

SINUMERIK

SINUMERIK 828D SINUMERIK Operate

Commissioning Manual

Introduction	1
Fundamental safety instructions	2
General settings	3
Machine and setting data	4
Connecting drives	5
Configuring the network	6
Customizing the "Machine" operating area	7
Simulation and simultaneous recording	8
Technologies and cycles	9
Spindle function	10
Service and diagnostics	11
Appendix	A

Valid for:

CNC software version 4.95

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.

Table of contents

1	Introduction	9
1.1	About SINUMERIK	9
1.2	About this documentation	10
1.3	Documentation on the internet	11
1.3.1	Documentation overview SINUMERIK 828D	11
1.3.2	Documentation overview SINUMERIK operator components	11
1.4	Feedback on the technical documentation	13
1.5	mySupport documentation	14
1.6	Service and Support.....	15
1.7	Important product information	17
2	Fundamental safety instructions.....	19
2.1	General safety instructions.....	19
2.2	Equipment damage due to electric fields or electrostatic discharge	23
2.3	Warranty and liability for application examples	24
2.4	Security information	25
2.5	Residual risks of power drive systems	26
3	General settings.....	27
3.1	Access levels	27
3.1.1	Definition of access levels.....	27
3.1.2	Modifying the access levels password	28
3.1.3	Work station safety	30
3.1.4	Access levels for programs	30
3.1.5	Access rights for files.....	32
3.1.6	Access levels for softkeys.....	33
3.1.7	This is how you define new access levels for softkeys	34
3.2	Set date/time	37
3.3	Using the time server.....	39
3.4	Device registration.....	41
3.5	Activation of the display for the remaining time of spindle utilization.....	42
3.6	Changing the language.....	43
3.7	Configuring Caps Lock.....	45
3.8	Inserting a user-specific run up screen	46
3.9	Creating screenshots.....	47
3.10	Changing the skin design	48

3.11	Setting how login data (credentials) are handled when archiving.....	49
3.12	VNC remote connections.....	50
3.13	Setting the WEB Browser.....	54
3.14	Activate and configure the Sidescreen window.....	56
3.14.1	Activate Sidescreen.....	56
3.14.2	Display Sidescreen ABC keyboard.....	59
3.14.3	Display Sidescreen machine control panel.....	59
3.14.3.1	Configuring your own keys on the machine control panel.....	60
3.14.3.2	Create own translations of texts for the machine control panel.....	63
3.15	Automatic save data archive.....	65
4	Machine and setting data.....	67
4.1	User views.....	67
4.1.1	Creating a user view.....	68
4.1.2	Editing the user view.....	69
4.1.3	Deleting a user view.....	71
4.2	Plain texts for machine and setting data.....	72
5	Connecting drives.....	75
5.1	Overview.....	75
5.2	Setting up drives.....	76
5.3	Setting EXTCALL.....	77
5.4	Creating a global part program memory.....	78
5.5	Program selection.....	79
5.5.1	Displaying the softkey.....	79
6	Configuring the network.....	81
6.1	Displaying the network overview.....	82
6.2	Settings of the factory network.....	83
6.3	Saving network settings.....	85
6.4	Station-related network diagnostics.....	86
6.4.1	Displaying the PPU network interfaces.....	86
6.4.2	TCP/IP diagnostics and configuration.....	87
6.4.3	Ethernet network diagnostics.....	90
6.4.3.1	Diagnostics of the network and DHCP server.....	90
6.4.3.2	Accessible nodes.....	91
7	Customizing the "Machine" operating area.....	95
7.1	Setting the font size of the actual value display.....	96
7.2	Inserting a user-specific logo.....	97
7.3	Configuring the display of the G-code groups.....	99
7.4	Configuring the channel operational message.....	101
7.5	Deactivating program test.....	103

7.6	Display articulated joint position STAT and rotary axis position TU	104
7.7	Activating the "Teach In" function	105
7.8	Block search	107
7.8.1	Activating block search mode	107
7.8.2	Accelerated block search for execution from external	108
7.9	Manual machine	109
7.10	Multi-channel support	112
7.11	User status display (OEM)	114
7.12	Configuring the offset display	116
7.13	Activate machining time recording	117
8	Simulation and simultaneous recording	119
8.1	Simulation overview	119
8.2	Setting the technology for simulation	121
8.3	Simultaneous recording overview	124
8.4	Clamping a blank	126
9	Technologies and cycles	129
9.1	Activating turning/milling/drilling/grinding technologies	129
9.2	Technology cycles for drilling	134
9.3	Manufacturer cycles	136
9.3.1	Manufacturer cycles	136
9.3.2	PROG_EVENT.SPF standard cycle	137
9.3.3	Manufacturer cycle for tool change CUST_T and CUST_M6	138
9.3.4	CUST_TECHCYC.SPF manufacturer cycle	139
9.3.5	CUST_MULTICHAN user cycle	142
9.3.6	Commissioning multi-channel machines with programSYNC	143
9.4	Milling	150
9.4.1	General	150
9.4.2	Technology cycles for milling	150
9.4.3	Setting-up ShopMill cycles for milling	151
9.4.4	Cylinder surface transformation (TRACYL_K/TRACYL)	154
9.4.5	Example: Milling machine with general axis configuration and TRACYL	155
9.4.6	ShopMill cycles for multiple clamping	158
9.4.7	Activating/deactivating Surface Turning (CYCLE953)	160
9.5	Turning	163
9.5.1	General	163
9.5.2	Setting up cycles for turning	163
9.5.3	Setting-up ShopTurn cycles for turning	167
9.5.4	Setting up a counterspindle	169
9.5.5	Setting up the counterspindle under ShopTurn	172
9.5.6	Technology cycles for turning	173
9.5.7	Single-channel and two-channel sensors	177
9.5.8	Example: Two-channel turning machine with general axis configuration	179
9.5.9	Cylinder surface transformation (TRACYL_K/TRACYL)	180

9.5.10	End face machining (TRANSMIT_K/TRANSMIT)	182
9.5.11	Inclined Y axis (TRAANG_K/TRAANG)	183
9.5.12	X zero point not at the rotation center	184
9.5.13	Spindle speed limitation for current program	186
9.6	Grinding	187
9.7	Swiveling	190
9.7.1	Technology cycles for swiveling	190
9.7.2	CYCLE800 checklist for the identification of the machine kinematics	197
9.7.3	Setting up a tool carrier/swivel data set	198
9.7.4	Setting up the tool carrier based on kinematic chain	204
9.7.5	Examples of machine kinematics for commissioning the tool carrier	206
9.7.6	Manufacturer cycle CUST_800.SPF	216
9.7.7	Indirectly programming the name of the swivel data set	222
9.7.8	CYCLE9960 workpiece - Measurement of machine kinematic	223
9.8	Load-dependent controller setting	225
9.8.1	Adapt to load CYCLE782	225
9.8.2	CYCLE782: Help screens	226
9.8.3	Manufacturer cycle CUST_782.SPF	228
9.9	High Speed Cutting (HSC)	230
9.9.1	High speed settings: Configuring CYCLE832	230
9.9.2	Manufacturer cycle CUST_832.SPF	231
9.9.3	Top Surface: CYCLE832 extension	232
9.10	Measuring cycles and measurement functions	235
9.10.1	Measuring cycles and measurement functions, general	235
9.10.2	Manufacturer and user cycle CUST_MEACYC.SPF	238
9.10.3	Measuring in the JOG mode	240
9.10.3.1	Measuring workpieces at the milling machines	243
9.10.3.2	Measuring tools at the milling machines	245
9.10.3.3	Measuring tools at the turning machines	251
9.10.3.4	Measuring without electronic probe in JOG	252
9.10.4	Measuring in the AUTOMATIC mode	252
9.10.4.1	Measuring workpieces, general	254
9.10.4.2	Measuring workpieces at the milling machines	257
9.10.4.3	Measuring tools at the milling machines	258
9.10.4.4	Measuring workpieces at the turning machines	264
9.10.4.5	Measuring tools at the turning machines	265
9.11	Compare cycles version	266
9.11.1	Display cycles version	266
9.11.2	Specify cycles version	267
10	Spindle function	271
10.1	Spindle control	271
10.2	Spindle diagnostics	273
10.2.1	Spindle diagnostics	273
10.2.2	Temperatures	274
10.2.3	Motor temperature sensor	276
10.2.4	Additional temperature sensor	277
10.2.5	Temperature histograms	277
10.2.6	Speed/torque	278

10.2.7	Clamping system	278
10.2.8	Clamping system: Speed limits	280
10.2.9	Clamping system: Diagnostic statistics.....	281
10.2.10	Clamping system: Clamping time statistics	282
10.2.11	Fetching the logistics data.....	283
11	Service and diagnostics	285
11.1	NC/PLC variables	285
11.1.1	Displaying and editing PLC and NC variables	285
11.1.2	Saving and loading screen forms	289
11.2	Displaying the Service overview	291
11.2.1	Selecting axes and drives	292
11.2.2	Axis diagnostics	294
11.2.3	Service drive	298
11.3	PROFINET diagnostics.....	307
11.3.1	Displaying the PROFINET configuration	307
11.3.2	Displaying details for PROFINET devices	309
11.4	Backing up log data for service case	311
11.4.1	Backing up log data to USB.....	311
11.4.2	Backing up log data as ZIP file.....	311
11.5	Trace	312
11.5.1	Variables filter/search	313
11.5.2	Trace settings.....	315
11.5.2.1	Trace settings (PLC, NC, servo)	315
11.5.2.2	Trace settings (drive).....	316
11.5.2.3	Trace options	318
11.5.3	Starting the trace	318
11.5.4	Evaluate a trace	319
11.5.4.1	Selecting attributes of a variable	319
11.5.4.2	Setting trace views.....	323
11.5.4.3	Selecting a variable	324
11.5.4.4	Scaling the display	324
11.5.4.5	Zooming the display	325
11.5.4.6	Position the cursor	327
11.5.4.7	Acquiring measured values	327
11.5.5	Trace session files	328
11.5.5.1	Creating a session file.....	328
11.5.5.2	Saving the trace file.....	329
11.5.5.3	Load trace session file	330
11.6	Drive system	331
11.6.1	Displaying drive states.....	331
11.6.2	Displaying details of the drive objects	332
11.7	Action log	334
11.7.1	Setting the action log.....	334
11.7.2	Inserting variables	336
11.7.3	Displaying the log file.....	337
11.7.4	Searching in the log files	337
11.7.5	Storing a log	338
11.7.6	Structure of a log file	339

11.7.7	Advanced settings.....	341
11.8	HMI trace.....	343
A	Appendix.....	345
A.1	Information about third-party software used	345
A.2	Abbreviations	347
	Index	351

Introduction

1.1 About SINUMERIK

From simple, standardized CNC machines to premium modular machine designs – the SINUMERIK CNCs offer the right solution for all machine concepts. Whether for individual parts or mass production, simple or complex workpieces – SINUMERIK is the highly dynamic automation solution, integrated for all areas of production. From prototype construction and tool design to mold making, all the way to large-scale series production.

Visit our website for more information SINUMERIK (<https://www.siemens.com/sinumerik>).

1.2 About this documentation

Target group

This document is intended for commissioning personnel.

