.
	6.	If a corresponding directory or file is found, then it is marked.

Continue search / **OK**	7.	Press the "Continue search" and "OK" softkeys if the directory or the file does not correspond to the required result.
		- OR -
Cancel		Press the "Cancel" softkey when you want to cancel the search.

9.7 Displaying the program in the Preview.

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

Procedure

Program manager	1.	Select the "Program manager" operating area.
	2.	Select a storage location and place the cursor on the relevant program.
▶▶ / **Preview window**	3.	Press the ">>" and "Preview window" softkeys. The "Preview: ..." window opens.
Preview window	4.	Press the "Preview window" softkey again to close the window.

9.8 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

Selected files

If you have selected individual files in a directory, then this selection is canceled when the directory is closed.

If the complete directory with all of the files included in it are selected, then this selection is kept when closing the directory.

Procedure

Program manager	1.	Select the "Program manager" operating area.
Mark	2.	Choose the desired storage location and position the cursor on the file or directory from which you would like your selection to start.
Mark	3.	Press the "Select" softkey.
Mark		The softkey is active.
Mark	4.	Select the required directories/programs with the cursor keys or mouse.
Mark	5.	Press the "Select" softkey again to deactivate the cursor keys.

Canceling a selection

By reselecting an element, the existing selection is canceled.

Selecting via keys

Key combination	Meaning
⬭ SELECT	Renders or expands a selection. You can only select individual elements.
⇧ SHIFT ▼ ▲	Renders a consecutive selection.
INSERT	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.

9.9 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

Program manager	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.
Copy	3.	Press the "Copy" softkey.
	4.	Select the directory in which you want to paste your copied directory/program.
Paste	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.

9.10 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

Program manager	1.	Select the "Program manager" operating area.
▶▶ Delete	2. 3.	Choose the desired storage location and position the cursor on the file or directory that you would like to delete. Press the ">>" and "Delete" softkeys. A prompt appears as to whether you really want to delete the file or directory.
OK	4.	Press the "OK" softkey to delete the program/directory.
Cancel		- OR -. Press the "Cancel" softkey to cancel the process.

9.11 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

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

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

For NC files, you have the option to set the access rights from keyswitch 0 to the current access level, to be set separately for each file.

If an access level is higher than the current access level, it cannot be changed.

For external files (e.g. on a local drive), the access rights are displayed to you only if settings have been executed for these files by the machine manufacturer. They cannot be changed via the "Properties" window.

Settings for the access rights to directories and files

Via a configuration file and MD 51050, access rights of the directories and file types of the NC and user memory (local drive) can be changed and pre-assigned.

References

A detailed description of the configuration can be found in the following documentation:

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

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.
... Properties		
	4.	Enter any necessary changes. **Note**: You can save changes via the user interface in the NC memory.
✓ OK	5.	Press the "OK" softkey to save the changes.

9.12 Set up drives

9.12.1 Overview

Up to eight connections to so-called logical drives (data carriers) can be configured. These drives can be accessed in the "Program manager" and "Startup" operating areas.

The following logical drives can be set up:

- USB interface
- CompactFlash card of the NCU, only for SINUMERIK Operate in the NCU
- Network drives
- Local hard disk of the PCU, only for SINUMERIK Operate on the PCU

Software option

In order to use the CompactFlash card as data carrier, you require the "Additional HMI user memory on CF card of the NCU" option (not for SINUMERIK Operate on PCU or PC).

Note

The USB interfaces of the NCU are not available for SINUMERIK Operate and therefore cannot be configured.

9.12.2 Setting up drives

The "Set Up Drives" window is available in the "Start-up" operating area for configuration.

File

The created configuration data is stored in the "logdrive.ini" file. This file is located in the /user/sinumerik/hmi/cfg directory.

General information

Entry		Meaning
Type	No drive	No drive defined.
	USB local	Access to the USB memory medium is only realized via the TCU to which it is connected. USB drives are automatically identified if the memory medium is inserted when SINUMERIK Operate powers-up.
	USB global	All of the TCUs in the plant network can access the USB memory medium. - USB global is not possible under Windows!
	NW Windows/Linux	Network drive
	Local drive	Local drive Hard disk or user memory on the CompactFlash card.
	FTP	Access to an external FTP server.
Connection	Front	USB interface that is located at the front of the operator panel.
	X203/X204	USB interface X203/X204 that is located at the rear of the operator panel.
	X204	For SIMATIC Thin Client the USB interface is X204.
Device		Names of the TCU to which the USB storage medium is connected, e.g. tcu1. The NCU must already know the TCU name.
Partition		Partition number on the USB memory medium, e.g. 1. If a USB hub is used, specify the USB port of the hub.
Port		Interface for the FTP connection. The default port is 21.
Disconnect		After a disconnect timeout, the FTP connection is disconnected. The timeout can be between 1 and 150 s. 10 s is the default setting.
Path		• Start direct directory of the data carrier that is connected via the local drive. • Network path to a directory that has been released in the network. This path must always start with "//", e.g. //Server01/share3.

Entry		Meaning
Access level		Assign access rights to the connections: From access level 7 (keyswitch position 0) to access level 1 (manufacturer).
		The particular assigned access level applies to all operating areas.
Softkey text		2 lines are available as labeling text for the softkey. %n is accepted as a line separator.
		If the first line is too long, then it is automatically separated into several lines. If a blank is present, then this is used as line separator.
Softkey icon	No icon	No icon is displayed on the softkey.
	sk_usb_front.png	The icon file name displayed on the softkey.
	sk_local_drive.png	
	sk_network_drive_ftp.png	
Text file	slpmdialog	File for softkey dependent on the language. If nothing is specified in the input fields, the text appears on the softkey as was specified in the input field "Softkey text".
Text context	SlPmDialog	
		If your own text files are saved, then the text ID, which is used to search for the text file, is specified in the "Softkey text" input field.
User name		Enter the user name and the corresponding password for which the directory is enabled on the server. The password is displayed in encoded form as string of "*" characters and is stored in the "logdrive.ini" file.
Password		

Error messages

Error message	Meaning
Error occurred when closing a connection.	An existing drive was not able to be deactivated.
Error occurred when establishing a connection.	Drive connection was not able to be established.
Error occurred while establishing a connection: Incorrect entry or no authorization.	Drive connection was not able to be established.
Incorrect data	The entered data is either incorrect or inconsistent.
Function not available	The function is not supported with the current software release.
Unknown error - error code:%1	Error was not able to be assigned.

Procedure

Setup	1.	Select the "Start-up" operating area.
HMI / **Log. drives**	2.	Press the "HMI" and "Log. drive" softkeys. The "Set Up Drives" window opens.
	3.	Select the data for the corresponding drive or enter the necessary data.
Activate drive	4.	Press the "Activate drive" softkey. The drive is activated. The operating system now checks the entered data and whether the connection is established. An OK message is output in the dialog line if an error is not identified. The drive can be accessed. - OR - If the operating system identifies an error, then you receive an error message.
OK		Press the "OK" softkey. You then return to the "Set Up Drives" window. Check and correct your entries and re-activate the drive.
OK		If you press the "OK" softkey, the modified data is accepted without any feedback message. You do not receive any message about a successful or unsuccessful connection.
Cancel		If you press the "Cancel" softkey, then all of the data that has not been activated is rejected.

9.13 Viewing PDF documents

You have the option of displaying HTML documents, as well as PDFs, on all drives of the program manager via the data tree of the system data.

Note

A preview of the documents is only possible for PDFs.

Procedure

NC NC **USB** USB	1.	In the "Program manager" operating area, select the desired storage medium.
		- OR -
System data		Select the desired storage location in the "Commissioning" operating area in the data tree of the "System data".
Open	2.	Position the cursor on the PDF or the HTML file that you want to display, and press the "Open" softkey. The selected file is displayed on the screen. The storage path of the document is displayed in the status bar. The current page as well as the total number of pages of the document are displayed.
Zoom + **Zoom -**	3.	Press the "Zoom +" or "Zoom -" softkey to enlarge of reduce the size of the display.
Search	4.	Press the "Search" softkey if you want to search for specific texts in the PDF.
View	5.	Press the "View" softkey to change the representation of the PDF. A new vertical softkey bar appears.
Zoom page width	6.	Press the "Zoom page width" softkey to display the document in full width on the screen.
		- OR -
Zoom page length		Press the "Zoom page height" softkey to display the document with full height on the screen.
		- OR -
		Press the "Rotate left" softkey to rotate the document through 90 degrees to the left.
		- OR -

	Press the "Rotate right" softkey to rotate the document through 90 degrees to the right.
	7. Press the "Back" softkey to return to the previous window.
	8. Press the "Close" softkey to exit the PDF display.

9.14 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 SD $SC42700 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 SD $SC42700 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 SD $SC42700 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.

● For programs that were generated on external storage media (Windows system) observe upper- and lower-case syntax.

Note

Maximum path length for EXTCALL

The path length must not exceed 112 characters. The path comprises the contents of the setting data (SD $SC42700) and the path data for EXTCALL call from the part program.

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 the USB front connection (USB-FlashDrive), if SD $SC 42700 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 the network drive, if SD $SC42700 "//Computer name/enabled drive" contains: EXTCALL "TEST.SPF"

● Use of the HMI user memory (local drive):

– On the local drive, you have created the directories part programs (mpf.dir), subprograms (spf.dir) and workpieces (wks.dir) with the respective workpiece directories (.wpd):

SD42700 is empty: EXTCALL "TEST.SPF"

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

– On the local drive, you have created your own directory (e.g. my.dir):

Specification of the complete path: e.g. EXTCALL "/card/user/sinumerik/data/prog/my.dir/TEST.SPF"

A search is performed for the specified file.

Note

Abbreviations for local drive, CompactFlash card and USB front connection

As abbreviation for the local drive, the CompactFlash card and the USB front connection you can use the abbreviation LOCAL_DRIVE:, CF_CARD: and USB: (e.g. EXTCALL "LOCAL_DRIVE:/spf.dir/TEST.SPF").

Alternatively, you can also use the abbreviations CF_Card and LOCAL_DRIVE.

Software options

To display the "Local drive" softkey, you require the "Additional HMI user memory on CF card of the NCU" option (not for SINUMERIK Operate on PCU50 / PC).

NOTICE
Possible interruption when executing 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.

Machine manufacturer

Processing EXTCALL calls can be enabled and disabled.

Please refer to the machine manufacturer's specifications.

9.15 Backing up data

9.15.1 Generating an archive in the Program Manager

You have the option of archiving individual files from the NC memory and the local drive.

Archive formats

You have the option of saving your archive in the binary and punched tape format.

Save target

The archive folder of the system data in the "Startup" operating area as well as USB and network drives are available as save target.

Procedure

Program manager	1.	Select the "Program Manager" operating area.
NC	2.	Select the storage location for the file/files to be archived.
Mark	3.	In the directories, select the required file from which you want to create 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.
▶▶ Archive	4.	Press the ">>" and "Archive" softkeys.
Generate archive	5.	Press the "Generate archive" softkey. The "Generate Archive: Select archiving" window opens.
Search OK	6.	Position the cursor to the required storage location, press the "Search" softkey, enter the required search term in the search dialog and press the "OK" softkey if you wish to search for a specific directory or subdirectory. **Note:** The place holders "*" (replaces any character string) and "?" (replaces any character) make it easier for you to perform a search. - OR -

Select the required storage location, press the "New directory" softkey, enter the required name in the "New directory" window and press the "OK" softkey to create a directory.

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

| 9. | Select the format (e.g. archive ARC (binary format)), enter the desired name and press the "OK" softkey. |
| | A message informs you if archiving was successful. |

9.15.2 Generating an archive via the system data

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

Archive formats

You have the option of saving your archive in the binary and punched tape format.

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.

Precondition

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

Storage locations

● CompactFlash card under
/**user**/sinumerik/data/archive, or
/**oem**/sinumerik/data/archive

● All configured logical drives (USB, network drives)

Software option

In order to save archives on the CompactFlash card in the user area you require the option "Additional HMI user memory on CF card of NCU".

> **NOTICE**
>
> **Possible data loss when using USB flash drives**
>
> USB-FlashDrives are not suitable as persistent memory media.

Procedure

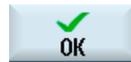|		
Setup	1.	Select the "Startup" operating area.
System data	2.	Press the "System data" softkey. The data tree opens.
	3.	In the data tree, select the required files from which you want to generate an archive. - OR -
Mark		If you want to back up several files or directories, press the "Select" softkey 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.
Preview window	5.	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.
Properties **OK**	6.	Press the "Properties" softkey. Information about the selected file is displayed in a small window. Press the "OK" softkey to close the window.
Search **OK**	7.	Press the "Search" softkey. Enter the required search term in the search dialog and press the "OK" softkey if you wish to search for a specific directory or subdirectory. **Note**: The place holders "*" (replaces any character string) and "?" (replaces any character) make it easier for you to perform a search.
Archive **Generate archive**	8.	Press the "Archive" and "Generate archive" softkeys. The "Generate Archive: Select Storage Location" window opens. The "Archive" folder with the subfolders "User" and "Manufacturer" as well as the storage media (e.g. USB) are displayed.

New directory	9.	Select the required location for archiving and press the "New directory" softkey to create a suitable subdirectory.
		The "New Directory" window opens.
✓ **OK**	10.	Enter the required name and press the "OK" softkey.
		The directory is created below the selected folder.
✓ **OK**	11.	Press the "OK" softkey.
		The "Generate Archive: Name" window opens.
✓ **OK**	12.	Select the format (e.g. archive ARC (binary format)), enter the desired name and press the "OK" softkey to archive the file/files.
		A message informs you if archiving was successful.
✓ **OK**	13.	Press the "OK" softkey to confirm the message and end the archiving operation.
		An archive file in .arc format is created in the selected directory.

9.15.3 Reading in an archive in the Program Manager

In the "Program Manager" operating area, you have the option of reading in archives from the archive folder of the system data as well as from configured USB and network drives.

Software option

In order to read-in user archives in the "Program Manager" operating area, you require the option "Additional HMI memory on CF Card of NCU" (not for SINUMERIK Operate on PCU50 / PC).

Procedure

Program manager	1.	Select the "Program Manager" operating area.
Archive **Read-in archive**	2.	Press the "Archive" and "Read in archive" softkeys.
		The "Read in archive: Select archive" window opens.

	3.	Select the archive storage location and position the cursor on the required archive.
		Note: When the option is not set, the folder for user archives is only displayed if the folder contains at least one archive.
		- OR -
Search		Press the "Search" softkey and in the search dialog, enter the name of the archive file with file extension (*.arc) if you wish to search for a specific archive and press the "OK" softkey.
OK		
OK	4.	Press the "OK" or "Overwrite all" softkey to overwrite existing files.
...		
Overwrite all		
		- OR -
No overwriting		Press the "Do not overwrite" softkey if you do not want to overwrite already existing files.
		- OR -
Skip		Press the "Skip" softkey if the read-in 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.
		You will then obtain a "Read error log for archive" in which the skipped or overwritten files are listed.
Cancel	5.	Press the "Cancel" softkey to cancel the read-in process.

See also

Searching directories and files (Page 242)

9.15.4 Read in archive from system data

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

Procedure

Setup	1.	Select the "Start-up" operating area.
System data	2.	Press the "System data" softkey.
	3.	In the data tree below the "Archive" directory, in the "User" folder, select the file that you wish to read in.
Read in	4.	Press the "Read in" softkey.
OK	5.	Press the "OK" or "Overwrite all" softkey to overwrite existing files.
...		
Overwrite all		
		- OR -
No overwriting		Press the "Do not overwrite" softkey if you do not want to overwrite already existing files.
		- OR -
Skip		Press the "Skip" softkey if the read-in 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.
		You will then obtain a "Read error log for archive" in which the skipped or overwritten files are listed.
Cancel	6.	Press the "Cancel" softkey to cancel the read-in process.

9.16   Setup data

Apart from programs, you can also save tool data and zero point settings.

You can use this option, for example, to backup the required tools and zero point data for a specific G code program. If you want to execute this program at a later point in time, you will then have quick access to the relevant settings.

Even tool data that you have measured on an external tool setting station can be copied easily into the tool management system using this option.

Note

Backing up setup data from part programs

Setup data from part programs can only be backed up if they have been saved in the "Workpieces" directory.

For part programs, which are located in the "Part programs" directory, "Save setup data" is not listed.

Backing up data

Data	
Tool data	• No • Complete tool list
Magazine assignment	• Yes • No
Zero points	• No The selection box "Basis zero point" is hidden • All
Basic zero points	• No • Yes
Directory	The directory is displayed, in which the selected program is located.
File name	Here you have the option of changing the suggested file names.

Note

Magazine assignment

You can only read out the magazine assignments if your system provides support for loading and unloading tool data to and from the magazine.

Procedure

Program manager	1.	Select the "Program Manager" operating area.
NC	2.	Position the cursor on the program whose tool and zero point data you wish to back up.
...		
Local drive		
▶▶	3.	Press the ">>" and "Archive" softkeys.
Archive		
Save setup data	4.	Press the "Setup data" softkey. The "Backup setup data" window opens.
	5.	Select the data you want to back up.
	6.	When required, change the specified name of the originally selected program here in the "File name" field.
OK	7.	Press the "OK" softkey. The setup data will be set up in the same directory in which the selected program is stored. The file is automatically saved as INI file.

Note

Program selection

If a main program as well as an INI file with the same name are in a directory, when selecting the main program, initially, the INI file is automatically started. In this way, unwanted tool data can be changed.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

9.16.1 Reading-in set-up data

When reading-in, you can select which of the backed-up data you wish to read-in:

- Tool data
- Magazine assignment
- Zero points
- Basic zero point

Tool data

Depending on which data you have selected, the system behaves as follows:

- Complete tool list

    First, all tool management data are deleted and then the saved data are imported.

- All tool data used in the program

    If at least one of the tools to be read in already exists in the tool management system, you can choose between the following options.

Overwrite all	Select the "Replace all" softkey to import all tool data. Any existing tools will now be overwritten without a warning prompt.
	- OR -
No overwriting	Press the "Do not overwrite" softkey if existing tools must not be overwritten. Already existing tools are skipped, without you receiving any queries.
	- OR -
Skip	Press the "Skip" softkey if already existing tools are not to be overwritten. For an already existing tool, you receive a query.

Selecting loading point

For a magazine, if more than one loading point was set-up, using the "Select loading point" softkey, you have the option of opening a window in which you can assign a loading point to a magazine.

Procedure

Program manager	1.	Select the "Program Manager" operating area.
NC Local drive	2.	Position the cursor on the file with the backed-up tool and zero point data (*.INI) that you wish to re-import.
▶	3.	Press the <Cursor right> key - OR - Double-click the file. The "Read-in setup data" window opens.
SELECT	4.	Select the data (e.g. magazine assignment) that you wish to read-in.
OK	5.	Press the "OK" softkey.

9.17 RS-232-C

9.17.1 Reading-in and reading-out archives

Availability of the RS-232-C serial interface

You have the option of reading-out and reading-in archives in the "Program manager" operating area as well as in the "Start-up" operating area via the RS-232-C serial interface.

- SINUMERIK Operate on the NCU:

  The softkeys for the RS-232-C are available as soon as an option module is connected and the slot is occupied.

- SINUMERIK Operate on the PCU:

  The softkeys for the RS-232-C are always available.

Reading-out archives

The files to be sent (directories or individual files) are zipped in an archive (*.ARC). If you send an archive (*.arc), this is sent directly without being additionally zipped. If you have selected an archive (*.arc) together with an additional file (e.g. directory), then these are zipped into a new archive and are then sent.

Reading-in archives

Only archives can be read-in via the RS-232-C. These are transferred and then subsequently unzipped.

Note

Start-up archive

If you read in a start-up archive via the RS-232-C, then this is immediately activated.

Externally processing the punched tape format

If you wish to externally process an archive, then generate this in the punched tape format. Using the SinuCom commissioning and service tool SinuCom ARC, you can process the archive in the binary format and in the commissioning archive.

Procedure

Program manager **NC** NC ... **Local drive**	1.	Select the "Program manager" operating area, and press the "NC" or "Local drive" softkey.
		- OR -
Setup **System data**		Select the "Start-up" operating area and press the "System data" softkey.

Reading-out archives

	2.	Select the directories or the files that you wish to send via RS-232-C.
▶▶ **Archive**	3.	Press the ">>" and "Archive" softkeys.
Send RS-232-C	4.	Press the "RS-232-C send" softkey.
		- OR -

Reading in an archive

Receive RS-232-C	Press the "RS-232-C receive" softkey if you wish to read-in files via RS-232-C.

V24 setting	Meaning
Protocol	The following protocols are supported for transfer via the V24 interface: • RTS/CTS (default setting) • Xon/Xoff
Transfer	It is also possible to use a secure protocol for data transfer (ZMODEM protocol). • Normal (default setting) • secure For the selected interface, secure data transfer is set in conjunction with handshake RTS/CTS.
Baud rate	Transfer rate: Transfer rates of up to 115 kbaud can be selected. The baud rate that can be used depends on the connected device, the cable length and the general electrical conditions. • 110 • .... • 19200 (default) • ... • 115200
Archive format	• Punched tape format (default setting) • Binary format (PC format)
V24 settings (details)	
Interface	• COM1
Parity	Parity bits are used for error detection: The parity bits are added to the coded characters to make the number of positions set to "1" an uneven number (uneven parity) or to an even number (even parity). • None (default setting) • Odd • Even
Stop bits	Number of stop bits for asynchronous data transfer. • 1 (default) • 2
Data bits	Number of data bits for asynchronous data transfer. • 5 bits • ... • -8 bits (default setting)
XON (hex)	Only for punched tape format
XOFF (hex)	Only for punched tape format

V24 setting	Meaning
End of data transfer (hex)	Only for punched tape format
	Stop with end of data transfer character
	The default setting for the end of data transfer character is (HEX) 1A
Time monitoring (sec)	Time monitoring
	For data transfer problems or at the end of data transfer (without end of data transfer character) data transfer is interrupted after the specified number of seconds.
	The time monitoring is controlled by a time generator (clock) that is started with the first character and is reset with each transferred character. The time monitoring can be set (seconds).

Procedure

Program manager	1.	Select the "Program Manager" operating area.
NC Local drive	2.	Press the "NC" or "Local drive" softkey.
▶▶ Archive	3.	Press the ">>" and "Archive" softkeys.
RS-232-C settings	4.	Select the "V24 settings" softkey. The "Interface: V24" window is opened.
	5.	The interface settings are displayed.
Details	6.	Press the "Details" softkey if you wish to view and process additional settings for the interface.

Teaching in a program

<div style="text-align: right; font-size: large;">10</div>

10.1 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.

10.2 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 using the cursor key and input key.

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.

10.3 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.

10.3.1 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

10.4 Teach-in via Windows

10.4.1 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.

Precondition

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

Procedure

Machine	1	Select the "Machine" operating area.
AUTO / **MDA**	2.	Press the <AUTO> or <MDA> key.
TEACH IN	3.	Press the <TEACH IN> key.
Teach prog.	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.
Rap. tra. G0 **Circ. end pos. CIP**	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.
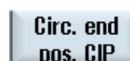
8. Press the "Accept" softkey.

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

    - OR -

    Press the "Cancel" softkey to cancel your input.

10.4.2 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.

10.4.3 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.

10.4.4 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.

10.4.5 Teach-in A spline

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".

Software option

You require the "Spline-Interpolation" option for A Spline interpolation.

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

M Machine	1.	Select the "Machine" operating area.
→| AUTO / □) MDA	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.

10.5 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 processed is selected.

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.	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.

10.6   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.

10.7 Deleting a block

You have the option of deleting a program block entirely.

Requirement

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

Procedure

[Machine]	1.	Select the "Machine" operating area.
[AUTO] [MDA]	2.	Press the <AUTO> or <MDA> key.
[TEACH IN]	3.	Press the <TEACH IN> key.
[Teach prog.]	4.	Press the "Teach prog." softkey.
	5.	Click the program block to be deleted.
[>>] [Delete block]	6.	Press the ">>" and "Delete block" softkeys. The program block on which the cursor is positioned is deleted.