The plant or system is installed, connected and ready to start. For the following steps, e.g. configuring the individual components, the Commissioning Manual contains all the necessary information or at least references.

Benefits

The Commissioning Manual allows the target group to test and commission the plant or the system in a professional and safe way.

Usage phase: Setup and commissioning phase

Standard scope

This documentation only describes the functionality of the standard version. This may differ from the scope of the functionality of the system that is actually supplied. Please refer to the ordering documentation only for the functionality of the supplied drive system.

It may be possible to execute other functions in the system which are not described in this documentation. This does not, however, represent an obligation to supply such functions with a new control or when servicing.

For reasons of clarity, this documentation cannot include all of the detailed information on all product types. Further, this documentation cannot take into consideration every conceivable type of installation, operation and service/maintenance.

The machine manufacturer must document any additions or modifications they make to the product themselves.

Websites of third-party companies

This document may contain hyperlinks to third-party websites. Siemens is not responsible for and shall not be liable for these websites and their content. Siemens has no control over the information which appears on these websites and is not responsible for the content and information provided there. The user bears the risk for their use.

1.3 Documentation on the internet

1.3.1 Documentation overview SINUMERIK 828D

Comprehensive documentation about the functions provided in SINUMERIK 828D Version 4.8 SP4 and higher is provided in the 828D documentation overview (<https://support.industry.siemens.com/cs/ww/en/view/109766724>).



You can display documents or download them in PDF and HTML5 format.

The documentation is divided into the following categories:

- User: Operating
- User: Programming
- Manufacturer/Service: Configuring
- Manufacturer/Service: Commissioning
- Manufacturer/Service: Functions
- Manufacturer/Service: Safety Integrated
- SINUMERIK Integrate/MindApp
- Info & Training

1.3.2 Documentation overview SINUMERIK operator components

Comprehensive documentation about the SINUMERIK operator components is provided in the Documentation overview SINUMERIK operator components (<https://support.industry.siemens.com/cs/document/109783841/technische-dokumentation-zu-sinumerik-bedienkomponenten?dti=0&lc=en-WW>).

You can display documents or download them in PDF and HTML5 format.

The documentation is divided into the following categories:

- Operator Panels
- Machine control panels

1.3 Documentation on the internet

- Machine Pushbutton Panel
- Handheld Unit/Mini handheld devices
- Further operator components

An overview of the most important documents, entries and links to SINUMERIK is provided at SINUMERIK Overview - Topic Page (<https://support.industry.siemens.com/cs/document/109766201/sinumerik-an-overview-of-the-most-important-documents-and-links?dti=0&lc=en-WW>).

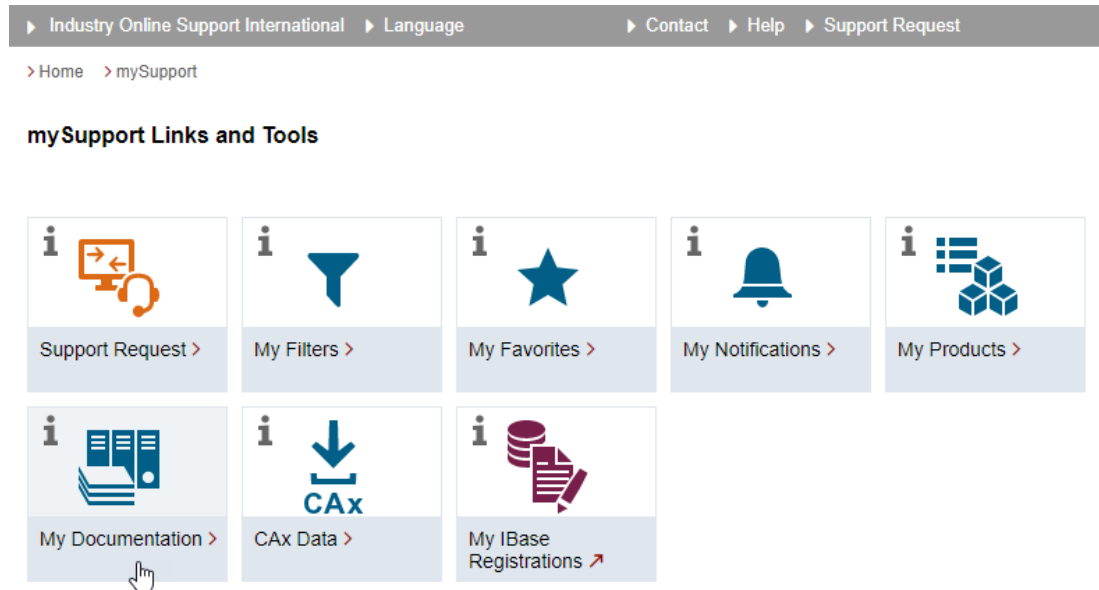
1.4 Feedback on the technical documentation

If you have any questions, suggestions or corrections regarding the technical documentation which is published in the Siemens Industry Online Support, use the link "Send feedback" link which appears at the end of the entry.

1.5 mySupport documentation

With the "mySupport documentation" web-based system you can compile your own individual documentation based on Siemens content, and adapt it for your own machine documentation.

To start the application, click on the "My Documentation" tile on the mySupport homepage (<https://support.industry.siemens.com/cs/ww/en/my>):



The configured manual can be exported in RTF, PDF or XML format.

Note

Siemens content that supports the mySupport documentation application can be identified by the presence of the "Configure" link.

1.6 Service and Support

Product support

You can find more information about products on the internet:

Product support (<https://support.industry.siemens.com/cs/ww/en/>)

The following is provided at this address:

- Up-to-date product information (product announcements)
- FAQs (frequently asked questions)
- Manuals
- Downloads
- Newsletters with the latest information about your products
- Global forum for information and best practice sharing between users and specialists
- Local contact persons via our Contacts at Siemens database (→ "Contact")
- Information about field services, repairs, spare parts, and much more (→ "Field Service")

Technical support

Country-specific telephone numbers for technical support are provided on the internet at address (<https://support.industry.siemens.com/cs/ww/en/sc/4868>) in the "Contact" area.

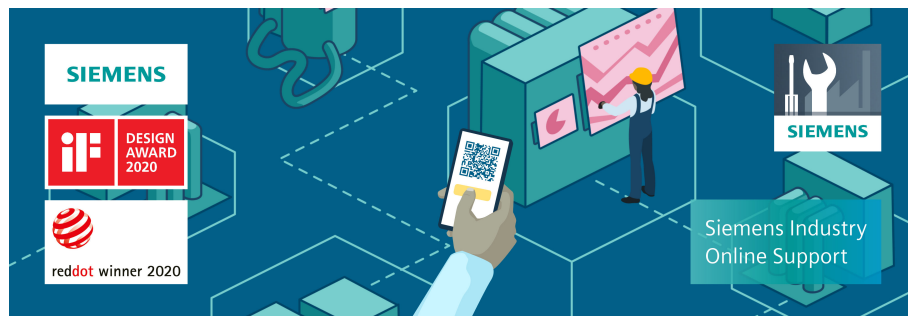
If you have any technical questions, please use the online form in the "Support Request" area.

Training

You can find information on SITRAIN at the following address (<https://www.siemens.com/sitrain>).

SITRAIN offers training courses for automation and drives products, systems and solutions from Siemens.

Siemens support on the go





With the award-winning "Siemens Industry Online Support" app, you can access more than 300,000 documents for Siemens Industry products – any time and from anywhere. The app can support you in areas including:

- Resolving problems when implementing a project
- Troubleshooting when faults develop
- Expanding a system or planning a new system

Furthermore, you have access to the Technical Forum and other articles from our experts:

- FAQs
- Application examples
- Manuals
- Certificates
- Product announcements and much more

The "Siemens Industry Online Support" app is available for Apple iOS and Android.

Data matrix code on the nameplate

The data matrix code on the nameplate contains the specific device data. This code can be read with a smartphone and technical information about the device displayed via the "Industry Online Support" mobile app.

1.7 Important product information

Using OpenSSL

This product can contain the following software:

- Software developed by the OpenSSL project for use in the OpenSSL toolkit
- Cryptographic software created by Eric Young.
- Software developed by Eric Young

You can find more information on the internet:

- OpenSSL (<https://www.openssl.org>)
- Cryptsoft (<https://www.cryptsoft.com>)

Compliance with the General Data Protection Regulation

Siemens observes standard data protection principles, in particular the data minimization rules (privacy by design).

For this product, this means:

The product does not process or store any personal data, only technical function data (e.g. time stamps). If the user links this data with other data (e.g. shift plans) or if he/she stores person-related data on the same data medium (e.g. hard disk), thus personalizing this data, he/she must ensure compliance with the applicable data protection stipulations.

Fundamental safety instructions

2.1 General safety instructions



WARNING

Electric shock and danger to life due to other energy sources

Touching live components can result in death or severe injury.

- Only work on electrical devices when you are qualified for this job.
- Always observe the country-specific safety rules.

Generally, the following steps apply when establishing safety:

1. Prepare for disconnection. Notify all those who will be affected by the procedure.
2. Isolate the drive system from the power supply and take measures to prevent it being switched back on again.
3. Wait until the discharge time specified on the warning labels has elapsed.
4. Check that there is no voltage between any of the power connections, and between any of the power connections and the protective conductor connection.
5. Check whether the existing auxiliary supply circuits are de-energized.
6. Ensure that the motors cannot move.
7. Identify all other dangerous energy sources, e.g. compressed air, hydraulic systems, or water. Switch the energy sources to a safe state.
8. Check that the correct drive system is completely locked.

After you have completed the work, restore the operational readiness in the inverse sequence.



WARNING

Electric shock due to connection to an unsuitable power supply

When equipment is connected to an unsuitable power supply, exposed components may carry a hazardous voltage. Contact with hazardous voltage can result in severe injury or death.

- Only use power supplies that provide SELV (Safety Extra Low Voltage) or PELV- (Protective Extra Low Voltage) output voltages for all connections and terminals of the electronics modules.



WARNING

Electric shock due to equipment damage

Improper handling may cause damage to equipment. For damaged devices, hazardous voltages can be present at the enclosure or at exposed components; if touched, this can result in death or severe injury.

- Ensure compliance with the limit values specified in the technical data during transport, storage and operation.
- Do not use any damaged devices.



WARNING

Electric shock due to unconnected cable shields

Hazardous touch voltages can occur through capacitive cross-coupling due to unconnected cable shields.

- As a minimum, connect cable shields and the cores of cables that are not used at one end at the grounded housing potential.



WARNING

Electric shock if there is no ground connection

For missing or incorrectly implemented protective conductor connection for devices with protection class I, high voltages can be present at open, exposed parts, which when touched, can result in death or severe injury.

- Ground the device in compliance with the applicable regulations.

NOTICE

Damage to equipment due to unsuitable tightening tools.

Unsuitable tightening tools or fastening methods can damage the screws of the equipment.

- Be sure to only use screwdrivers which exactly match the heads of the screws.
- Tighten the screws with the torque specified in the technical documentation.
- Use a torque wrench or a mechanical precision nut runner with a dynamic torque sensor and speed limitation system.

**WARNING****Spread of fire from built-in devices**

In the event of fire outbreak, the enclosures of built-in devices cannot prevent the escape of fire and smoke. This can result in serious personal injury or property damage.

- Install built-in units in a suitable metal cabinet in such a way that personnel are protected against fire and smoke, or take other appropriate measures to protect personnel.
- Ensure that smoke can only escape via controlled and monitored paths.

**WARNING****Unexpected movement of machines caused by radio devices or mobile phones**

Using radio devices or mobile telephones in the immediate vicinity of the components can result in equipment malfunction. Malfunctions may impair the functional safety of machines and can therefore put people in danger or lead to property damage.

- Therefore, if you move closer than 20 cm to the components, be sure to switch off radio devices or mobile telephones.
- Use the "SIEMENS Industry Online Support app" only on equipment that has already been switched off.

**WARNING****Fire due to inadequate ventilation clearances**

Inadequate ventilation clearances can cause overheating of components with subsequent fire and smoke. This can cause severe injury or even death. This can also result in increased downtime and reduced service lives for devices/systems.

- Ensure compliance with the specified minimum clearance as ventilation clearance for the respective component.

NOTICE**Overheating due to inadmissible mounting position**

The device may overheat and therefore be damaged if mounted in an inadmissible position.

- Only operate the device in admissible mounting positions.



WARNING

Unexpected movement of machines caused by inactive safety functions

Inactive or non-adapted safety functions can trigger unexpected machine movements that may result in serious injury or death.

- Observe the information in the appropriate product documentation before commissioning.
- Carry out a safety inspection for functions relevant to safety on the entire system, including all safety-related components.
- Ensure that the safety functions used in your drives and automation tasks are adjusted and activated through appropriate parameterizing.
- Perform a function test.
- Only put your plant into live operation once you have guaranteed that the functions relevant to safety are running correctly.

Note

Important safety notices for Safety Integrated functions

If you want to use Safety Integrated functions, you must observe the safety notices in the Safety Integrated manuals.



WARNING

Malfunctions of the machine as a result of incorrect or changed parameter settings

As a result of incorrect or changed parameterization, machines can malfunction, which in turn can lead to injuries or death.

- Protect the parameterization against unauthorized access.
- Handle possible malfunctions by taking suitable measures, e.g. emergency stop or emergency off.

2.2 Equipment damage due to electric fields or electrostatic discharge

Electrostatic sensitive devices (ESD) are individual components, integrated circuits, modules or devices that may be damaged by either electric fields or electrostatic discharge.



NOTICE

Equipment damage due to electric fields or electrostatic discharge

Electric fields or electrostatic discharge can cause malfunctions through damaged individual components, integrated circuits, modules or devices.

- Only pack, store, transport and send electronic components, modules or devices in their original packaging or in other suitable materials, e.g. conductive foam rubber or aluminum foil.
- Only touch components, modules and devices when you are grounded by one of the following methods:
 - Wearing an ESD wrist strap
 - Wearing ESD shoes or ESD grounding straps in ESD areas with conductive flooring
- Only place electronic components, modules or devices on conductive surfaces (table with ESD surface, conductive ESD foam, ESD packaging, ESD transport container).

2.3 Warranty and liability for application examples

Application examples are not binding and do not claim to be complete regarding configuration, equipment or any eventuality which may arise. Application examples do not represent specific customer solutions, but are only intended to provide support for typical tasks.

As the user you yourself are responsible for ensuring that the products described are operated correctly. Application examples do not relieve you of your responsibility for safe handling when using, installing, operating and maintaining the equipment.

2.4 Security information

Siemens provides products and solutions with industrial security functions that support the secure operation of plants, systems, machines and networks.

In order to protect plants, systems, machines and networks against cyber threats, it is necessary to implement – and continuously maintain – a holistic, state-of-the-art industrial security concept. Siemens' products and solutions constitute one element of such a concept.

Customers are responsible for preventing unauthorized access to their plants, systems, machines and networks. Such systems, machines and components should only be connected to an enterprise network or the internet if and to the extent such a connection is necessary and only when appropriate security measures (e.g. firewalls and/or network segmentation) are in place.

For additional information on industrial security measures that may be implemented, please visit

<https://www.siemens.com/industrialsecurity> (<https://www.siemens.com/industrialsecurity>).

Siemens' products and solutions undergo continuous development to make them more secure. Siemens strongly recommends that product updates are applied as soon as they are available and that the latest product versions are used. Use of product versions that are no longer supported, and failure to apply the latest updates may increase customer's exposure to cyber threats.

To stay informed about product updates, subscribe to the Siemens Industrial Security RSS Feed under

<https://www.siemens.com/industrialsecurity> (<https://new.siemens.com/global/en/products/services/cert.html#Subscriptions>).

Further information is provided on the Internet:

Industrial Security Configuration Manual (<https://support.industry.siemens.com/cs/ww/en/view/108862708>)



WARNING

Unsafe operating states resulting from software manipulation

Software manipulations, e.g. viruses, Trojans, or worms, can cause unsafe operating states in your system that may lead to death, serious injury, and property damage.

- Keep the software up to date.
- Incorporate the automation and drive components into a holistic, state-of-the-art industrial security concept for the installation or machine.
- Make sure that you include all installed products into the holistic industrial security concept.
- Protect files stored on exchangeable storage media from malicious software by with suitable protection measures, e.g. virus scanners.
- On completion of commissioning, check all security-related settings.

2.5 Residual risks of power drive systems

When assessing the machine- or system-related risk in accordance with the respective local regulations (e.g., EC Machinery Directive), the machine manufacturer or system installer must take into account the following residual risks emanating from the control and drive components of a drive system:

1. Unintentional movements of driven machine or system components during commissioning, operation, maintenance, and repairs caused by, for example,
 - Hardware and/or software errors in the sensors, control system, actuators, and cables and connections
 - Response times of the control system and of the drive
 - Operation and/or environmental conditions outside the specification
 - Condensation/conductive contamination
 - Parameterization, programming, cabling, and installation errors
 - Use of wireless devices/mobile phones in the immediate vicinity of electronic components
 - External influences/damage
 - X-ray, ionizing radiation and cosmic radiation
2. Unusually high temperatures, including open flames, as well as emissions of light, noise, particles, gases, etc., can occur inside and outside the components under fault conditions caused by, for example:
 - Component failure
 - Software errors
 - Operation and/or environmental conditions outside the specification
 - External influences/damage
3. Hazardous shock voltages caused by, for example:
 - Component failure
 - Influence during electrostatic charging
 - Induction of voltages in moving motors
 - Operation and/or environmental conditions outside the specification
 - Condensation/conductive contamination
 - External influences/damage
4. Electrical, magnetic and electromagnetic fields generated in operation that can pose a risk to people with a pacemaker, implants or metal replacement joints, etc., if they are too close
5. Release of environmental pollutants or emissions as a result of improper operation of the system and/or failure to dispose of components safely and correctly
6. Influence of network-connected communication systems, e.g. ripple-control transmitters or data communication via the network

For more information about the residual risks of the drive system components, see the relevant sections in the technical user documentation.

General settings

3.1 Access levels

3.1.1 Definition of access levels

Access to programs, data and functions is protected in a user-oriented hierarchical system of seven access levels. These are divided into

- Three password levels for manufacturer, service and users
- Four keyswitch positions for users

There are access levels 1 to 7 (see table below); where

- 1 is the highest and
- 7 is the lowest level.

Table 3-1 Access level concept

Access level	Protected by	Area
1	Password: SUNRISE (default value)	Manufacturer
2	Password: EVENING (default value)	Service
3	Password: CUSTOMER (default value)	User
4	Keyswitch 3	Programmer, machine setter
5	Keyswitch 2	Qualified operator
6	Keyswitch 1	Trained operator
7	Keyswitch 0	Semi-skilled operator

Password security

Note

Assigning secure passwords

Observe the following rules when creating new passwords:

- When assigning new passwords, make sure that you never assign easily guessed passwords, such as simple words, easily guessed key combinations on the keyboard, etc.
- Passwords must always contain a combination of upper-case and lower-case letters as well as numbers and special characters. Passwords must comprise at least eight characters. PINs must comprise an arbitrary sequence of digits.
- Wherever possible and where it is supported by the IT systems and the software, always set the highest complexity of characters for the selection of a password.

The German Federal Office for IT Security (BSI) (https://www.bsi.bund.de/SharedDocs/Downloads/DE/BSI/Grundschrift/International/GSK_15_EL_EN_Draft.pdf?__blob=publicationFile&v=2) provides additional rules for creating secure passwords.

You can use a program for password management for assistance when dealing with passwords. With its help, you can save passwords and PINs in encrypted form, manage them and generate secure passwords.

3.1.2 Modifying the access levels password

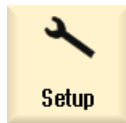
Overview

You can activate the access levels protected with passwords from the user interface.

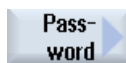
The following options are available for editing the passwords:

- Set new password
- Change password
- Delete password

Setting the password



1. Select the "Setup" operating area.



2. Press the "Password" softkey.



3. Press the "Set password" softkey.
The "Set Password" window opens.
4. Enter one of the possible default passwords and press the "OK" softkey to confirm the entry.
A permissible password is set and the valid access level is displayed.
Invalid passwords will be rejected.

Change password

You must change the standard passwords to obtain a secure access protection.



1. Press the "Change password" softkey.
The "Change Password" window opens.
2. Select the area (e.g. user) for which you want to assign a new password.
3. Enter the new password in the input fields "New password" and "Repeat password".
Always assign a secure password.
4. Press the "OK" softkey to confirm your entry.
A new, valid password is only accepted if the two terms entered are identical.

Note

Invalid passwords are rejected by the operating software. Note the rules for strong passwords:

- A password must contain at least 8 and can contain up to 32 characters.
- Permissible characters are:
 - Uppercase characters
 - Lowercase characters
 - Numbers
 - Special characters (ASCII 0x20 - 0x7E)
- The character combination of a password must contain:
 - At least one number
 - At least one uppercase letter
 - At least one lowercase letter
- A password must not be too similar to the following names:
 - User names (manufacturer, service, user)
 - Computer name
 - The additional values that can be stored for a user in the Linux password file (so-called GECOS data)

We recommend that you confine yourself to the characters available on the operator panel front when selecting a password.

Delete password



Press the "Delete password" softkey. The access authorization is reset.

Note

Access authorization is not automatically deleted at POWER ON.

3.1.3 Work station safety

In order to secure machines against manipulation and protect people from accidents, proceed as follows when leaving the work station:



1. Set the keyswitch to 0 and then remove it.
2. Press the "Delete password" softkey.
Access authorization is then reset.

3.1.4 Access levels for programs

Objective

Access levels can be allocated to protect information; they can also be used to prevent unauthorized operation or deliberate damage to the control system or machine.

Access levels for files and directories specify which users are authorized to execute which operations on a directory or a file in a file system. Operations in this sense include:

- Reading the contents of a file (READ), copying the file or placing the file in an archive.
- Changing the content of a file (WRITE) or deleting the file.
- Executing a file (EXECUTE).
- Listing and displaying a file in a directory (SHOW).
- Creating a file in a directory.
- Displaying or reading the contents of a directory.
- Creating or deleting a directory.