10.8   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.

HT 8   11

11.1   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:

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

Customer keys

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

Machine manufacturer

Please refer to the machine manufacturer's specifications.

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 Section "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 refer to the machine manufacturer's specifications.

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:

Operator Components and Networking Manual; SINUMERIK 840D sl/840Di sl

See also

Channel switchover (Page 57)

11.2 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.

11.3 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.
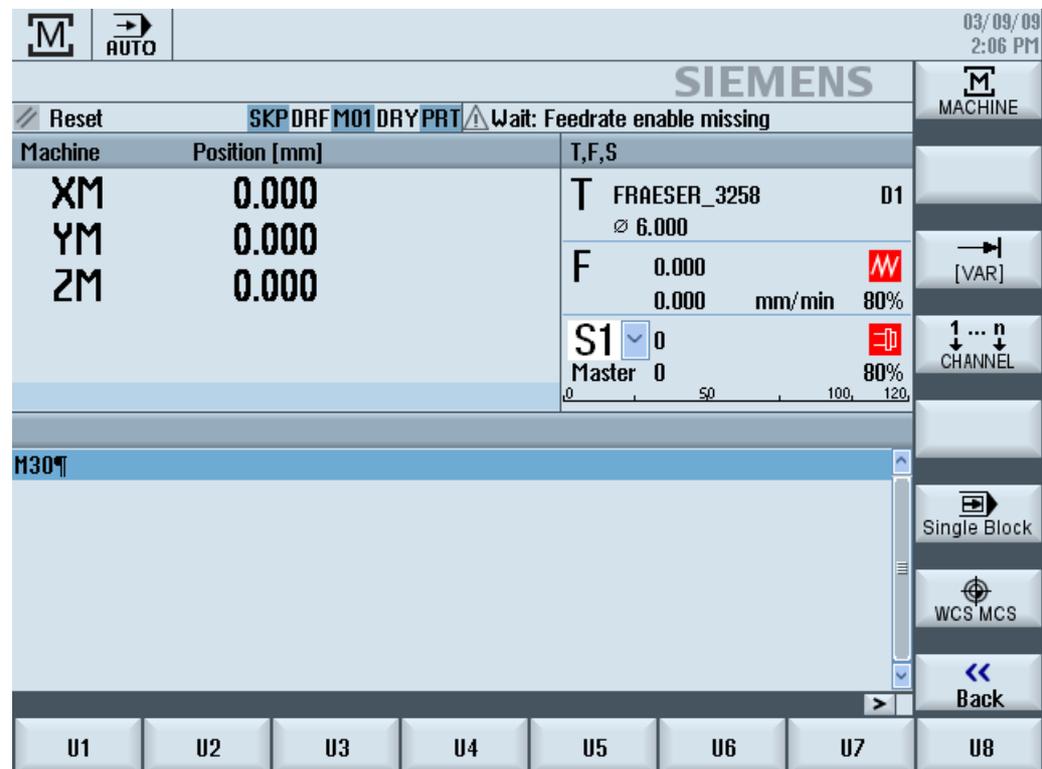
See chapter "Controls on the machine control panel" for a description of the individual keys.

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.

11.4 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
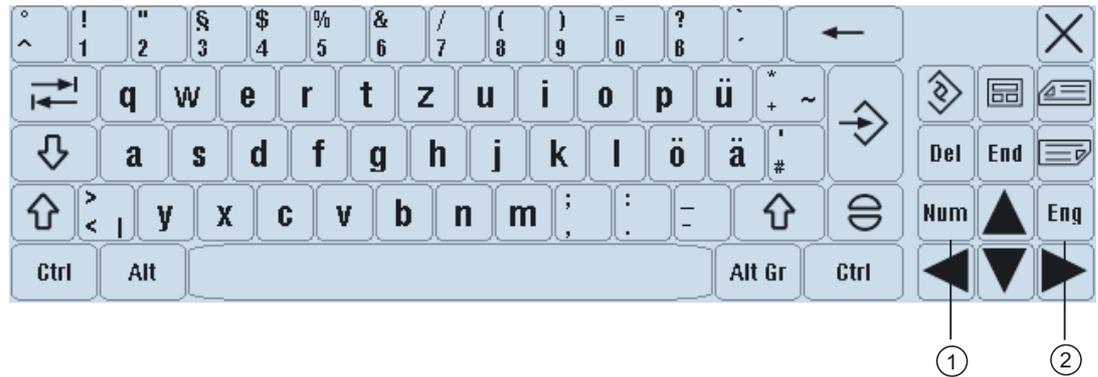
1   Num:

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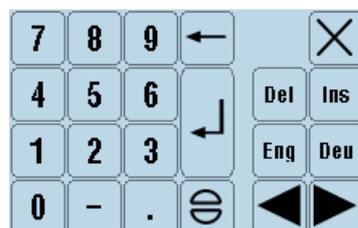
Use the "Deu" or "Eng" keys to return to the full keyboard with the English keyboard assignment or the keyboard assignment of the current language setting.
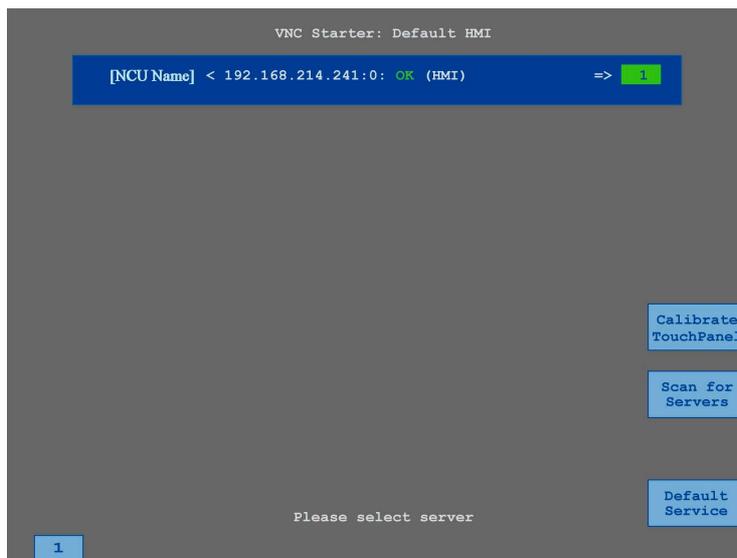
11.5 Calibrating the touch panel

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.

Ctrl-Energy

<div style="text-align: right; font-size: 2em;">12</div>

12.1 Overview

The "Ctrl-Energy" function provides you with the following options to improve the energy utilization of your machine.

Ctrl-E Analysis: Measuring and evaluating the energy consumption

Acquiring the actual energy consumption is the first step to achieving better energy efficiency. The energy consumption is measured and displayed at the controller using the SENTRON PAC multi-function device.

Depending on the configuration and connection of the SENTRON PAC, you have the possibility of either measuring the power of the whole machine or only a specific load.

Independent of this, the power is determined directly from the drives and displayed.

Ctrl-E Profiles: Control of energy saving states of the machine

To optimize the energy consumption, you have the option of defining energy saving profiles and saving them. For instance, your machine has a basic and a more sophisticated energy-saving mode - or under certain conditions, automatically switches itself off.

These defined energy states are saved as profiles. At the user interface, you have the possibility of activating these energy-saving profiles (e.g. the so-called tea break key).

Note
Ctrl-E Deactivating profiles

Disable Ctrl-E profiles before a series startup in order to prevent the NCU unintentionally shutting down.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

See also

Using the energy-saving profile (Page 299)

12.2 Displaying energy consumption

You have the option of displaying the actual energy consumption values of the individual axes and the total energy consumption.

| Display	Meaning
Manual	• No input • Entering and displaying a fixed value • Displaying a value from a data source
Sentron PAC	Displaying an actual value measured by a SENTRON PAC device.
Total, drives	Display of the total of all measured drive values.
Total, machine	Addition of the values of all axes as well as a fixed value and Sentron PAC.

The display in the table depends on the configuration.

References

Information on the configuration is provided in the following reference:

System Manual "Ctrl-Energy", SINUMERIK 840D sl / 828D

Procedure
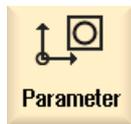
1. Select the "Parameter" operating area and press the "Ctrl-Energy" softkey.

   - OR -

   + Press the <Ctrl> + <E> keys.

2. Press the "Energy analysis" softkey.
   The "SINUMERIK Ctrl-Energy Analysis" window opens.
   You obtain a display of the actual active power for all of the axes.

12.3 Measuring and saving the energy consumption

For the currently selected axes, you have the option of measuring and recording the energy consumption.

Measurement of the energy consumption by part programs

The energy consumption of part programs can be measured. The measurement should take single drives into account.

They specify in which channel the start and stop of the part program should be initiated and which number of repetitions are to be measured.

Save measurements

Save the measured consumption values so that you can subsequently compare the data.

Note

Up to three data records are displayed. The oldest data record is automatically overwritten if there are more than three measurements.

Measurement duration

The measurement duration is limited. When the maximum measurement duration is reached, the measurement is terminated and a corresponding message is output in the dialog line.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

	1.	The "Ctrl-Energy Analysis" window opens.
Start measuremen	2.	Press the "Start measurement" softkey. The "Setting Measurement: Select Device" window opens.
OK	3.	Select the desired device in the list, possibly activate the "Measure part program" checkbox, enter the number of repetitions, possibly select the desired channel, and press the "OK" softkey. The trace is started.
Stop measuremen	4.	Press the "Stop measurement" softkey. The measurement is ended.

Display meas. curves	5.	Press the "Graphic" softkey, to track the measurement characteristic.
Details	6.	If you want to track the consumption values, press the "Details" softkey.
Save measuremen	7.	Press the "Save measurement" softkey to save the consumption values of the actual measurement.

The selection of the axis to be measured depends on the configuration.

References

Information on the configuration is provided in the following reference:

System Manual "Ctrl-Energy", SINUMERIK 840D sl / 828D

12.4 Displaying measured curves

Display	Meaning
Start of the measurement	Shows the time at which the measurement was started by the pressing the "Start measurement" softkey.
Duration of the measurement [s]	Shows the measuring duration in seconds until the "Stop measurement" softkey is pressed.
Device	Displays the selected measured component. • Manual • Total, drives • Total, machine
Supplied energy [kWh]	Shows the supplied energy of the selected measured component in kilowatts per hour.
Regenerated energy [kWh]	Shows the regenerated energy of the selected measured component in kilowatts per hour.
Energy totals [kWh]	Shows the total of all measured drive values or the total of all axes as well as fixed value and Sentron PAC.

Procedure

1. The "SINUMERIK Ctrl-Energy Analysis" window opens and you have already performed and saved measurements.

2. Press the "Graphic" and "Saved measurements" softkeys.

   The measurement curves are displayed in the "SINUMERIK Ctrl-Energy Analysis" window.

3. Press the "Saved measurements" softkey again if you only want to see the actual measurement.

4. Press the "Details" softkey for the exact measurement data and consumption values of the last three saved measurements and possibly a current measurement.

12.5   Long-term measurement of the energy consumption

The long-term measurement of energy consumption is performed in the PLC and saved. The values from times in which the HMI is not active are also recorded.

Measured values

The infeed and regenerative power values as well as the sum of the power are displayed for the following periods:

- Current and previous day
- Current and previous month
- Current and previous year

Requirement

SENTRON PAC is connected.

Procedure

	1.	The "Ctrl-Energy Analysis" window opens.
Long-term measuremnt	2.	Press the "Long-term measurement" softkey. The "SINUMERIK Ctrl-Energy Analysis Long-term Measurement" window opens. The results of the long-term measurement are displayed.
« Back	4.	Press the "Back" softkey to terminate the long-term measurement.

12.6 Using the energy-saving profile

In the "SINUMERIK Ctrl-Energy Energy-Saving Profile" window, you can display all of the defined energy-saving profiles. Here, directly activate the required energy-saving profile - or inhibit or release profiles.

SINUMERIK Ctrl-Energy energy-saving profiles

Display	Meaning
Energy-saving profile	All energy-saving profiles are listed.
active in [min]	The remaining time up until the defined profile is reached is displayed.

Note

Disable all energy-saving profiles

For example, in order not to disturb the machine while measurements are being made, select "Disable all".