Function description

The **individual allocation of access rights** acts, in the passive NC file system, on the following directories:

- Part programs
- Subprograms

- Manufacturer cycle directory
- User cycle directory
- Workpiece directory
- Workpiece directories

Furthermore, the function acts on the files of the main program type (*.mpf) and subprograms or cycle (*.spf).

In addition, the technique can be expanded to include the following directories on the local drive:

- Part programs
- Subprograms
- Workpiece directory
- Workpiece directories

Note**Working via remote access**

When working via remote access (e.g. WinSCP or Access MyMachine), the same access rights apply for editing or saving files and directories as when working directly at the control system.

Additional information

Information about access rights with WinSCP or Access MyMachine is provided in the:

- NCU Operating System
- SINUMERIK Integrate Access MyMachine /OPC UA

Access rights via machine data

The following machine data is still available to **globally allocate access rights**:

MD51050 \$MNS_ACCESS_WRITE_PROGRAM

Default setting = 4 (access level: User)

Note**Effect of access rights**

The more restrictive setting is always effective both when individual access rights have been allocated and when settings have been made via MD51050 \$MNS_ACCESS_WRITE_PROGRAM.

Settings in the "slfsfileattributes.ini" file

The "slfsfileattributes.ini" file is available as a template in the following directory:

../siemens/sinumerik/hmi/template/cfg

The values from 1 ... 7 correspond to the access levels that, for example, also apply to the password. The 4-digit numbers designate the sequence according to the following rights:

Meaning (left to right): R W X S		
R	READ	Reading files and directories
W	WRITE	Changing, deleting or creating files and directories
X	EXECUTE	Executing a program: for files, for directories, always 7
S	SHOW	Outputting a list of files and directories

Copy the "slfsfileattributes.ini" file into one of the following directories:

../oem/sinumerik/hmi/cfg or ../user/sinumerik/hmi/cfg

The operating software must be restarted for the changes to become effective.

Example

The paths are composed of the data type identifiers. In the following example, "\wks.dir*.wpd *.mpf" stands for the data type for workpiece directories "wks.dir" the workpieces "*.wpd" and the part programs "*.mpf".

For all workpieces, 7777 is defined as the default setting for new part programs.

All entries are in the section [ACCESSMASKS] and have the following form:

```
[ACCESSMASKS]
```

```
\wks.dir\*.wpd\*.mpf = 7777
\mpf.dir\*.mpf = 6577
\cus.dir\*.spf = 6577
```

See also

Definition of access levels (Page 27)

3.1.5 Access rights for files

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.

Note**Settings for the access rights to directories and files**

Via the configuration file and MD51050, access rights of the directories and file types of the NC memory and user memory (local drive) can be changed and pre-assigned.

Configuration file "slfsfileattributes.ini"

The access rights defined in the configuration file "slfsfileattributes.ini" and in MD51050 become valid in the NC when creating new files and can then be changed individually in the Program Manager using the "Properties" function. The access rights are an individual attribute of each file in the NC.

The access rights defined in the configuration file "slfsfileattributes.ini" and MD51050 become valid for external data on the CompactFlash Card of the NCU, or on the hard disk of the PCU/PC immediately for all the specified directories and file types, and cannot be changed through operation. The access rights can only be set globally for external data.

3.1.6 Access levels for softkeys

Use

The display and operation of softkeys can be suppressed by both the manufacturer as well as the user. This allows the operating software to be specifically adapted to the required functional scope and therefore be configured as transparently as possible.

This means that the functional scope of the system is restricted, to prevent access to functions in the operating software, or to restrict the possibility of operator errors.

Access hierarchy

A new access level can only be allocated for those softkeys that are also visible and can be used in the actual access level.

Example: If a softkey is only accessible with the "Manufacturer" access level, then the access level cannot be changed by an operator with access level "Key-operated switch setting 3".

Restrictions

To ensure that the system remains stable and operational, access rights to certain softkeys cannot be customized, e.g.:

- Softkey "Customizing softkeys"
- Softkey "<< Back" on the vertical softkey bar

3.1 Access levels

The following softkeys cannot be hidden or the access level changed:

- All "OK" softkeys
- All "Cancel" softkeys
- All "Accept" softkeys
- All "<<" Back and ">>" Continue softkeys

The access level for softkeys in the cycle dialog boxes can also be changed; this does not apply to softkeys in Easy Screen dialog boxes.

Note

Software upgrade

As all changes of the access levels for softkeys are described using the softkey ID, they still remain valid after a software upgrade, which may have resulted in softkeys being shifted.

Softkeys, where access is already protected by machine data - for example MD51045 \$MNS_ACCESS_TEACH_IN - are only protected by the allocated access level after a new one has been allocated. The machine data is no longer evaluated.

All of the softkeys hidden using this mechanism can be temporarily displayed again using the access rights specified by the system. This is done using the "Show all softkeys" softkey. The softkey is deactivated as long as no softkeys have been allocated new access levels: This setting remains effective until the system is rebooted.

3.1.7 This is how you define new access levels for softkeys

General sequence

In order to allocate a new access level to a softkey, proceed as follows:

- Activate the adaptation mode.
- Change the access level of a softkey.
- Accept the changes and confirm.

Preconditions

Access to the "Customize softkeys" is protected using a general MD51073 \$MNS_ACCESS_SET_SOFTKEY_ACCESS. This machine data is pre-assigned access level 3, i.e. the function is available from "Service" password and higher. In order that an operator with "User" access level can use this function, set this machine data to a value > 3.

An external mouse is required in order to make changes.

Changing the access level of a softkey

Procedure

1. In the "Start-up" operating area, select the "HMI" menu.
2. On the expansion bar ">>" select the "Customize softkeys" softkey.
You can obtain precise information about the procedure in the following dialog.
3. Confirm with "OK", in order to activate the "Customize the softkey access levels active" mode.
A table is displayed.
4. Select an operating area.
5. Place the mouse cursor on a softkey and using the right-hand mouse key call the context menu "Define access level", in order to define a new access level for this softkey.
 - The displayed access level is the default setting.
 - A softkey is practically made invisible with "Always hide".
 - You can reset the additionally allocated access level using "Original setting". This selection is only listed after an individual access level was set.
6. In order to accept changes, return to the "Start-up" operating area and there, press the "Close customizing" softkey. This means that you exit the "Customize the softkey access levels active" mode.
If you acknowledge the following query with "OK", then the changes are accepted and become effective once the system has rebooted.
7. If the changes to a softkey are not to be applied, select the line in the "Adapt softkey access levels" window and press the "Original setting" softkey. The identifier "Original setting" is entered in the "New access level" column. After the end of the softkey adaptation and system restart, the original setting of the softkey is restored and the line entry in the "Adapt softkey access levels" window is deleted.

Example

The following access levels were changed:

Softkey	New access level
SISuMainScreenMenuHu::0::3	Manufacturer
SIMaJogMillMenuHU::jogHuMore::6	User
SISuPaUserDataMenu::::4	Service

Once selected, the identifier and position of the softkey are transferred to the "Softkey" column.

Result

When the changes have been accepted, a configuration file with the name "slsoftkeyaccess.xml" is created and saved in the file system under the following path: **../user/sinumerik/hmi/cfg**. Files can also be saved on the **/oem** path; these files are evaluated when the system boots. The following priority applies: oem < user.

Note

Editing the file "slsoftkeyaccess.xml"

In order to avoid errors in the xml syntax and for the softkey identifier the "slsoftkeyaccess.xml" file may only be edited via the operating software as described above, and not directly in the file system under "System data". It is possible that the complete file or individual entries for softkeys can no longer be interpreted.

3.2 Set date/time

You have the possibility of manually setting the date and time for the SINUMERIK control. In so doing, you can select from various display formats.

Formats of the date display

Formats	Example
d.M.yy	3.1.09
dd.MM.yy	03.01.09
dd.MM.yyyy	03.01.2009
d/M/yy	3/01/09
dd/MM/yy	03/01/09
dd/MM/yyyy	03/01/2009
M/d/yy	1/3/09
MM/dd/yy	01/03/09
MM/dd/yyyy	01/03/2009
yyyy/MM/dd	2009/01/03

The following applies:

Date	Description of the entry
d	Single or double digit day of the month (1 - 31).
dd	Double digit day of the month, with leading zero (01 - 31).
M	Single or double digit month of the year (1 - 12).
MM	Double digit month of the year, with leading zero (01 - 12).
yy	Double digit year (00 - 99).
yyyy	Four-digit year (e.g. 2009).

Time display formats

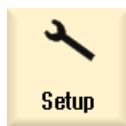
Formats	Example	Hour system
H:m:s AP	1:2:4 PM	24 hour system, with display of "AM" / "PM"
h:m:s ap	1:2:8 pm	12 hour system, with display of "am" / "pm"
HH:mm:ss AP	13:02:08 PM	24 hour system, with display of "AM" / "PM"
hh:mm:ss ap	01:02:09 pm	12 hour system, with display of "am" / "pm"
H:mm:ss AP	1:02:09 PM	24 hour system, with display of "AM" / "PM"
h:mm:ss ap	1:02:09 pm	12 hour system, with display of "am" / "pm"
h:mm:ss	13:02:09	24 hour system
hh:mm:ss	13:12:04	24 hour system

The following applies:

Time symbol:	Description of the entry
h	Single or double digit hour (0 - 23 or 1 - 12 for am/pm).
hh	Double digit hour, with leading zero (00 - 23 or 01 - 12 for am/pm).

Time symbol:	Description of the entry	
H	Single or double digit hour (0 - 23 for AM/PM).	
HH	Double digit hour, with leading zero (00 - 23 for AM/PM).	
m	Single or double digit minute (0 - 59).	
mm	Double digit minute, with leading zero (00 - 59).	
s	Single or double digit second (0 - 59).	
ss	Double digit second, with leading zero (00 - 59).	
AP	Display of the time of day:	AM or am = before midday or PM or pm = after midday

Procedure



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



2. Press the "HMI" softkey.



3. Press the "Date/Time" softkey.
The "Set Date and Time" window opens.



4. Select the required formats for the date and time with the <SELECT> key in the "Format" field.



5. In the entry fields after "new" enter the actual date and time.
6. Confirm the entry with the "OK" softkey.
The new date and time details are transferred.

3.3 Using the time server

To ensure that the correct time information is always automatically available to the control, you can connect up to three time servers to the operating software. Make your settings in the "Set Date and Time" window.

Requirement

You require as a minimum access level 3 (User) to use time servers.

Checking the connection status

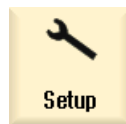
The icon on the right in field "Name / IP" shows the status of the server connection.

Icon	Meaning
	The connection status is unknown. The settings have not yet been processed or set.
	You are connected to the time server. The system is successfully synchronizing times.
	The system is still attempting to synchronize with a time server.

Note

The system checks the time server connection cyclically. If a change is made, it may take a few seconds for the setting to take effect.

Procedure



1. Select the "Setup" operating area.



2. Press the "HMI" softkey.



3. Press the "Date/Time" softkey.
The "Set Date and Time" window opens.
4. Select the "Use time server" check box.
The input fields "Name / IP" and "Time zone" open.
5. In field "Name / IP," enter the DNS name or IP address of the time server.
You can enter up to three time servers separated by blanks.
6. In the field "Time zone," enter your time zone.

3.3 Using the time server



7. Press the "Set time server" softkey.
If a connection to a time server exists, the settings will be synchronized with the system immediately. After that, a cyclic check is made as to whether the settings are effective. The "Set Date and Time" window remains open.



8. Confirm the entry with the "OK" softkey.
The time server settings and the current times are taken over from the control.

Note

If the field "Use time server" is cleared again, the standard "UTC" time zone will apply again.

3.4 Device registration

You must register your machine so it can be supported by Siemens Support. The registration is relevant for the warranty.

Message 150412 is displayed if the machine is not registered:

No information about SIEMENS warranty available.

Note

Message 150412 cannot be acknowledged.

Register the machine in the Diagnostics operating area:

"Diagnostics" → "Version" → "Register machine"

Additional information

Additional information on how to register the machine is provided in the online documentation Commissioning Functions and Service.

3.5 Activation of the display for the remaining time of spindle utilization

In order to protect the spindle from overload, you can display the maximum remaining time of the spindle utilization in seconds in the T,F,S window in addition to the control display of the spindle utilization. The time and spindle utilization symbol are displayed when the remaining time is less than 120 seconds at the current spindle load.

Axis machine data

Set the following machine data:

MD53035 \$MAS_SHOW_SPINDLE_UTIL_TIME	Show spindle utilization time
= 0	Do not show spindle utilization time
= 1	Show spindle utilization time

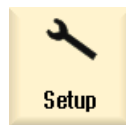
3.6 Changing the language

The operating software is available as standard in several languages.

Selecting a language

You can define the language selection mode using the following machine data.

MD9100 \$MM_CHANGE_LANGUAGE_MODE	Language selection mode
= 1	The user-interface language is specified via the "Language Selection" window (default value).



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



2. Press the "Change language" softkey.
The "Language Selection" window opens. The language most recently set is highlighted.



3. Switch the cursor to the required language.
4. Press the <INPUT> key.

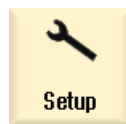
- OR -



Press the "OK" softkey.
The operating software switches to the language selected.

Defining two languages

MD9100 \$MM_CHANGE_LANGUAGE_MODE	Language selection mode
= 2	Selection of two languages. During operation, the "Change language" softkey can be used to toggle between these languages.



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



2. Press the "HMI" softkey.



3. Press the ">>" softkey.



4. Press the "Language selection" softkey.
The "Language Selection" window opens.
All the installed languages are displayed in the fields "First language" and "Second language".
5. Select a language in each of these.

3.6 Changing the language



6. Press the <INPUT> key.

- OR -



Press the "OK" softkey.



During operation, you can switch between the two languages by pressing the "Change language" softkey.

3.7 Configuring Caps Lock

The "CAPSLOCK on" function ensures that text entered via an external keyboard is always entered in upper case and not in lower case.

The key behavior is set using the following display machine data.

Setting

MD9009 \$MM_KEYBOARD_STATE	
= 0	CAPSLOCK off (default value)
= 2	CAPSLOCK on

If the machine data is changed, the system must be rebooted.

3.8 Inserting a user-specific run up screen

OEM-specific run up screen

You can replace the Siemens run up screen by your own run up screen.

Procedure:

1. Call your own run up screen "splash.png".

Note

The file name must not contain any lower case letters.

2. Save your own run up screen in one of the following directories:
/user/sinumerik/hmi/ico/ico640 or
/oem/sinumerik/hmi/ico/ico640 or
/addon/sinumerik/hmi/ico/ico640

Note

The size of the screen is of no significance. A resolution of 640x480 pixels is always set.

3.9 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`

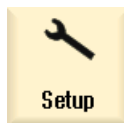
Procedure

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 are in ascending order from "SCR_SAVE_0001.png" to "SCR_SAVE_9999.png". You can create up to 9,999 screenshots.

Copy file



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



2. Press the "System data" softkey.



3. Open the folder specified above, and select the required screenshots.

4. Press the "Copy" softkey.

- OR -



Press the "Cut" softkey.



5. Open the required archive directory, e.g. on a USB flash drive and press the "Paste" softkey.

3.10 Changing the skin design

Display machine data

You can change the skin design via the following item of machine data.

MD9112 \$MM_HMI_SKIN		Skin design The number of the skin is indicated
= 0	Skin 0	
= 1	Skin 1 (standard)	
= 2	Skin 2	

After changing the machine data, the operating software must be restarted.

3.11 Setting how login data (credentials) are handled when archiving

If you generate a commissioning archive, then SINUMERIK Operate issues a dialog query as to whether all of the login data for the set up drives and OPC UA should also be saved to the archive. You parameterize the login data handling using the following display machine data:

Display machine data

MD9112 \$MM_SAVE_CREDENTIALS		Backing up access authorization for network drives when archiving
= 0	Dialog is displayed (default setting)	
= 1	No dialog, access authorizations are not saved	
= 2	No dialog, access authorizations are always saved	

Note

Using shortcut key CTRL+ALT+S, no login data is saved to the archive.

3.12 VNC remote connections

From SINUMERIK Operate, you have the possibility of operating external systems via VNC:

- Define a remote access, and link the selection via a softkey configured in the operating area.
- Define key forwarding to use key combinations in the VNC control area as an alternative.

The remote connection is started using a softkey configured in the operating area menu. If the sidescreen is activated, the VNC operating area can be additionally called using the navigation bar of the sidescreen.

Note

The VNC operating area only supports hextile encoding with color depths of 16 or 32 bits.

Requirement

In the plant or company network, there is at least one system where a VNC server has been set up.

Examples

The following use cases are possible, for example:

- Connection to additional controls
- Connection to the CAD system
- Connection to tool measuring station
- Connection to other PCs

Procedure

You make the remote connection settings via the "Setting up the operating area" window:

Operating area "Commissioning" → "HMI" → "Operating area menu"

Press the "Details" softkey if you want to parameterize additional key forwarding or the "Display VNC source" drop-down list.

You can define an alternative shortcut key for the "Select menu" key.

Note

Reserved softkeys

Softkeys reserved for the system are marked, and are not available for free configuration.

General information

Entry		Description
Operating area 1 - 24		
Type	VNC	Remote access to an external control or PC
	Empty	A VNC that has already been set up is deleted
Computer name		Name or IP address of the external system
Password		Password for logging on to the VNC server The password is displayed encrypted as "*".
Status bar	Off Top Bottom	Display position of the VNC access information: • Name of the system or IP address • Port • Connection status (e.g. for an incorrect connection)
Port		Interface to the VNC server The default port is 5900.
Display VNC source (only visible if "Details" softkey selected)	Cutout Left Width Top Height Full screen	Definition of the displayed range The VNC viewer display shows the entire source screen. To enable the source screen to be completely visible in the VNC viewer, it must often be stretched or compressed (possibly by changing the aspect ratio). Configuring the size and position of the picture details defines the area of the VNC source screen which you would like to see in the VNC viewer. This picture detail is also stretched or compressed (possibly by changing the aspect ratio) to enable it to be completely visible in the VNC viewer.
Operate header	On Off	Display bar for operating area and message line active/inactive

Specifications for the configured softkey

Entry		Description
Access level	Key-operated switch, position 0 ... manufacturer	Assign access rights to the actual connection: from access level 7 (key operated switch position 0) to access level 1 (manufacturer).
Softkey text		Two-line label on the softkey %n is accepted as a line separator. If the first line is too long, then a line break is automatically inserted. If a space is present, it is taken as a line separator. For language-dependent softkey texts, the text ID is entered, which is used to search in the text file. If nothing is specified in the entry field, then the symbolic name of the operating area, which is automatically assigned, is used as softkey text (e.g. AreaVNC_500).
Softkey icon	No icon	No icon is displayed on the softkey.
	hd_vnc.png 	File names of the icon displayed on the softkey.
Text file		File and context for the language-dependent softkey text. If nothing is specified in the input fields, the text appears on the softkey as was specified in the "Softkey text" input field.
Text context		

Key transfer to the external system

Press the "Details" softkey to display the area "Key forwarding", and activate the appropriate checkbox.

When you press "Menu Select" an additional window opens in which you can select the key combination that you want to use.

Note

Be aware that, depending on the operator panel that you are using, not all keys will be available to you.

Designation	Key combination	Description
M	Shift+F10	Calls the "Machine" operating area (M key)
Cancel alarm	Esc	Deletes alarms and messages
Machine	Shift+F11	Calls the "Machine" operating area (hard key)
Program manager	Home (numpad)	Calls the "Program Manager" operating area
Channel	F11	Switches further for multiple channels

Designation	Key combination	Description
Program	End (Numpad)	Calls the "Program" operating area
Alarm	Page Up (Numpad)	Calls the "Diagnosis" operating area
Menu Select	F10	Opens the operating area menu You can change the shortcut key in the input field. When attempting to edit, the "Alternative key combination for Menu Select" dialog opens, allowing the following parameterization be made: A - Z, F1 - F12, Home, End, Page Up, Page Down AND combinable with SHIFT, CTRL, ALT
Help	F12	Calls the context-sensitive online help for the selected window
Offset	Page Down (Numpad)	Calls the "Parameter" operating area
Custom	Shift+F12	Calls the "Custom" operating area
Next Window	Home	Toggles between the windows

Automatic key transfer

If the VNC operating area is selected, then the following keys and/or key combinations are directly transferred to the VNC server:

Keys of the operator panel	Key combination
Horizontal softkeys (upper bar, HU)	F1 – F8
Vertical softkeys (right bar, VR)	Shift+F1 - Shift+F8
Horizontal softkeys (lower bar, HL)	Ctrl+F1 - Ctrl+F8
Vertical softkeys (left bar, VL)	Shift+Ctrl+F1 - Shift+Ctrl+F8
Recall	F9
ETC switchover	Shift+F9
Extended horizontal softkeys (upper bar, HU)	F13 - F20
Extended, vertical softkeys (right bar, VR) and right direct keys HT8	Shift+F13 - Shift+F20
Extended horizontal softkeys (lower bar, HL)	Ctrl+F13 - Ctrl+F20
Extended, vertical softkeys (left bar, VL) and left direct keys HT8	Shift+Ctrl+F13 - Shift+Ctrl+F20
Lower and left-hand softkey bar (HL, VL)	(Shift)+Ctrl+F1 - (Shift)+Ctrl+F8

3.13 Setting the WEB Browser

In SINUMERIK Operate you have the possibility to set up the additional operating area "WEB Browser".

With the WEB Browser you can display web pages directly on the control.

You adapt the WEB Browser operating area to your requirements by configuring it accordingly.

The WEB Browser is started by means of a configured softkey configured in the operating area menu. If the sidescreen is activated, the VNC operating area can be additionally called via the navigation bar of the sidescreen.

Requirement

The control system is connected in the network with the Internet, intranet or a web server.

Example

To obtain additional information for your activity, you use web pages on the Internet or intranet.