Once the pre-warning time of a profile has been reached, an alarm window that shows the remaining time is displayed. Once the energy-saving mode has been reached, then an appropriate message is displayed in the alarm line.

Energy-saving profiles

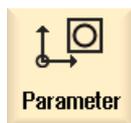| Energy-saving profile	Meaning
Simple energy-saving mode (machine standby)	Machine units that are not required are either throttled or shut down.
	When required, the machine is immediately ready to operate again.
Full energy-saving mode (NC standby)	Machine units that are not required are either throttled or shut down.
	Wait times are incurred at the transition into the ready to operate state.
Maximum energy-saving mode (auto shut-off)	The machine is completely switched-off.
	Longer wait times are incurred at the transition into the ready to operate state.

Machine manufacturer

The selection and function of the displayed energy-saving profiles can differ.

Please refer to the machine manufacturer's specifications.

References

Information on the configuration of the energy-saving profiles is provided in the following reference:

System Manual "Ctrl-Energy", SINUMERIK 840D sl / 828D

Procedure

Parameter	1.	Select the "Parameter" operating area.
`>` **Ctrl Energy**	2.	Press the menu forward key and then the "Ctrl-Energy" softkey. - OR - Press the <CTRL> + <E> keys.
CTRL +		
E		
Machine standby		The "SINUMERIK Ctrl-Energy Energy-Saving Profiles" window opens.
Activate immediately	3.	Position the cursor on the required energy-saving profile and press the "Activate immediately" softkey if you wish to directly activate this state.
Disable state	4.	Position the cursor on the required energy-saving profile and press the "Disable profile" softkey if you wish to disable this state. The profile is disabled. The energy-saving profile cannot be active, i.e. it is grayed out and displayed without time details. The labeling of the "Disable profile" softkey changes to "Enable profile".
Enable state		Press the "Enable profile" softkey in order to withdraw the energy-saving profile disable.
Disable all	5.	Press the "Disable all" softkey in order to disable all states. All the profiles are disabled and can be non-active. The labeling of the "Disable all" softkey changes to "Enable all".
Enable all	6.	Press the "Enable all" softkey to withdraw the disable for all profiles.

Easy Message (828D only)

<div style="text-align: right; font-size: large;">13</div>

13.1 Overview

Easy Message enables you to be informed about certain machine states by means of SMS messages via a connected modem:

- For example, you would like to be informed about emergency stop states
- You would like to know when a batch has been completed

Control commands

- HMI commands are used to activate or deactivate a user.

  Syntax: [User ID] deactivate, [User ID] activate

- A special area is reserved in the PLC to which you can send commands in the form of PLC bytes using SMS commands.

  Syntax: [User ID] PLC DataByte

  The user ID is optional and required only if a corresponding ID has been specified in the user profile. The string PLC indicates that a PLC byte is to be written. It is followed by the data byte to be written in the following format: Base#Value. Base can take the values 2, 10 and 16 and defines the number base. Then follows the separator # and the value of the byte. Only positive values are allowed to be sent.

  Examples:

  2#11101101

  10#34

  16#AF

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

Active users

In order to receive an SMS message for certain events, you must be activated as user.

User logon

As registered user, you can log on via SMS to inquire about messages.

Action log

You can obtain precise information about incoming and outgoing messages via SMS logs.

References

Information on the GSM modem can be found in the

PPU SINUMERIK 828D Manual

Calling the SMS Messenger

	1.	Select the "Diagnostics" operating area.
	2.	Press the "Easy Msg." softkey.

13.2 Activating Easy Message

To commission the connection to the modem for the SMS Messenger, activate the SIM card at the initial start-up.

Requirement

The modem is connected and activated.

Machine manufacturer

The modem is activated via the machine data 51233 $MSN_ENABLE_GSM_MODEM.

Please refer to the machine manufacturer's specifications.

Procedure

Activating the SIM card

1. Press the "Easy Msg." softkey.
   The "SMS Messenger" window appears.
   "Status" shows that the SIM card has not been activated with a PIN.

2. Enter the PIN, repeat the PIN and press the "OK" softkey.

3. If you made an incorrect entry several times, enter the PUK code in the "PUK Input" window and press the "OK" softkey to activate the PUK code.
   The "PIN input" window is opened and you can enter the PIN number as usual.

Activating a new SIM card

1. Press the "Easy Msg." softkey.
   The "SMS Messenger" window appears.
   "Status" shows that the connection to the modem has been activated.

2. Press the "Settings" softkey.

3. Press the "Delete PIN" softkey to delete the stored PIN.
   Enter the new PIN in the "PIN Input" window at the next power up.

13.3 Creating/editing a user profile

User identification

Display	Meaning
User name	Name of the user to be created or logged on.
Telephone number	Telephone number of the user to which the messages are to be sent. The telephone number must include the country code in order that control commands can identify the sender (e.g. +491729999999)
User ID	The user ID has 5 digits (e.g. 12345) • It is used to activate and deactivate the user via SMS. (e.g. "12345 activate") • The ID is used to additionally verify the incoming and outgoing messages and to activate the control commands.

Events that can be selected

You must set-up the events for which you receive notification.

Requirement

The modem is connected.

Procedure

Creating a new user

1. Press the "User profiles" softkey.
   The "User Profiles" window appears.

2. Press the "New" softkey.

3. Enter the name and telephone number of the user.

4. If required, enter the ID number of the user.

5. In the area "send SMS for the following events" area, activate the appropriate checkbox and when required, enter the desired value (e.g. the unit quantity, which when it is reached, a notification should be sent).
   - OR -
   Press the "Default" softkey.
   The appropriate window is opened and displays the default values.

Send test SMS	6.	Press the "Send test message" softkey. An SMS message with predefined text is sent to the specified telephone number.

Editing user data and events

Edit	1.	Select the user whose data you want to edit and press the "Edit" softkey. The input fields can be edited.
	2.	Enter new data and activate the desired settings. - OR -
Standard		Press the "Default" softkey to accept the default values.

13.4 Setting-up events

In the "Send SMS for the following events" area, select the events using the check box, which when they occur, an SMS is sent to the user.

- Programmed messages from the part program (MSG)

  In the part program, program an MSG command via which you receive an SMS.

  Example: MSG ("SMS: An SMS from a part program")

- Select the following events using the <SELECT> key

  – The workpiece counter reaches the following value

    An SMS is sent if the workpiece counter reaches the set value.

  – The following program progress is reached (percent)

    An SMS is sent if, when executing a part program, the set progress is reached.

  – Actual NC program reaches runtime (minutes)

    An SMS is sent after the set runtime has been reached when executing an NC program.

  – Tool usage time reaches the following value (minutes)

    An SMS is sent if the usage time of the tool reaches the set time when executing a part program (derived from $AC_CUTTING_TIME).

- Messages/alarms from the Tool Manager

  An SMS is sent if messages or alarms are output to the Tool Manager.

- Measuring cycle messages for tools

  An SMS is sent if measuring cycle messages are output that involve tools.

- Measuring cycle messages for workpieces

  An SMS is sent if measuring cycle messages are output that involve workpieces.

- Sinumerik messages/alarms (error when executing)

  An SMS is sent if NCK alarms or messages are output that cause the machine to come to a standstill.

- Machine faults

  An SMS is sent if PLC alarms or messages are output that cause the machine to come to a standstill (i.e. PLC alarms with Emergency Off response).

- Maintenance intervals

  An SMS is sent if the service planner registers pending maintenance work.

- Additional alarm numbers:

  Here, specify additional alarms where you should be notified if they occur.

  You can enter individual alarms, several alarms or alarm number ranges.

  Examples:

  1234,400

  1000-2000

  100,200-300

Requirement

- The user profile window is opened.
- You selected the event "Measuring cycle messages for tools", "Measuring cycle messages for workpieces", "Sinumerik messages/alarms (errors when executing)", "Machine faults" or "Maintenance intervals".

Editing events

Details	1.	Activate the required check box and press the "Details" softkey. The appropriate window opens (e.g. "Measuring cycle messages for workpieces") and shows a list of the defined alarm numbers.
Delete	2.	Select the corresponding entry and press the "Delete" softkey to remove the alarm number from the list.
		- OR -
New		Then press the "New" softkey if you wish to create a new entry. The "Create new entry" window opens.
OK		Enter the data and press the "OK" softkey to add the entry to the list.
Save		Press the "Save" softkey to save the settings for the result.
Standard	3.	Press the "Standard" softkey to return to the standard settings for the events.

13.5 Logging an active user on and off

Only active users receive an SMS message for the specified events.

You can activate users, already created for Easy Message, with certain control commands via the user interface or via SMS.

Easy Msg. active	The connection has been established to the modem.

Procedure

User profiles	1.	Press the "User profiles" softkey.
User active	2.	Select the desired user in the User name field and press the "User active" softkey
User active		**Note** Repeat step 2 to activate further users. - OR - Send an SMS with the User ID and the "activate" text to the control (e.g. "12345 activate")
User active		If the telephone number and the user ID match the stored data, the user profile is activated. You receive a message of success or failure per SMS.
User active	3.	Press the "User active" softkey to log off an activated user. - OR - Send an SMS with the "deactivate" text (e.g. "12345 deactivate") to log off from the Messenger. An SMS message is not sent to the deactivated user for the events specified in the user profile.

13.6 Displaying SMS logs

The SMS data traffic is recorded in the "SMS Log" window. In this way, it is possible to see the chronological sequence of activates when a fault occurs.

Symbols	Description
⬇	Incoming SMS message for the Messenger.
⬇	Message that has reached the Messenger, but which has not been processed (e.g. incorrect user ID or unknown account).
⬆	SMS message sent to a user.
⬆	Message that has not reached the user because of an error.

Requirement

Easy Msg. active   The connection has been established to the modem.

Procedure

SMS protocol   1. Press the "SMS log" softkey.

Incoming

Outgoing   The "SMS Log" window appears.

All the messages that have been sent or received by the Messenger are listed.

Note

Press the "Incoming" or "Outgoing" softkey to restrict the list.

13.7 Making settings for Easy Message

You can change the following Messenger configuration in the "Settings" window:

- Name of the controller that is part of an SMS message
- Number of sent messages
  - The SMS counter provides information on all sent messages.
  - Limit the number of sent messages in order to receive an overview of the costs through SMS messages, for example.

Setting the SMS counter to zero

When a set limit is reached, no further SMS messages are sent.

Press the "Reset SMS counter" softkey to reset the counter to zero.

Requirement

The connection has been established to the modem.

Procedure

	1.	Press the "Settings" softkey.
	2.	Enter an arbitrary name for the controller in the "Machine name" field.
	3.	If you want to limit the number of sent SMS messages then select the "Specify limit for SMS counter" entry and enter the desired number.

When the maximum number of messages is reached, you obtain a corresponding error message.

Note

Check the SMS log to see the exact time when the limit was reached.

- OR -

| | 3. | Press the "Default" softkey. |

If you have freely selected a machine name, this is replaced by a default name (e.g. 828D).

Easy Extend (828D only)

<div style="text-align: right; font-size: 3em;">14</div>

14.1 Overview

Easy Extend enables machines to be retrofitted with additional units, which are controlled by the PLC or that require additional NC axes (such as bar loaders, swiveling tables or milling heads), at a later point in time. These additional devices are easily commissioned, activated, deactivated or tested with Easy Extend.

The communication between the operator component and the PLC is performed via a PLC user program. The sequences to be executed for the installation, activation, deactivation and testing of a device are stored in a statement script.

Available devices and device states are displayed in a list. The view of the available devices can be controlled for users according to their access rights.

The subsequent chapters are selected for example only and are not available in every statement list.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Up to 64 devices can be managed.

References

SINUMERIK 828D Turning and Milling Commissioning Manual
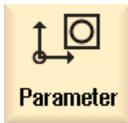
14.2 Enabling a device

The available device options are protected by a password.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Procedure

Parameter	1.	Select the "Parameter" operating area.
> **Easy-Extend**	2.	Press the menu forward key and then the "Easy Extend" softkey. A list of the connected devices is displayed.
Enable function	3.	Press the "Enable function" softkey. The "Enabling of the Devices Option" window is opened.
✓ **OK**	4.	Enter the option code and press the "OK" softkey. A tick appears in the appropriate checkbox in the "Function" column and the function is enabled.

14.3 Activating and deactivating a device

Status	Meaning
	Device activated
	System waiting for PLC checkback signal
	Device faulty
	Interface error in the communication module

Procedure

	1.	Easy Extend is opened.
	2	You can select the desired device in the list with the <Cursor up> and <Cursor down> keys.
	3.	Position the cursor on the device option for which the function has been unlocked and press the "Activate" softkey.
		The device is marked as activated and can now be used.
	4.	Select the desired activated device and press the "Deactivate" softkey to switch the device off again.

14.4 Commissioning Easy Extend

As a rule, the "Easy Extend" function has already been commissioned by the machine manufacturer. If a first commissioning has not been performed or if, for example, function tests are to be performed again (e.g. when retrofitting additional devices), this is possible at any time.

The "Start-up" softkey has been declared as Manufacturer data class (M).

Procedure
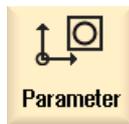
Parameter	1.	Select the "Parameter" operating area.
> / **Easy-Extend**	2.	Press the menu forward key and then the "Easy Extend" softkey.
Start-up	3.	Press the "Start-up" softkey. A new vertical softkey bar appears.
IBN StartUp	4.	Press the "Comm. start-up" softkey to start the commissioning. Before starting, a complete data backup is generated which you can then use in an emergency.
	5.	Press the "Cancel" softkey if you want to abort the commissioning prematurely.
Re-store	6.	Press the "Restore" softkey to load the original data.
Function test device	7.	Press the "Device function test" softkey to test the machine manufacturer's intended function.

Service Planner (828D only)

<div style="text-align:right; font-size:3em; font-weight:bold">15</div>

15.1 Performing and monitoring maintenance tasks

With the "Service Planner", maintenance tasks have been set up that have to be performed at certain intervals (e.g. top up oil, change coolant).

A list is displayed of all the maintenance tasks that have been set up together with the time remaining until the end of the specified maintenance interval.

The current status can be seen in the status display.

Messages and alarms indicate when a task has to be performed.

Acknowledging a maintenance task

Acknowledge the message when a maintenance task has been completed.

Note
Protection level

You require protection level 2 (service) to acknowledge completed maintenance tasks.

Service Planner

Display	Meaning	
Pos	Position of the maintenance task in the PLC interface.	
Maintenance task	Name of the maintenance task.	
Interval [h]	Maximum time until next servicing in hours.	
Remaining time [h]	Time until the interval expires in hours.	
Status		Display of the current status of a maintenance task.
	✔	The maintenance task has been started. The maintenance task is completed.
	✖	The maintenance task is deactivated.

Procedure

Diag-nostics	1.	Select the "Diagnostics" operating area.
> Mainten	2.	Press the menu forward key and then the "Service planner" softkey. The window with the list of all the maintenance tasks that have been set up appears.
	3.	Perform the maintenance task when the maintenance interval has nearly expired or when prompted to do so by alarms or a warning.
Maintenance completed	4.	After you have performed a pending maintenance task and the task is signaled as "Completed", position the cursor at the appropriate task and press the "Servicing performed" softkey. A message is displayed confirming the acknowledgment, and the maintenance interval is restarted.

Note

You can perform the maintenance tasks before the interval expires. The maintenance interval is restarted.

15.2 Set maintenance tasks

You can make the following changes in the list of maintenance tasks in the configuration mode:

- Set up a maximum of 32 maintenance tasks with interval, initial warning and number of warnings to be acknowledged
- Change the interval, time of the initial warning and the number of warnings to be output
- Delete a maintenance task
- Reset the times of the maintenance tasks

Acknowledging a maintenance task

You can acknowledge the maintenance tasks with the "Servicing performed" softkey.

Note

Protection level

You require protection level 1 (manufacturer) to set up and edit maintenance tasks.

Service Planner

Display	Meaning	
Pos	Position of the maintenance task in the PLC interface.	
Maintenance task	Name of the maintenance task.	
Interval [h]	Maximum time until next servicing in hours.	
1. warning [h]	Time in hours at which an initial warning is displayed.	
Number of warnings	Number of warnings that can be acknowledged by the operator before an alarm message is output for the last time.	
Remaining time [h]	Time until the interval expires in hours. The remaining time cannot be edited.	
Status	✔	Display of the current status of a maintenance task. The maintenance task has been started.
	🔴	The maintenance task is completed.
	✖	The maintenance task is deactivated, i.e. the time has been stopped.
	The status cannot be edited.	

Procedure

1. Select the "Diagnostics" operating area.

2. Press the menu forward key and then the "Service planner" softkey.

   The window opens and displays a list of all the tasks that have been set up.

   The values cannot be edited.

3. Press the "New maintenance task" softkey to set up a new maintenance task.

   A message informs you that a new maintenance task will be set up at the next free position. Enter the required data in the columns and press the "OK" softkey.

   - OR -

   Position the cursor on the desired maintenance task and press the "Change task" softkey to change the associated times.

   All columns except Remaining time and Status can be edited.

   - OR -

   Press the "Reset all" softkey to reset all the times.

   - OR -

   Position the cursor on the desired maintenance task and press the "Delete task" softkey to remove the maintenance task from the list.

Ladder Viewer and Ladder add-on (828D only) 16

16.1 PLC diagnostics

A PLC user program consists to a large degree of logical operations to implement safety functions and to support process sequences. These logical operations include the linking of various contacts and relays. These logic operations are displayed in a ladder diagram.

Ladder add-on tool

As a rule, the failure of a single contact or relay results in a failure of the whole system.

Using the Ladder add-on tool, you can perform a PLC diagnosis in order to find fault causes or program errors.

Editing of interrupt routines

You can edit the following interrupt programs:

- INT_100 - interrupt program, (is executed before the main program)
- INT_101 - interrupt program, (is executed after the main program)

Marshalling data

Using the Ladder add-on tool, you can "re-wire" inputs (via INT_100) or outputs (via INT_101) for service purposes.

Generating INT_100 / INT_101 block

If one or several INT_100 or INT_101 blocks are missing, they can be added via the vertical softkey bar. If these INT blocks exist in a project, they can be deleted via the vertical softkey bar. You also have the opportunity to change the networks of a program on the control as well as to save and load these changes.

Note

Saving the PLC project when changing the operating area

If you have created INT_100/INT_101 blocks or inserted, removed or edited networks in an INT block, you must save the project before you change from the PLC area into another operating area. Transfer the project into the PLC using the "Download to CPU" softkey. If this is not done, all of the changes will be lost and must be re-entered.
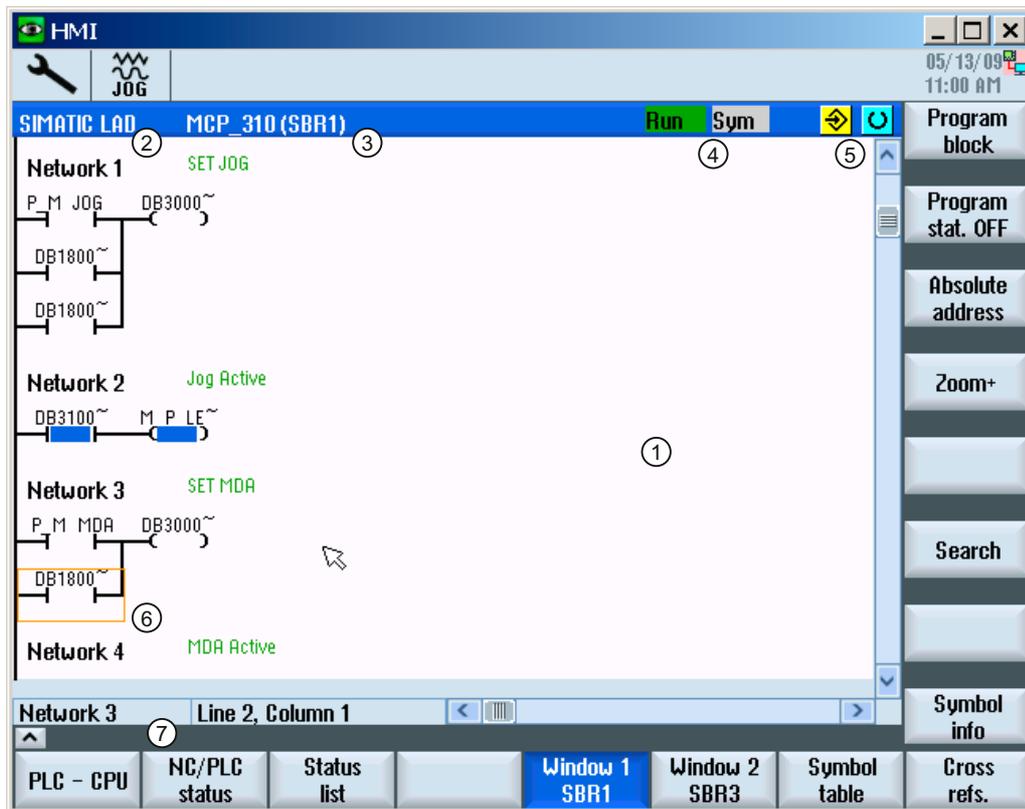

16.2 Structure of the user interface

Figure 16-1   Screen structure

Table 16- 1   Key to screen layout

Screen element	Display	Meaning
1	Application area	
2	Supported PLC program language	
3	Name of the active program block	
	Representation: Symbolic name (absolute name)	
4	Program status	
	Run Abs	
	Run	Program is running
	Stop	Program is stopped
	Status of the application area	
	Sym	Symbolic representation
	Abs	Absolute representation
5	Display of the active keys (<INPUT>, <SELECT>)	
	[icons]	

Screen element	Display	Meaning
6	Focus	
	Performs the tasks of the cursor	
7	Information line	
	Displays information, e.g. for searching	

16.3 Control options

In addition to the softkeys and the navigation keys, there are further shortcuts in this area.

Shortcuts

The cursor keys move the focus over the PLC user program. When the window borders are reached, scrolling is performed automatically.

Shortcuts	Action
[NEXT WINDOW] [CTRL] [◄]	To the first column of the row
[END] [CTRL] [►]	To the last column of the row
[PAGE UP]	Up a screen
[PAGE DOWN]	Down a screen
[◄] [►] [▲] [▼]	One field to the left, right, up or down
[CTRL] [NEXT WINDOW] -or-	To the first field of the first network

Shortcuts		Action
CTRL	▲	
CTRL	END	To the last field of the last network
-or-		
CTRL	▼	
CTRL	PAGE UP	Open the next program block in the same window
CTRL	PAGE DOWN	Open the previous program block in the same window
SELECT		The function of the Select key depends on the position of the input focus. • Table line: Displays the complete text line • Network title: Displays the network comment • Command: Displays all operands
INPUT		If the input focus is positioned on a command, all operands including the comments are displayed.

16.4   Displaying PLC properties

The following PLC properties can be displayed in the "SIMATIC LAD" window:

- Operating state
- Name of the PLC project
- PLC system version
- Cycle time
- Machining time of the PLC user program

Procedure

Setup	1.	Select the "Start-up" operating area.
PLC **PLC** **PLC – CPU**	2.	Press the "PLC" softkey. The ladder diagram display opens and displays the PLC information.
Reset pro. time	3.	Press the "Reset mach.time" softkey. The data of the machining time is reset.

16.5   Displaying and editing NC/PLC variables

The "NC/PLC Variables" window enables the monitoring and modification of NC system variables and PLC variables.

You receive the following list in which you enter the desired NC and PLC variables in order to display the actual values.

- Variable

  Address for NC/PLC variable.

  Faulty variables have a red background and # is displayed in the Value column.

- Comment

  Arbitrary comment on the variables.

  The column can be shown or hidden.

- Format

  Specification of the format in which the variable is to be displayed.

  The format can be permanently defined (e.g. floating-point).

- Value

  Display of the current value of the NC/PLC variable.

Procedure

PLC - CPU	1.	Ladder add-on tool is opened.
NC/PLC status	2.	Press the "NC/PLC variables" softkey. The "NC/PLC Variables" window appears.