Procedure

You make the WEB Browser settings via the "Setting up the operating area" window:

Operating area "Commissioning" → "HMI" → "Operating area menu"

General information

Entry		Description
Operating area 1 - 24		
Type	WEB Browser	Displays web pages on the control
Start URL		URL which is displayed when the WEB browser is started.
Address specification	On Off	Field for entering the URL/Internet address in the WEB browser
Status display	On Off	Status display in the WEB browser, e.g. loading the progress indicator for URL
Search	On Off	Search field in WEB Browser
Home URL		If a Home URL is specified, the corresponding icon appears in the WEB Browser. If you press the Home URL icon, the Home URL is accessed.
Operate header	On Off	Display bar for operating area and message line
Navigation	On Off	Navigation arrow in WEB Browser

Specifications for the configured softkey

Entry		Description
Access level	Keyswitch position 0	Operability of the configured softkey
	...	Access level 7 - job-trained operator
Softkey text	Manufacturer	...
		Access level 1 - manufacturer
Softkey icon	No icon	A text-ID for language-specific text or a two-line label for the softkey %n is accepted as a line separator. If the first line is too long, a line break is automatically inserted. Any spaces are used as line breaks. For language-specific softkey texts, the text ID is entered which is used to search in the text file.
	sk_web.png	No icon is displayed on the softkey.
		File names of the icon displayed on the softkey.
Text file		File and context for language-dependent softkey text. If nothing is specified in the entry fields, the text in input field "Softkey text" is used for all languages.
Text context		

3.14 Activate and configure the Sidescreen window

3.14.1 Activate Sidescreen

In addition to SINUMERIK Operate, Multitouch Panels with widescreen displays offer the option of displaying more elements on the additional area at the side of the screen. Widgets that quickly and continually display important information are shown by default. Pages that include a virtual keyboard and virtual operator panel front are also available. In addition, you can also configure your own pages.

Additional information

More information and examples on the sidescreen function can be found in:

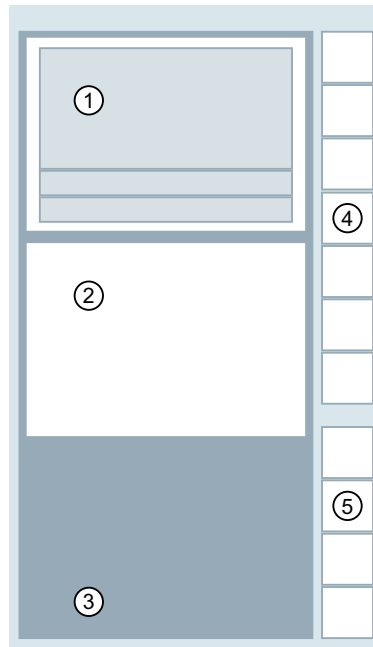
- Easy XML Programming Manual
- Run MyScreens Programming Manual
- Universal/Turning/Milling/Grinding Operating Manual

Requirement

To use the function, you need a Multitouch panel in the widescreen format (resolution, 1366x768, 1920x1080).

Sidescreen structure

Several pages can be displayed on the sidescreen. Only one page is active at a time. You can toggle between the pages using the page selection keys. The standard page contains a standard element to display widgets. You can configure additional elements on this page.



- ① Widgets in the Siemens standard element
- ② Element (OEM) on a standard page
- ③ Page
- ④ Navigation bar for SINUMERIK Operate
- ⑤ Keys to select pages

Activation



WARNING

Unexpected machine movements caused by incorrect operation

Incorrect operation of devices with touch screen can result in unexpected machine movements. This can result in personal injury or damage to property.

- Prevent unintentional incorrect operation by configuring the user interface appropriately, e.g. by placing the buttons for starting and stopping machine components far enough apart.
- Configure the machine in such a way that safety-related functions are not operated with the touch screen.

3.14 Activate and configure the Sidescreen window

You activate the Sidescreen function via the following display machine data:

MD9114 \$MM_SIDESCREEN	SINUMERIK Operate Sidescreen
= 1	Enables the additional sidescreen display area of the Multitouch Panel (HD Ready or Full HD).

The bar and value displays of the widgets are defined via machine data:

Axis load display

You can activate the detection of additional drive actual values for the axis load display in the sidescreen:

MD36730 \$MA_DRIVE_SIGNAL_TRACKING	Detection of additional drive actual values
= 1	\$AA_LOAD: Drive load \$AA_POWER: Drive efficiency \$AA_TORQUE: Drive torque setpoint \$AA_CURR: Smoothed current actual value (quadrature-axis current) of the drive

Note

Detection for the axis load display is only possible if actual values are available from the drive.

Bar display for axis and spindle power

You can assign parameters for the bar display for the axis power using the following axis-machine data:

MD53030 \$MAS_AXIS_MAX_POWER	Maximum value of the axis power display
= 100	Default Maximum permitted axis power in percent The bar display is green in the range between 0 and the value of AXIS_MAX_POWER.

MD53031 \$MAS_AXIS_POWER_RANGE	Display range of the axis power display
= 100	Default Full-scale value for axis power display in percent This value must be greater than or equal to AXIS_MAX_POWER. The bar display is red in the range between the values of AXIS_MAX_POWER and AXIS_POWER_RANGE.

You can assign parameters for the bar display for the spindle power using the following general machine data:

MD51030 \$MNS_SPIND_MAX_POWER	Maximum value of the spindle power display
= 100	Default Maximum permitted spindle power in percent The bar display is green in the range between 0 and the value of SPIND_MAX_POWER.

MD51031 \$MNS_SPIND_POWER_RANGE	Display range of the spindle power display
= 100	Default Full-scale value for spindle power display in percent The value must be equal to or greater than SPIND_MAX_POWER. The bar display is red in the range between the values of SPIND_MAX_POWER and SPIND_POWER_RANGE.

Note

If you do not need a standard widget, set the comment character ";" before the corresponding keyword in the sample file "slsidescreen.ini".

3.14.2 Display Sidescreen ABC keyboard

Not only the standard windows but also the window for an ABC keyboard can be displayed in the Sidescreen display of the Multitouch panel.

Procedure

1. Copy the sample file "slsidescreen.ini" from the directory
/siemens/sinumerik/hmi/template/sidescreen
2. Place the copy of the file in one of the following directories:
/oem/sinumerik/hmi/cfg or
/user/sinumerik/hmi/cfg
3. Open the copy and delete the comment sign ";" in front of the keyword PAGE002.

Note

The values of the attributes `name` and `implementation` must not be changed.

4. Save the setting.
The ABC keyboard can be selected after a run-up.

3.14.3 Display Sidescreen machine control panel

Not only the standard windows but also the window for machine control panel functions can be displayed in the Sidescreen display of the Multitouch panel.

Multiple machine control panel windows can be optionally displayed.

The example of a machine control panel is displayed by the following procedure:

3.14 Activate and configure the Sidescreen window

Procedure

1. Copy the sample files "slsidescreenmcpconfig.xml", "slsidescreenmcpconfig_eng.ts" and the subdirectories of the icon files from the directory
/siemens/sinumerik/hmi/template/sidescreen/mcp/
The icon files are stored in the following subdirectories:
/ico1024 for HD Ready
/ico1920 for Full HD
2. Place the copy of "slsidescreenmcpconfig.xml" in one of the following directories:
/oem/sinumerik/hmi/proj or
/user/sinumerik/hmi/proj
3. Place the copy of "slsidescreenmcpconfig_eng.ts" in one of the following directories:
/oem/sinumerik/hmi/lng or
/user/sinumerik/hmi/lng

Note

The TS file is automatically integrated by the system during startup.

4. Save the icon files under the following path in one of the intended subdirectories:
/oem/sinumerik/hmi/ico or
/user/sinumerik/hmi/ico
Depending on the screen resolution, there are the following subdirectories:
/ico1024 for HD Ready
/ico1920 for Full HD
5. Copy the sample file "slsidescreen.ini" from the directory
/siemens/sinumerik/hmi/template/sidescreen
6. Place the copy of "slsidescreen.ini" in one of the following directories:
/oem/sinumerik/hmi/cfg or
/user/sinumerik/hmi/cfg
7. Open the copy of the file "slsidescreen.ini", and delete the comment sign ";" in front of the keyword PAGE003.
Multiple machine control panels are optionally available:
PAGE004, PAGE005 and PAGE006

Note

The values of the attributes name and implementation must not be changed.

8. Save the setting.
The machine control panel can be selected after a run-up.

3.14.3.1 Configuring your own keys on the machine control panel

Your own keys can also be displayed in the sidescreen for the machine control panel. To do this, the keys must be configured in the file "slsidescreenmcpconfig.xml". The properties, position of the keys and the headers are specified here.

Example of the file "slsidescreenmcppageconfig.xml"

```

<sideScreenControlPanel>
  <mainBlock id="sideScreenMcpBlock">
    <block position="1" title="SPINDLE">
      <row position="1">
        <buttonGroup spacing="false" title="COOLANT">
          <button position="1" text="ON" icon="slsidescreenmcp_cooling.png"
style="redStyle" title="Spindle">
            <onPressed address="HmiUserKey1"/>
            <active address="HmiUserLed1"/>
            <disabled address="DisableHmiUserKey1"/>
          </button>
          <button position="2" text="OFF" icon="slsidescreenmcp_cooling_off.png">
            <onPressed address="HmiUserKey2"/>
            <active address="HmiUserLed2"/>
          </button>
        </buttonGroup>
      </row>
    </block>
  </mainBlock>
</sideScreenControlPanel>

```

Adaptation options of the file "slsidescreenmcppageconfig.xml"

You can make the settings for your own keys in the `<sideScreenControlPanel>` tag:

Tag	Description	
<code><mainBlock id="sideScreenMcpBlock"></code>	Is the main section in the Sidescreen display	
<code><block position="1" title="SPINDLE"></code>	position	Order of the blocks
	title	Block header
<code><row position="1"></code>	Each block has any number of lines.	
	position	Positioning of the lines
<code><buttonGroup spacing="false" title="COOLANT"></code>	A key group combines two keys under one header.	
	spacing	The property enables the display type of the keys to be determined: together "false" or separate "true". The default is separate.
	title	All the keys in a key group have the same header.

3.14 Activate and configure the Sidescreen window

Tag	Description	
<pre><button position="1" text="ON" icon="slsidescreenmcp_cooling.png" style="redStyle" title="Spindle"></pre>	Defines the key properties	
	position	Position of the key
	text	Labeling shown on the key
	icon	Icon of the key The icon is displayed above the text label. If there is no labeling, then the icon also takes up the labeling space.
	style	Changes the background color assigned to the key Possible values are: "redStyle" for a red key "greenStyle" for a green key The default is "defaultStyle"
<pre><onPressed address="HmiUserKeyX"/></pre>	If the key is pressed, the PLC bit is then set to 1. The value remains at 1 as long as the key remains actuated. With a brief press, the value remains at 1 for at least 300 ms.	
	address	Alias name "HmiUserKeyX" for the addressed PLC bit Values of 1 to 64 are permitted for X. HmiUserKey1 to HmiUserKey64 stand for DB1900.DBB22.0 to DB1900.DBB29.7.
<pre><active address="HmiUserLedX"/></pre>	The key is shown as active (LED animation at the top edge of the key).	
	address	Alias name of the PLC bit from which the active identifier is derived Values of 1 to 64 are permitted for X. HmiUserLed1 to HmiUserLed64 stand for DB1900.DBB5026.0 to DB1900.DBB5033.7.
<pre><disabled address= "DisableHmiUserKeyX"/></pre>	The key is shown as inactive (icon and text grayed out).	
	address	Alias name of the PLC bit from which the inactive identifier is derived. Values of 1 to 64 are permitted for X. DisableHmiUserKey1 to DisableHmiUserKey64 stand for DB1900.DBB5034.0 to DB1900.DBB5041.7.

Note

Distribute the keys over several pages if you wish to configure a larger number of keys via the alias names `HmiUserKeyX`. This will help to prevent key display errors for machine control panels with a lower screen resolution.

Store own icons

You can create your own icon files under one of the following paths:

/oem/sinumerik/hmi/ico or

/user/sinumerik/hmi/ico

Depending on the screen resolution, under the following subdirectories:

/ico1024 for HD Ready

/ico1920 for Full HD

Note

The icon file names must be the same as the names used in the XML file, e.g. `icon="slsidescreenmcp_cooling.png"`.

3.14.3.2 Create own translations of texts for the machine control panel

You can display all machine control panel texts in different languages. Translations are created in the file "slsidescreenmcp_XXX.ts".

"XXX" stands for a language code, e.g. "deu" for German, "eng" for English, and so on.

Procedure

1. Create the file "slsidescreenmcp_XXX.ts" with the desired language code in one of the following directories:
/oem/sinumerik/hmi/lng or
/user/sinumerik/hmi/lng
2. Open the language file and insert the designations accordingly.
 The following texts may be displayed in translation:
 - All headers, e.g. `title="COOLANT"`
 - All key labels e.g. `text="ON"`
3. Save the language file.
 The translations are displayed after a run-up and the language has been changed.

Adaptation options of the file "slsidescreenmcp_XXX.ts"

You can insert the translations in the `<message>` tag:

Tag	Description
<code><source>ON</source></code>	This is the language code of the original language.
<code><translation>ON</translation></code>	This is the translation.
<code><chars>8</chars></code>	This is the maximum length of the translation. The tag serves merely a note for the translator.

3.14 Activate and configure the Sidescreen window

Example extract of the file "slsidescreenmcppage_eng.ts"

```
...  
<message>  
  <source>ON</source>  
  <translation>EIN</translation>  
  <chars>8</chars>  
</message>  
...
```


3.15 Automatic save data archive

An automatic save data archive can reset your control to the status of a specific time, e.g. the status before startup. If the function is activated, the system automatically saves all control components (NC, PLC, HMI system and drive data) at regular intervals. You set the time schedule.

If the machine can no longer be operated with the existing software, it is often easier to install the automatic save data archive than to install a new software version. This means the commissioning data can be made available to the machine, e.g. if parts have to be replaced including PPU generation replacement, up to date and independent of the software version.

Requirement

To configure and read in an automatic save data, the access level "Manufacturer" is required.

Storage location and file format

- The following directories are available for the automatic save data archives:
 - CompactFlash card at: System CF-Card/**oem**/sinumerik/data/AutoSave
 - All configured logical drives (USB, network drives)
- An automatic save data archive is saved as an .asd (Auto Save Data) file.
- The system assigns an archive name:
autosavedataarc_<Date>_<Time>.asd

Note

An automatic save data archive is intended only for the machine on which the archive was created. Transferring it to another machine is not possible.

Activating the function

You can activate the data archive function on the user interface via the following display machine data:

MD9119 \$MM_ENABLE_AUTO_SAVE_DATA_ARCHIVE		Activate automatic save data archives
=1		

Using the automatic save data

You can specify the time for saving data and read in the archive again via the SINUMERIK Operate user interface.

- In the "Commissioning" operating area, switch to the "Configure automatic save data archive" display via "MENU SELECT > Commissioning > Commissioning archives > Configure data save" and set the desired time for saving the data.
- Switch to the "Commissioning" operating area and follow the "Automatic save data archive > Read in" selection to restore a data save.

Note

Message for execution or failure

SINUMERIK Operate displays a message for the scheduled execution up to 10 minutes before the start of an automatic data save operation.

If a data save schedule is active but cannot be executed, for example due to a running program, automatic archiving is executed 24 hours later. You receive error message 150413 with the scheduled time for the next attempt.

More information

You can find further information on using archives in the online help under Commissioning functions and service.

Machine and setting data

4.1 User views

Usage

In the "User Views" window you can create and adapt all machine data relevant to a particular functionality. Machine data is individually collated to simplify the user's task.

You can include additional comments when creating and/or editing user views.

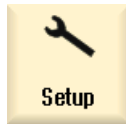
Display in the the "Parameter" operating area

Proceed as follows to show user views also in the "Parameters" operating area:

1. Select the "Start-up" operating area.
2. Press the "Machine data" softkey.
3. Press the "User views" softkey.
4. Press the "Manage view" softkey.
5. Press the "New view" softkey.
6. Activate the "Use display also under parameters / setting data" checkbox if you also wish to display the new user view in the "Parameters" → "Setting data" → "Data lists" operating area.
7. Enter a name for the user view.
8. Confirm with "OK".
9. Insert the machine data that is to be displayed in the "Parameters" operating area.

4.1.1 Creating a user view

Procedure



1. Select the "Start-up" operating area and press the "Mach. data" softkey.



2. Press the softkeys "User views", "Manage view" and



"New view".

The "New View" window opens.

3. Enter the desired view name.

Press the "OK" softkey.

If a user view already exists with the same name, you receive a safety prompt as to whether the existing view should be overwritten.



4. Press the "Enter data" softkey.

The "Insert data" selection box opens.



5. Open the selection list with the <INSERT> key, select a machine data area and press the <INPUT> key.



Select a machine data item with the cursor keys or using the "Find" softkey.



6. Press the "Insert before line" or "Insert after line" softkey to insert the selected machine data at the required position in the user view.



Note:

You can navigate in the open user view with the mouse and select a line without closing the list box.

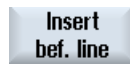


7. Press the "Back" softkey to close the list box and return to the user view you want to edit.

Your entries are automatically stored.



8. Press the "Insert text" softkey to insert any text in the open user view. The "Insert Text" input window opens.



9. Enter a text and explanatory description for the text and press the "Insert before line" or "Insert after line" softkey to accept the text.



10. Press the "Back" softkey to save your entries and return to the user view you are editing.

Selecting a particular machine data item



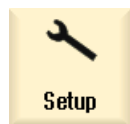
You can use the Search dialog box to look for a particular data item.

4.1.2 Editing the user view

Depending on the row you have selected, you can use the "Properties" softkey to change comments and, in the case of machine data, the data source (channel, axis, drive unit) and view the description and, if required, change.

- Machine data
Depending on the data type you can select either a fixed or a variable data source for machine data.
The entry "variable(*)" means that when you can select another data source (channel, axis, drive unit) with vertical softkeys 1- 3 (+, - and direct selection) the value displayed changes to refer to the data source currently selected.
Alternately, you can select a fixed data source so that the displayed value does not change when you select a different data source.
- Comment texts
Text and description

Procedure



1. Select the "Start-up" operating area and press the "Mach. data" softkey.



2. Press the "User views" softkey.
The "User Views" window opens.

4.1 User views



3. Press the "Edit view" softkey. Additional softkeys to edit the user view are listed.



4. Press the "Insert data" softkey if you wish to enter another data item.



5. Select a machine data item with the cursor keys.



6. Press the "Up" or "Down" softkeys to move the selected row.



- OR -



Press the "Delete line" softkey to remove the selected line from the view. The data item is removed without a prompt.

- OR -



Press the "Properties" softkey to view the texts for comments and, if required, change.

The "Properties" window is opened. For machine data, depending on the setting, the description or the machine data is displayed in the lower left-hand window.



7. Press the "Back" softkey to save your changes.

Selecting a particular machine data item



You can use the Search dialog box to look for a particular data item.



Press the "Go to start" softkey to start the search at the first entry.

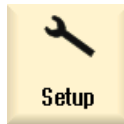


Press the "Go to end" softkey to start the search at the last entry.

Press the "Continue search" softkey if the data found during the search does not match up with what you are looking for.

4.1.3 Deleting a user view

Procedure



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



2. press the "Machine data" softkey.



3. Press the "User views" softkey.



4. Press the "Manage view" softkey.

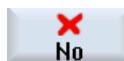


5. Select the desired view from the selection list.
6. Press the "Delete view" softkey.



With the final confirmation prompt you will be requested to confirm with either "Yes" or cancel with "No".

or



The "User views" dialog is then redisplayed.

Just like before, as an alternative, it is possible to delete a user view under "System data" under the following path: `../user/sinumerik/hmi/template/user_views`

Note

User views that are part of the system cannot be deleted.

4.2 Plain texts for machine and setting data

Application-specific and language-dependent plain texts can be configured for machine and setting data depending on the version [Index].

Displaying the plain texts

You set the display of the plain texts using the following display machine data:

MD9900 \$MM_MD_TEXT_SWITCH	Plain texts instead of MD identifier
= 1	Plain text is displayed in the table
= 0	Plain text is displayed in the dialog line

Files

File	Machine/setting data
nctea_xxx.txt	General machine/setting data
chtea_xxx.txt	Channel-specific machine/setting data
axtea_xxx.txt	Axis-specific machine/setting data
hmitea_xxx.txt	Display machine data

The specified language code must be used for "xxx", e.g. "deu" for German, "eng" for English, etc.

Ensure that the file name is written in lower case letters.

Procedure

1. Depending on the machine or setting data for which you wish to change the plain text, create the corresponding file, e.g. "nctea_xxx.txt".
2. Save the file with the UTF-8 coding in order that umlauts and special characters are correctly displayed in SINUMERIK Operate.
3. Store the file in the SINUMERIK Operate in the following directory:
 .../**user**/sinumerik/hmi/Ing or
 .../**oem**/sinumerik/hmi/Ing or
 .../**addon**/sinumerik/hmi/Ing

Note

The files are searched through in the sequence of the directories "user", "oem" and "addon". The first file that is found is evaluated. If, e.g. text files are available in the "user" and "oem" directories, the files are evaluated in the "user" directory.

Note

If a machine data item has different indices and you do not specify an index for the plain text, the same plain text appears for all the indices of the machine data item.

The parameters and indices must be sorted in an ascending order.

Example of a file with German texts

```

14510 Mein Integer-Text für PLC_MD 14510
14510[1] Mein Integer-Text für MD 14510[1]
14512[1] Mein Text für HEX-Datum 14512[1]

```

Result in the display

The plain text is displayed in the table or in the diagnostics line in the operating area "Startup", "General MD".

Machine data	Text
14510	My integer text for PLC_MD 14510
14510[1]	My integer text for MD 14510[1]
14512[1]	My text in HEX data 14512[1]

Connecting drives

5.1 Overview

Up to 21 connections to what are known as logical drives (data carriers) can be configured. These drives can be accessed in the "Program Manager" and "Commissioning" operating areas.

The following logical drives can be set up:

- USB interface
- CompactFlash card
- Network drives

Note

If a USB flash drive is set up in SINUMERIK Operate and is accessible, then the connection saved can also be used via external tools, for example Access MyMachine /P2P. You then find the drive in the tool under directory `.../user/sinumerik/data/mnt`.