16.6 Displaying and editing PLC signals

PLC signals are displayed and can be changed here in the "PLC status list" window.

The following lists are shown

Inputs (IB)

Bit memories (MB)

Outputs (QB)

Variables (VB)

Data (DB)

Setting the address

You can go directly to the desired PLC address to monitor the signals.

Editing

You can edit the data.

Procedure

PLC – CPU	1.	Ladder add-on tool is opened.
Status list	2.	Press the "Status list" softkey. The "Status List" window appears.
Set address	3.	Press the "Set address" softkey. The "Set Address" window appears.
OK	4.	Activate the desired address type (e.g. DB), enter the value and press the "OK" softkey. The cursor jumps to the specified address.
Change	5.	Press the "Change" softkey. The "RW" input field can be edited.
OK	6.	Enter the desired value and press the "OK" softkey.

16.7 Displaying information on the program blocks

You can display all the logic and graphic information of a program block.

Display program block

In the "Program block" list, select the program block that you want to display.

Logic information

The following logic information is displayed in a ladder diagram (LAD):

- Networks with program parts and current paths
- Electrical current flow through a number of logical operations

Further information

- Properties

  Name of the block, author, number of the subprogram, data class, date it was generated, date of the last change and comment.

- Local variable

  Name of the variable, variable type, data type and comment.

Access protection

| Protected | If a program block is password-protected. The ladder diagram display can be enabled using the "Protection" softkey. |

Displaying the program status

| Program stat. OFF | 1. | Press the "Program stat. OFF" softkey to hide the program status in the status display. |
| Program stat. ON | 2. | Press the "Program stat. ON" softkey to unhide the program status in the status display. |

Display progress status

If your PLC has the function "Progress status" available, the status values are displayed at the time of execution of the operations. The status of the local data memory and the accumulators is also displayed.

| Program stat. OFF | 1. | Press the "Program stat. OFF" softkey to hide the program status in the status display. |
| Program stat. ON | 2. | Press the "Program stat. ON" softkey to unhide the program status in the status display. |

Change colors for displaying of progress or program status

In progress status, different colors are used to display information.

Display	Color
Signal flow of power rail, when status active	Blue
Signal flow in the networks	Blue
All operations that are active and that are executed without error (corresponds to signal flow)	Blue
Status of the Boolean operations (corresponds to signal flow)	Blue
Times and counters active	Green
Error during execution	Red
No signal flow	Gray
No network executed	Gray
STOP mode	Gray

Note

Colored display in program status

In the display of the program status only the color of the signal flow is relevant.

Enlarging/reducing the display of the ladder diagram

Zoom +	1.	Press the "Zoom +" softkey to enlarge the section of the ladder diagram.
		After enlarging, the "Zoom -" softkey is available.
Zoom -	2.	Press the "Zoom -" softkey to reduce the section of the ladder diagram again.

Procedure

PLC - CPU	1.	Ladder add-on tool is opened.
Window 1 OB1	2.	Press the "Window 1" or "Window 2" softkey.
...		
Program block	3.	Press the "Program block" softkey.
		The "Program block" list is displayed.
Proper- ties	4.	Press the "Properties" softkey if you wish to display additional information.
		- OR -
Local variables		Press the "Local variables" softkey if you wish to display data of a variable.

16.8 Downloading a PLC user program

Download the project data into the PLC if some changes have been made to the project data and a new PLC user program is available.

When the project data is loaded, the data classes are saved and loaded to the PLC.

Requirement

Check whether the PLC is in Stop mode.

Note

PLC in RUN mode

If the PLC is in RUN mode, a corresponding message is displayed and the "Load in Stop" and "Load in Run" softkeys appear.

With "Load in Stop", the PLC is set to Stop mode and the project is stored and loaded to the CPU.

With "Load in Run", the loading operation is continued and the PLC project loaded to the PLC. Only those data classes that have really been changed are loaded, i.e. generally INDIVIDUAL data classes.

Procedure

PLC - CPU	1.	Ladder add-on tool is opened. You have changed project data.
PLC STOP	2.	Press the "PLC Stop" softkey if the PLC is in the run mode.
Loading in CPU	3.	Press the "Load to CPU" softkey to start the loading operation. All data classes are loaded.
PLC START	4.	When the PLC project has been loaded, press the "PLC Start" softkey to switch the PLC to Run mode.

16.9 Editing the local variable table

You have the option of editing the local variable table of an INT block.

Insert local variable

If you have inserted new networks or operands, it may be necessary to insert new variables in the local variable table of an INT block.

Name	Freely assign.
Variable type	Selection:
	• IN
	• IN_OUT
	• OUT
	• TEMP
Data type	Selection:
	• BOOL
	• BYTE
	• WORD
	• INT
	• DWORD
	• DINT
	• REAL
Comment	Freely assign.

Procedure

Window 1 OB1	1.	The ladder diagram display (LAD) is opened.
...		
Window 2 SBR0		
Program block	2.	Press the "Program block" softkey.
Local variables	3.	Press the "Local variables" softkey. The "Local Variables" window appears and lists the created variables.
Edit	4.	Press the "Edit" softkey. The fields can be edited.

SELECT	5.	Enter a name, select the variable and data type and, if required, enter a comment.
Add line	6.	Press the "Attach line" softkey if you want to add a further variable and enter the data.
		- OR -
Delete line		Select the relevant variable and press the "Delete line" softkey to remove the variable from the list.

16.10   Creating a new block

Create INT blocks i you wish to make changes with the PLC user program.

Name	INT _100, INT_101
	The number from the selection field "Number, interrupt program" is taken for the name of the INT block.
Author	A maximum of 48 characters is permitted.
Number of the interrupt program	100
	101
Data class	Individual
Comment	A maximum of 100 lines and 4096 characters are permitted.

Note
Access protection

You have the option of protecting blocks that have been newly created against being accessed.

Procedure

Window 1 OB1	1.	The ladder diagram display (LAD) is opened.
...		
Window 2 SBR0		
Program block	2.	Press the "Program block" softkey to open the list of program blocks.

Add	3.	Press the "Add" softkey. The "Properties" window appears.
	4.	Enter the name of the author, number of the INT block and, if required, a comment. The data class of the block is specified.
✓ **OK**	5.	Press the "OK" softkey to transfer the block to the list.

16.11 Editing block properties

You can edit the title, author and comments of an INT block.

Note

You cannot edit the block name, interrupt number and data class assignment.

Procedure

Window 1 OB1 **Window 2 SBR0**	1.	The ladder diagram display is opened.
Program block	2.	Select the relevant block and press the "Program block" softkey.
Proper-ties	3.	Press the "Properties" softkey. The "Properties" window appears.

16.12   Inserting and editing networks

You can create a new network and then insert operations (bit operation, assignment, etc.) at the selected cursor position.

Only empty networks can be edited. Networks, that already include statements, can only be deleted.

A simple, single line can be edited for each network. You can create a maximum of 3 columns per network.

Column	Operation	
Column 1	• NO contact • NC contact	-| |- -|/|-
Column 2 (optional)	NOT Rising edge Falling edge Assign Set Reset	-|NOT|- -|P|- -|N|- -( ) -(S) -(R)
Column 3 (only possible if no assign, set or reset operations were specified in the second column)	Assign Set Reset	-( ) -(S) -(R)

Note

Logical AND (serial contact) and logical OR (parallel contact) are not possible.

The bit combinations comprise one or several logical operations and the assignment to an output / bit memory.

If the cursor is moved further to the left with the arrow key, the type of assignment or a logic operation can be selected. A further logic operation cannot be placed to the right of an assignment. A network must always be terminated with an assignment.

References

For further information about PLC programming, please refer to:

Function Manual Basic Functions, PLC for SINUMERIK 828D (P4)

Procedure

Window 1 INT101	1.	An interrupt routine has been selected.
Edit	2.	Press the "Edit" softkey.
	3.	Position the cursor on a network.
Insert network	4.	Press the "Insert network" softkey.
		- OR -
INSERT		Press the <INSERT> key.
		If the cursor is positioned on "Network x", a new, empty network is inserted behind this network.
Insert operation	5.	Position the cursor on the desired element below the network title and press the "Insert operation" softkey.
		The "Insert Operation" window appears.
OK	6.	Select the desired bit operation (NC contact or NO contact) or assignment and press the "OK" softkey.
Insert operand	7.	Press the "Insert operand" softkey.
INPUT	8.	Enter the logic operation or the command and press the <INPUT> key to complete the entry.
Delete operation	9.	Position the cursor on the operation that you want to delete and press the "Delete operation" softkey.
		- OR -
Delete network		Position the cursor on the title of the network that you want to delete and press the "Delete network" softkey.
		- OR -
DEL		Press the <DEL> key.
		The network, including all the logic operations and operands, or the selected operation is deleted.

16.13 Editing network properties

You can edit the network properties of an INT block.

Network title and network comment

The title can have a maximum of three lines and 128 characters. The comment can have a maximum of 100 lines and 4096 characters.

Procedure

Window 1 OB1 Window 2 SBR0	1.	The ladder diagram display (LAD) is opened.
▼ ▶	2.	Use the cursor keys to select the network that you want to edit.
SELECT	3.	Press the <SELECT> key. The "Network title / comment" window opens and shows the title and a possibly assigned comment for the selected network.
Change	5.	Press the "Change" softkey. The fields can be edited.
OK	6.	Enter the changes and press the "OK" softkey to transfer the data to the user program.

16.14 Displaying and editing symbol tables

You can display the symbol tables that are used to obtain an overview of the global operands available in the project - which you can then edit.

The name, address and possibly also a comment is displayed for each entry.

Procedure

PLC – CPU	1.	Ladder add-on tool is opened.
Symbol table	2.	Press the "Symbol table" softkey. The list with the symbol table entries is displayed.
Edit	3.	Press the "Edit" softkey if you want to change entries. The display fields can be edited.
▼ ► ◄ ▲	4.	Use the cursor keys to select the desired entry and the field to be changed.
	5.	Enter the value to be changed. - OR -
Add line		Press the "Attach line" softkey to insert an empty line after the selected entry. - OR -
Delete line		Press the "Delete line" softkey to remove the selected entry from the list. - OR - Enter a new value in the selected field.
✓ OK	7.	Press the "OK" softkey to confirm your action.

16.15 Inserting/deleting a symbol table

New user symbol tables can be generated and changed. Tables that are no longer used can be deleted.

Note

Delete symbol table

The "Delete" softkey is only available if a user symbol table has been selected.

Procedure

Symbol table	1.	The symbol table is opened.
Select sym table	2.	Press the "Select sym. table" softkey. The "Symbol Table - Selection" window appears.
Insert sym table	3.	Position the cursor at the desired position and press the "Insert sym. table" softkey. The "Create Symbol Table" window appears.
OK	4	Enter a symbolic name and press the "OK" softkey. The new user symbol table is inserted in the line after the cursor position. - OR -
Change sym table		Select a symbol table and press the "Change symbol table" softkey if you wish to change the properties of the symbol table.
Delete	5.	Position the cursor on the symbol table that you want to delete and press the "Delete" softkey.

16.16 Searching for operands

You can use the search function to quickly reach points in very large programs where you would like, for example, to make changes.

Restricting the search

- "Window 1" / "Window 2","Symbol table"

  With "Go to", you can jump directly to the desired network.

- "Cross references"

  With "Go to", you can jump directly to the desired line.

Requirement

Window 1 / window 2, the symbol tables or the list of cross references is opened.

Procedure

Search	1.	Press the "Find" softkey.
		A new vertical softkey bar appears. The "Find / Go To" window opens at the same time.
◯ SELECT	2.	Select the "Find operand" entry in the first input field if you are searching for a specific operand and enter the search term in the "Find" input field.
◯ SELECT	3.	Select the search range (e.g. Find all).
◯ SELECT	4.	Select the "In this program unit" or "In all program units" entry if you are in "Window 1" or "Window 2" or in the symbol table in order to restrict the search.
✓ OK	5.	Press the "OK" softkey to start the search.
		If the operand you are searching for is found, the corresponding line is highlighted.
Continue search		Press the "Continue search" softkey if the operand found during the search does not correspond to the element you are looking for.
		- OR -
✗ Cancel		Press the "Cancel" softkey if you want to cancel the search.

Further search options

Go to start	1.	Press the "Go to start" softkey to jump to the start of the ladder diagram in window 1 or window 2, or the list (cross references, symbol table).
Go to end	2.	Press the "Go to end" softkey to jump to the end of the ladder diagram in window 1 or window 2, or the list (cross references, symbol table).

16.17 Displaying the network symbol information table

All of the symbolic identifiers used in the selected network are displayed in the "Network symbol information table" window.

The following information is listed:

- Names

- Absolute addresses

- Comments

The symbol information table remains empty for networks that do not contain any global symbols.

Procedure

Window 1 OB1	1.	The ladder diagram display (LAD) is opened.
Window 2 SBR0		
Symbol info	2.	Select the desired network and press the "Symbol info" softkey. The "Network Symbol Information Table" window appears.
▼ ► ◄ ▲	3.	Use the cursor keys to move within the table.

16.18 Displaying/canceling the access protection

You can password protect your program organizational units (POUs) in the PLC 828 programming tool. This prevents other users from accessing this part of the program. This means that it is invisible to other users and is encrypted when it is downloaded.

A lock symbol is used to show a password-protected POU in the block overview and in the ladder diagram.

Procedure

Window 1 **OB1** **Window 2** **SBR0**	1.	The ladder diagram display (LAD) is opened.
Program block	2	Select the relevant program organizational unit (POU) in the overview and press the "Program block" softkey.
Protected	3.	Press the "Protection" softkey. The "Protection" window appears.
✓ **Accept**	4.	Enter your password and press the "Accept" softkey.

16.19 Displaying cross references

You can display all the operands used in the PLC user project and their use in the list of cross references.

This list indicates in which networks an input, output, bit memory etc. is used.

The list of cross references contains the following information:

● Block

● Address in the network

● Context (command ID)

Symbolic and absolute address

You can choose between specification in absolute or symbolic address.

Elements for which there are no symbolic identifiers are automatically displayed with absolute identifiers.

Opening program blocks in the ladder diagram

From the cross references, you have the option of going directly to the location in the program where the operand is used. The corresponding block is opened in window 1 or 2 and the cursor is set to the corresponding element.

Searching

Using a specific search, you can go directly to the location that you wish to view in more detail:

- Search for operand
- Jump to sought line

Procedure

PLC - CPU	1.	Ladder add-on tool is opened.
Cross refs.	2.	Press the "Cross refs." softkey. The list of cross references appears and the operands are displayed sorted according to absolute address.
Symbolic address	3.	Press the "Symbol. address" softkey. The list of operands is displayed sorted according to symbolic address.
Absolute address	4.	Press the "Absolute address" softkey to return to the display showing the absolute addresses.
Open in window 1 **Open in window 2**	5.	Select the desired cross reference and press the "Open in window 1" or "Open in window 2" softkey. The ladder diagram is opened, and the selected operand is marked.
Search	6.	Press the "Find" softkey. The "Find / Go To" window appears.
SELECT	7.	Select "Find operand" or "Go to" and enter the sought element or the desired line and select the search order (e.g. search up).
OK	8.	Press the "OK" softkey to start the search.
Continue search	9.	If an an element is found that corresponds to the sought element, but is not at the appropriate position, press the "Continue search" softkey to find where the search term occurs next.

Alarm, error, and system messages

<div style="text-align: right; font-size: 3em;">17</div>

17.1 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.

⚠ **CAUTION**

Dangers for persons and machines

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

● Cancel 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. All pending alarms are displayed. The "Hide SI alarms" softkey is displayed if safety alarms are pending.

| | 3. | Press the "Hide SI alarms" softkey if you do not wish to display SI alarms. |

Hide SI alarms

| | 4. | Position the cursor on an alarm. |

...

| | 5. | 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. |

Delete HMI alarm

- OR -

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

Acknowl. alarm

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

Acknowledgement symbols

Symbol	Meaning
I	Turn the unit off and back on (main switch), or press NCK POWER ON.
//	Press the <RESET> key.
⊖ ... HMI	Press the <ALARM CANCEL> key. - OR - Press the "Acknowl. HMI alarm" softkey.
PLC	Press the key provided by the manufacturer.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

17.2 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 of the coming and going events - that have occurred since the HMI 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.

17.3 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

Procedure

Diag-nostics	1.	Select the "Diagnostics" operating area.
Mes-sages	2.	Press the "Messages" softkey. The "Messages" window appears.

17.4 Sorting, alarms, faults and messages

If a large number of alarms, messages or alarm logs are displayed, you have the option of sorting these in an ascending or descending order according to the following criteria:

- Date (alarm list, messages, alarm log)

- Number (alarm list, messages)

As a consequence, for every extensive lists, you can obtain the required information faster.

Procedure

⚠️ Diag-nostics	1.	Select the "Diagnostics" operating area.
Alarm list	2.	Press the softkey "Alarm list", "Messages" or "Alarm log" to display the requested messages and alarms.
...		
Alarm protoc.		
Datum	3.	Press the "Sort" softkey. The list of entries is sorted in ascending order according to date, i.e. the most recent information is at the end of the list.
Decreasing	4.	Press the softkey "Descending" to sort the list in a descending order. The most recent event is shown at the beginning of the list.
Nummer	5.	Press the "Number" softkey if you wish to sort the alarm list or the list with messages according to numbers.
Ascending	6.	Press the "Ascending" softkey if you wish to display the list in an ascending order.

17.5 PLC and NC variables

17.5.1 Displaying and editing PLC and NC variables

The "NC/PLC Variables" window allows NC system variables and PLC variables to be monitored and changed.

You receive the following list in which you can enter the desired NC/PLC variables in order to display the actual values.

- Variable

  Address for NC/PLC variable.

  Incorrect variables have a red background and are displayed with a # character in the value column.

- Comment

  Any comment on the variable.

  The columns can be displayed and hidden.

- Format

  Specify the format in which the variable is to be displayed.

  The format can be specified (e.g. floating point).

- Value

  Displays the actual value of the NC/PLC variables.

PLC variables	
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)
Counters	Counter (Cx)
Data	Data block (DBx): Data bit (DBXx), data byte (DBBx), data word (DBWx), data double word (DBDx)

Formats	
B	Binary
H	Hexadecimal
D	Decimal without sign
+/-D	Decimal with sign
F	Floating point (for double words)
A	ASCII character

Notation for variables

- PLC variables

  EB2

  A1.2

  DB2.DBW2

- NC variables

  – NC system variables - notation

  $AA_IM[1]

  – User variables/GUDs - notation

  GUD/MyVariable[1,3]

  – OPI - notation

  /CHANNEL/PARAMETER/R[u1,2]

Note

NC system variables and PLC variables

- System variables can be dependent on the channel. When the channel is switched over, the values from the corresponding channel are displayed.
- For user variables (GUDs) it is not necessary to make a specification according to global or channel-specific GUDs. The indices of GUD arrays are, just like NC variables in the system variable syntax, 0-based; this means that the first element starts with the index 0.
- Using the tooltip, for NC system variables, you can display the OPI notation (with the exception of GUDs).

Changing PLC variables

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

⚠ WARNING

Incorrect parameterization

Changes in the states of NC/PLC variables 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

Diag-nostics	1.	Select the "Diagnostics" operating area.
NC/PLC variab.	2.	Press the "NC/PLC variables" softkey.
		The "NC/PLC Variables" window opens.
	3.	Position the cursor in the "Variable" column and enter the required variable.
INPUT	4.	Press the <INPUT> key.
		The operand is displayed with the value.
Details	5.	Press the "Details" softkey.
		The "NC/PLC Variables: Details" window opens. The information for "Variable", "Comment" and "Value" is displayed in full length.
SELECT	6.	Position the cursor in the "Format" field and choose the required format with <SELECT>.
Display comments	7.	Press the "Display comments" softkey.
		The "Comments" column is displayed. You have the option of creating comments or editing existing comments.
Display comments		Press the "Display comments" softkey once again to hide the column.
Change	8.	Press the "Change" softkey if you would like to edit the value.
		The "Value" column can be edited.
Insert variable	9.	Press the "Insert variable" softkey if you wish to select a variable from a list of all existing variables and insert this.
		The "Select Variable" window opens.
Filter/ search	10.	Press the "Filter/search" softkey to restrict the display of variables (e.g. to mode group variables) using the "Filter" selection box and/or select the desired variable using the "Search" input box.
Delete all		Press the "Delete all" softkey if you would like to delete the entries for the operands.
OK	11.	Press the "OK" softkey to confirm the changes or the deletion.
		- OR -
Cancel		Press the "Cancel" softkey to cancel the changes.

Note

"Filter/Search" when inserting variables

The start value for "Filter/Search" of variables differs.

For example, to insert the variable $R[0], set "Filter/Search":

- The start value is 0, if you filter according to "System variables".
- The start value is 1, if you filter according to "All (no filter)". In this case, all signals are displayed and shown in the OPI notation.

Changing operands

Depending on the type of operand, you can increment or decrement the address by 1 place at a time using the "Operand +" and "Operand -" softkeys.

Note

Axis names as index

For axis names, the softkeys "Operand +" and "Operand -" do not act as index, e.g. for $AA_IM[X1].

Examples

DB97.DBX2.5

Result: DB97.DBX2.6

$AA_IM[1]

Result: $AA_IM[2]

MB201

Result: MB200

/Channel/Parameter/R[u1,3]

Result: /Channel/Parameter/R[u1,2]

17.5.2 Saving and loading screen forms

You have the option of saving the configurations of the variables made in the "NC/PLC variables" window in a screen form that you reload again when required.

Editing screen forms

If you change a screen form that has been loaded, then this is marked using with * after the screen form name.

The name of a screen form is kept in the display after switching-off.

Procedure

	1.	You have entered values for the desired variables in the "NC/PLC variables" window.
▶▶	2.	Press the ">>" softkey.
Save screen	3.	Press the "Save screen" softkey. The "Save screen: Select archiving" window opens.
✓ OK	4.	Position the cursor on the template folder for variable screen forms in which your actual screen form should be saved and press the "OK" softkey. The "Save screen: Name" window opens.
✓ OK	5.	Enter the name for the file and press the "OK" softkey. A message in the status line informs you that the screen form was saved in the specified folder. If a file with the same name already exists, they you will receive a prompt.
Load mask	6.	Press the "Load screen" softkey. The "Load screen" window opens and displays the sample folder for the variable screen forms.
	7.	Select the desired file and press the "OK" softkey. You return to the variable view. The list of all of the predefined NC and PLC variables is displayed.

17.5.3 Load symbols

PLC data can also be edited via symbols.

To do this, the symbol tables and texts for the symbols in the PLC project must have been suitably prepared (STEP7) and made available in SINUMERIK Operate.

Preparing PLC data

Save the generated files in the /**oem**/sinumerik/plc/symbols directory.

Procedure

NC/PLC variab.	1.	The variable view is opened.
▶▶	2.	Press the ">>" and "Load icons" softkeys. The "Import PLC symbols: *.snh" window opens.
Load symbols		
✓ OK	3.	In the folder "/oem/sinumerik/plc/symbols", select the "PlcSym.snh" file to import the symbols and then click on "OK".
✓ OK	4.	In the folder "/oem/sinumerik/plc/symbols", select the "PlcSym.snt" file to import the symbols and then press the "OK" softkey. You will obtain an appropriate note if the tables were successfully imported.
✓ OK	5.	Press the "OK" softkey. You return to the "NC/PLC variables" window.
	6.	Restart the SINUMERIK Operate in order to activate the files.

17.6 Creating screenshots

You can create screenshots of the current user interface.

Each screenshot is saved as a file and stored in the following folder:

/**user**/sinumerik/hmi/log/screenshot

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.

Copy file

Setup	1.	Select the "Start-up" operating area.
System data	2.	Press the "System data" softkey and open the specified folder.

As you cannot open screenshots in SINUMERIK Operate, you must copy the files to a Windows PC either via "WinSCP" or via a USB-FlashDrive.

You can open the files using a graphics program, e.g. "Office Picture Manager".

17.7 Version

17.7.1 Displaying version data

The following components with the associated version data are specified in the "Version data" window:

- System software
- Basic PLC program
- PLC user program
- System extensions
- OEM applications
- Hardware

Information is provided in the "Nominal version" column as to whether the versions of the components deviate from the version supplied on the CompactFlash Card.

✔	The version displayed in the "Actual version" column matches the version of the CF card.
!	The version displayed in the "Actual version" column does not match the version of the CF card.

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

Diag-nostics	1.	Select the "Diagnosis" operating area.
Version	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.

▼

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

17.7.2　Save information

All the machine-specific information of the control is combined in a configuration via the user interface. You can save machine-specific information on the drives that have been set-up.

Procedure

△)) **Diag- nostics** 　　1.　Select the "Diagnostics" operating area.

V1.2 Version 　　2.　Press the "Version" softkey.
　　　　　It takes some time to call the version display. While the version data is being determined a progress message box and the appropriate text are displayed in the dialog line.

Save ▶ 　　3.　Press the "Save" softkey.

　　　　　The "Save Version Information: Select Archive" window opens. The following storage locations are offered depending on the configuration:

- Local drive
- Network drives
- USB
- Version data (archive: Data tree in the "HMI data" directory)

New directory 　　4.　Then press the "New directory" softkey if you wish to create your own directory.

✓ **OK** 　　5.　Press the "OK" softkey. The directory is created.

✓ **OK** 　　6.　Press the "OK" softkey again to confirm the storage location.

The "Save Version Information: Name" window opens. The following options are available:

- In the "Name:" text field, the file name is pre-assigned with <Machine name/no.>+<CF-card number>. "_config.xml" or "_version.txt" is automatically attached to the file names.
- In text field "Comment", you can add a comment that is stored with the configuration data.

Select the following data via a checkbox:

- Version data (.TXT): Output of pure version data in text format.
- Configuration data (.XML): Output of configuration data in XML format. The configuration file contains the data you entered under Machine identity, the license requirements, the version information and the logbook entries.

7. Press the "OK" softkey to start the data transfer.

See also

Logbook (Page 356)

17.8 Logbook

The logbook provides you with the machine history in an electronic form.

If service is carried out on the machine, this can be electronically saved. This means that it is possible to obtain a picture about the "History" of the control and to optimize service.

Editing the logbook

You can edit the following information:

- Editing information on the machine identity
    - Machine name/No.
    - Machine type
    - Address data
- Make logbook entries (e.g. "filter replaced")
- Deleting logbook entries

Note

Deleting logbook entries

Up to the 2nd commissioning, you have the option to delete all of the entered data up to the time of the first commissioning.

Output of the logbook

You have the possibility of exporting the logbook by generating a file using the "Save version" function in which the logbook is contained as section.

See also

Save information (Page 354)

17.8.1 Displaying and editing the logbook

Procedure

	1.	Select the "Diagnostics" operating area.
	2.	Press the "Version" softkey.
	3.	Press the "Logbook" softkey. The "Machine logbook" window opens.

Editing end customer data

	You have the option of changing the address data of the end customer using the "Change" softkey.
	- OR - Using the "Clean up" softkey, you can delete all logbook entries.
	All entries, except the date of the first commissioning, are deleted and the softkey "Clean up" is deactivated.

Note

Deleting logbook entries

As soon as the 2nd commissioning has been completed, the "Clean up" softkey to delete the logbook data is no longer available.

17.8.2 Making a logbook entry

Using the "New logbook entry" window to make a new entry into the logbook.

Enter your name, company and department and a brief description of the measure taken or a description of the fault.

Note

If you wish to make line breaks in the "fault diagnostics/measure" field, use the key combination <ALT> + <INPUT>.

The date and entry number are automatically added.

Sorting the entries

The logbook entries are displayed numbered in the "machine logbook" window.

More recent entries are always added at the top in the display.

Procedure

	1.	The logbook is opened.
New entry	2.	Press the "New entry" softkey. The "New logbook entry" window opens.
OK	3.	Enter the required data and press the "OK" softkey. You return to the "Machine logbook" window and the entry is displayed below the machine identity data.

Note
Deleting logbook entries

Up to the end of the 2nd commissioning, you have the option to delete the logbook entries up to the time of the first commissioning using the "Clean up" softkey.

Searching for a logbook entry

You have the option for searching for specific entries using the search function.

	1.	The "Machine logbook" window is opened.
Search	2.	Press the "Search..." softkey and enter the desired term in the search form. You can make a search according to date/time, company name/department or according to fault diagnostics/measure.
		The cursor is positioned on the first entry that corresponds to the search term.
Continue search	3.	Press the "Continue search" softkey if the entry found is not the one that you are looking for.

Additional search option

Go to start	Press the "Go to Beginning" softkey to start the search at the latest entry.
Go to end	Press the "Go to End" softkey to start the search at the oldest entry.
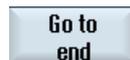
17.9 Remote diagnostics

17.9.1 Setting remote access

You can influence the remote access to your control in the "Remote diagnostics (RCS)" window.

Here, rights for all types of remote control are set. The selected rights are defined from the PLC and using the setting at the HMI.

The HMI can restrict the rights specified from the PLC, but however, cannot extend the rights beyond the PLC rights.

If the settings made permit access from outside, then this is still dependent on a manual or automatic confirmation.

Rights for remote access

The "Specified from PLC" field shows the access rights for remote access or remote monitoring specified from the PLC.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

In the "Selected in the HMI" selection box, you have the possibility of setting rights for remote control:

● Do not permit remote access

● Permit remote monitoring

● Permit remote control

The combination of the settings in the HMI and in the PLC show the valid status as to whether access is permitted or not. This is displayed in the "Resulting from" line.

Settings for the confirmation dialog box

If the settings made for "Specified from the PLC" and "Selected in the HMI" permit access from outside, then this is however, still dependent on either a manual or automatic confirmation.

As soon as a remote access is permitted, at all of the active operating stations, a query dialog box is displayed for the operator at the active operating station to either confirm or reject an access.

For the case that there is no local operation, then the control behavior can be set for this particular scenario. You define how long this window is displayed and whether, after the confirmation has expired, the remote access is automatically rejected or accepted.

Display of the state

Remote monitoring active

Remote control active

If remote access is active, using these icons you will be informed in the status line as to whether a remote access is presently active or whether only monitoring is permitted.

Procedure

Diagnostics	1.	Select the "Diagnostics" operating area.
Remote RCS diag.	2.	Press the "Remote diag." softkey. The "Remote diagnostics (RCS)" window is opened.
Change	3.	Press the "Change" softkey. The "Selected in the HMI" is activated.
>	4.	If you desire remote control, select the entry "Permit remote control". In order that remote control is possible, the entry "Permit remote control" must be specified in the fields "Specified from the PLC" and "Selected in the HMI".
	5.	Enter new values in the group "Behavior for confirming remote access" if you wish to change the behavior for confirming remote access.
OK	6.	Press the "OK" softkey. The settings are accepted and saved.

References

For a description of configuration options, refer to the

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

17.9.2 Permit modem

You can permit remote access to your control via a teleservice adapter IE connected at X127.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

Software option

You need the "MC Information System RCS Host" option to display the "Permit modem" softkey.

Procedure

Remote diag.	1. The "Remote diagnostics (RCS)" window is opened.
Allow rem. access	2. Press the "Permit modem" softkey. Access to the control via modem is enabled so that a connection can be established.
Allow modem	3. To block access again, press the "Permit modem" softkey again.

17.9.3 Request remote diagnostics

Using the "Request remote diagnostics" softkey, from your control you have the option of actively requesting remote diagnostics with your machinery construction OEM.

The access via modem must be enabled if the access is to be made via modem.

Machine manufacturer

Please refer to the machine manufacturer's specifications.

When requesting remote diagnostics, you obtain a window with the corresponding pre-assigned data and values of the ping service. If required, you can ask your machine manufacturer for this data.

Data	Meaning
IP address	IP address of the remote PC
Port	Standard port that is intended for remote diagnostics
Send duration	Duration of the request in minutes
Send interval	Cycle in which the message is sent to the remote PC in seconds
Ping send data	Message for the remote PC

Procedure

Remote diag.	1.	The "Remote diagnostics (RCS)" window is opened.
Request rem.diagn.	2.	Press the "Request remote diagnostics" softkey. The "Request remote diagnostics" window is displayed.
Change	3.	Press the "Change" softkey if you would like to edit the values.
OK	4.	Press the "OK" softkey. The request is sent to the remote PC.

References

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

See also

Permit modem (Page 362)

17.9.4 Exit remote diagnostics

Procedure

	1.	The "Remote diagnostics (RCS)" is opened and it is possible that remote monitoring or remote access is active.
	2.	Block the modem access if access via modem is to be blocked.
		- OR -
		In the "Remote Diagnostics (RCS)" window, reset the access rights to "Permit no remote access".

See also

Permit modem (Page 362)

Setting remote access (Page 360)

Appendix

A

A.1 840D sl documentation overview

SINUMERIK 840D sl documentation overview

General documentation

Sales brochure

Catalog NC 62

Catalog PM 21 SIMOTION,
SINAMICS S120 and motors
for production machines

User documentation

Operating Manual
– Universal
– Turning
– Milling

Programming manual
– Fundamentals
– Job planning
– Measuring cycles

Programming manual
– ISO turning
– ISO milling

Diagnostics manual

Manufacturer/service documentation

Device manual
– NCU
– operator components
  and networking

System manual
Guidelines for the machine
configuration

System manual
Ctrl Energy

Commissioning manual
– CNC: NCK, PLC,
  Drive
– Base software and
  operating software

List manual
– Machine data
– Interface signals
– Variables

Manufacturer/service documentation

Function manual
– Basic functions
– Extended functions
– Special functions
– Synchronous actions
– ISO dialects

Function manual
Tool management

Function manual
Drive functions

Function manual
Safety Integrated

Configuration manual
EMC Design Guidelines

Info/training

Training documents
– Simple milling
  with ShopMill
– Simple turning
  with ShopTurn

Manuals
Tool and
mold making

Electronic documentation

DOConCD

My Documentation
Manager

Industry Mall

Index

A

Actual-value display, 32
Adapter-transformed view, 220
Alarm log
    display, 343
    Sorting, 345
Alarms
    Displaying, 341
    Sorting, 345
Any file
    Creating, 236
Archive
    Generate in the program manager, 258
    generating in the system data, 259
    Punched tape format, 258
    read in archive from system data, 263
    reading-in in the Program Manager, 261
Asian characters, 44
Auxiliary functions
    H functions, 136
    M functions, 136
Axes
    Defined increment, 84
    Direct positioning, 86
    Referencing, 52
    Repositioning, 99
    Traversing, 84
    Variable increment, 85

B

Backing up
    Data - in the Program Manager, 258
    Data - via the system data, 259
    Setup data, 264
Base offset, 63
Basic blocks, 96
Binary format, 258
Block
    Searching, 101
    Searching - interruption point, 104
    Searching - search pointer, 105
Block search
    Mode, 107

    Program interruption, 104
    Search pointer, 105
    Search target parameters, 106
    Search target specification, 103
    Using, 101

C

Calculator, 40
Channel switchover, 57
Coarse and fine offsets, 63
Code carrier connection, 201
Collision avoidance, 177
Context-sensitive online help, 48
Coordinate system
    Switching over, 59
Copying
    Directory, 246
    Program, 246
Create
    Program block, 119
Creating
    Any file, 236
    Directory, 233
    G Code program, 235
    Job list, 238
    NC directory on the local drive, 227
    Program list, 239
    Workpiece, 234
Ctrl-Energy
    Energy analysis, 294
    Energy-saving profiles, 299
    Functions, 293
    Measuring the energy consumption, 295
    Saved measuring curves, 297

D

Data block (SB2), 94
Deleting
    Directory, 248
    Program, 248
Device
    Activate/deactivate, 313
    Enabling, 312
Directory
    Copying, 246

Virtual keyboard, 290

W

Wear, 205
Working area limitation
    Defining, 70
Workpiece
    Creating, 234
Workpiece counter, 139

Z

Zero offsets
    Active ZO, 64
    Delete, 69
    Displaying details, 68
    Overview, 65
    Settable ZO, 67
    Setting, 61
Zero point settings
    Backing up, 264
    reading in, 266