5.2 Setting up drives

The "Set Up Drives" window is available in the "Start-up" operating area for configuring the softkeys in the Program Manager.

Additional information

Additional information on the procedure and configuration can be found in the Operating Manual Universal/Turning/Milling/Grinding

File

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

5.3 Setting EXTCALL

Using the EXTCALL command, a program on an external program memory can be called from a part program.

EXTCALL does not need to be disabled.

Activating the function

Processing EXTCALL calls is switched-on or switched-off using the following display machine data.

MD9106 \$MM_SERVE_EXTCALL_PROGRAMS	
= 0	HMI ignores EXTCALL instructions and selection via PLC
= 1	HMI processes EXTCALL instructions and selection via PLC (default value)
= 2	HMI processes EXTCALL instructions and ignores selection via PLC
= 3	HMI ignores EXTCALL instructions and processes selection via PLC

Note

A main program selected from an external program memory is selected again automatically after a power-on if the same program memory is still available and execution of EXTCALL calls has been activated in MD9106.

5.4 Creating a global part program memory

With the global part program memory (GDIR), a structure is provided analog to the passive file system of the NC. It is integrated in the search path of the NC for subprogram calls. If you create a GDIR, all participants in the system have access to the same logical drive. The participants can directly execute part programs from the common program memory. The GDIR so replaces or extends the part program memory of the NC.

Note

The global part program memory plays a role only for the EES function. The creation of a GDIR is, however, not essential for the EES operation.

Possible procedure

1. Declare the global part program memory in the "Creating drives" window.
2. Shift the content of the local program memory with SPF, MPF and WCS directory to an external drive, e.g. to a USB flash drive (USB interfaces X135 or X145 of the PPU).
3. Copy the directories, which have been backed up, to the global part program memory.

Additional information

Additional information about the program memory can be found in the NC Programming Programming Manual.

5.5 Program selection

5.5.1 Displaying the softkey

The softkeys on the user interface for calling the program lists are activated via general configuration machine data.

Configuring a softkey

MD51041 \$MNS_ENABLE_PROGLIST_USER	
0	No softkey
1	The "Prog. list" softkey is displayed

MD51043 \$MNS_ENABLE_PROGLIST_MANUFACT	
0	No softkey
1	The "MANUFACTURER" softkey is displayed

Precondition

To create program lists, you require the following authorization:

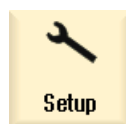
- "Program manager" operating area: Access level 3 (password: User) or
- "Start-up" operating area: Access level 1 (password: Manufacturer).

Program lists

You can create a program list by combining desired programs from the NC data, e.g. workpieces, part programs and subprograms. These programs can then be selected by the PLC for execution.

Program lists	Directory
plc_proglst_user.ppl	/user/sinumerik/hmi/plc/programlist
plc_proglst_manufacturer.ppl	/oem/sinumerik/hmi/plc/programlist

Procedure



1. Select the "Start-up" operating area if you have the authorization for access level 1 (password: Manufacturer).

- OR -



Select the "Program manager" operating area if you have the authorization for access level 3 (password: User).



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



3. Press the "Manufacturer" softkey if you want to create a program list that can be used for all machines of a certain series.



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

5. Press the "Select program" softkey.

The "Programs" window opens. The data tree of the NC memory with the directories "Workpieces", "Part programs" and "Subprograms" is displayed.



6. 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).

If required, the extension (.MPF) is added.

Note:

A syntax check is not performed.

With workpieces, make sure that there is a main program or a job list with the same name in the workpiece itself.



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

- OR -



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

Configuring the network

Overview

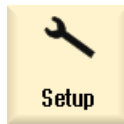
If you wish to perform network settings in SINUMERIK Operate, you have the following options:

- The Start-up operating area allows you to set up all of the basic configurations for the plant and company networks (see Section "Displaying the network overview (Page 82)").
- The Diagnostics operating area gives you an effective overview of all available network adapters and allows you to assign and check settings for the company network or the DHCP server, for example (see Section "TCP/IP diagnostics and configuration (Page 87)").

6.1 Displaying the network overview

All of the available network interfaces and their settings are listed in the "Overview of the network settings" window. For example, the IP addresses, subnet masks and MAC addresses for the company network are displayed here and can be processed.

Procedure



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



2. Press the menu forward key.



3. Press the "Network" softkey.



The "Overview of the Network Settings" window opens.



4. Press the "Change" softkey to edit settings.

See also

Settings of the factory network (Page 83)

Saving network settings (Page 85)

6.2 Settings of the factory network

The network interface, active protocols and firewall exceptions of the company network are configured in the "Settings of the company network" window.

Setting	Description
Gateway	If this value is not empty, the host specified there is used as the default gateway, which means that all IP packets that cannot be directly assigned are sent there for routing.
DNS server	If DNS name servers are specified here (maximum three), then they must be used to resolve symbolic host names, i.e. at most of the points where an IP address is expected, a computer name can also be used instead. The name server setting is also sent to its DHCP clients (TCU, PG) via the DHCP server of the NCU, so that they can also work with symbolic names.
Time server	Here you can specify up to three NTP servers (UDP/123) that can be used by the NTPD on the NCU for time synchronization.
Host name	Here you can define a name for the local host. This manually assigned name takes priority over all other names. The host name is determined in the following order: • This entry (identical to the host name from the basesys.ini file) • A name received from the DHCP server (reverse DNS lookup, i.e. which name belongs to the received IP address) • Default name ("NONAME_...") Because the host name is also used as DNS name, it must satisfy the corresponding requirements of the RFC: • ASCII letters (a-z, A-Z), digits (0-9) and '-' • maximum 63 characters
Active protocols	Specifies which protocols are to be used in the company network. DCP (Discovery and Basic Configuration Protocol) and LLDP (Link Layer Discovery Protocol) are available for selection.
Firewall exceptions	You can specify ports here that are to be enabled in the external network interface.
• S7 communication (TCP/102)	Port for the S7 communication
• VNC access (TCP/5900)	Port for the VNC access
• Additional ports	Any further ports can be enabled here. The entry starts with the protocol name (TCP or UDP), followed by a slash '/' and the port number of the service to be enabled. Several entries are separated by a space. Example (enabling of the VNC servers): TCP/5900 TCP/5904 TCP/5905

NOTICE

Security risk

Note that opening the ports in the firewall can represent a security risk. Only enable the ports that you actually require.

Note that every accessible service can have security gaps.

Procedure



1. The network overview is open.



2. Press the "Company network" softkey.
The "Settings of the Company Network" window opens.



3. Press the "Change" softkey to edit settings.

6.3 Saving network settings

You have the option of saving the NCU settings as file.

In window "Save network settings: Select archive", select a target folder. The network configuration is saved there, or an existing file is selected, which is overwritten with the new settings.

Procedure



1. The network overview is open and you have made network settings.
2. Press the "Save NCU settings" softkey.
Window "Save network settings: Select archive" is opened.
3. Select the folder in which the network configuration is be stored. If a file is already available, it will be overwritten.
- OR -
Press the "New directory" softkey to create a new directory.
- OR -
Press the "Find" softkey, for example, to find the archive location of the network configuration file.
4. Press the "Find next" softkey if the file found during the search does not correspond to the required configuration file.
5. Press the "Cancel" softkey to reject the settings.

- OR -
Press the "OK" softkey to save the network configuration.

6.4 Station-related network diagnostics

With the aid of the station-related network diagnostics, you can find and examine faulty, incorrectly parameterized or unconnected network and bus interfaces.

The network/bus interfaces of the SINUMERIK 828D are:

- Company network (X130)
- Service (X127)
- PROFINET (PLC I/O interface PN1, PN2)
- PROFIBUS (integrated)

6.4.1 Displaying the PPU network interfaces

The network interfaces, e.g. company network X130 and service X127, and their availability are displayed in a tree topology in the "TCP/IP Diagnostics" window.

Detailed view

You can display detailed information about a selected interface, e.g. IP address, DNS name, etc.

Procedure



1. Select the "Diagnostics" operating area.



2. Press the menu forward key and the "Bus TCP/IP" softkey.



3. Press the "TCP/IP diagnostics" softkey.
The "TCP/IP Diagnostics" window opens and displays the current availability of the network connections.
4. Position the cursor on the interface for which you wish to display the detailed information.



5. Press the "Details" softkey to display all available parameters of the configured network connections.



6. Press the "Details" softkey to hide the detailed display.



7. Press the "Return" softkey in order to exit the "TCP/IP Diagnostics" window and return to the "PROFINET/PROFIBUS Diagnostics" window.

6.4.2 TCP/IP diagnostics and configuration

The configuration of the network interfaces is displayed in the "TCP/IP Configuration" window.

Service

You use the service interface X127 for start-up and diagnostics.

A PG/PC communicates with the control via this interface as peer-to-peer. The IP address 192.168.215.1 is permanently set for the service interface.

Company network

The company network is used, for example, to access the network drives.

A PPU is connected to the company network via Ethernet interface X130.

Availability of the network connections

Network adapter connection



White Network cable inserted



Red Network cable not inserted

Availability

The availability describes the percentage of faulty packages compared to all the sent and received packages.

Problems in the company network or the settling time during the start-up can cause fluctuations in the availability.



Green Greater than 95%



Yellow 50 - 95%



Red Less than 50%

Details of the network connections

- **Computer name**
The computer name of the control.
- **DNS name**
The name of the network adapter as stored in the DNS (Domain Name Service) server.
- **MAC address**
The physical address of the network adapter.

- **Address type**

Information about the configuration of the network adapter in the configuration file "basesys.ini":

- **DHCP:** DHCP is active for this network adapter.
The information behind the hyphen "-" specifies the DHCP mode of operation:
Off - DHCP is deactivated for this adapter
Client - a DHCP client that receives an IP address and further data from a server, is running on the interface
Server - the SINUMERIK control provides a DHCP server at this interface that supplies clients on this network with IPs
If no changes have been made for the network adapters in the configuration file, then "Default" is also output.
- **Manual**
The "Change" softkey allows manual settings, see "Changeable parameters" and "Address types" sections.

Note

In the "Change" mode, either "Manual" or "DHCP" can be selected (only for company network, X130).

- **IP address assigned**

The current IP address of the network adapter.

- **Subnet mask assigned**

The subnet mask of the network adapter.

- **DHCP server**

The IP address of the DHCP server (for address type "DHCP").

- **State, DHCP server**

State of the synchronous DHCP server:

- **Active:** The current NCU is active in the group of NCUs in the "Synced Server" mode and transfers the IP addresses.
- **Standby:** The NCU is not active as DHCP server. If the active DHCP server fails, this NCU can take over the task of the DHCP servers.

- **Mode, DHCP synchronous**

Priority of the synchronous DHCP server:
Low, high, master

- **DNS server 1, DNS server 2**

The IP address of the DNS server.

- **Gateway**

The IP address of the host that is to be assigned as gateway.

Note

All information that is not available is marked in the relevant table line with a hyphen "-".

Modifiable parameters

The following parameters of the company network interface (X130) can be set:

- Address type
- IP address
- Subnet mask
- DNS server 1 and 2
- Gateway (applies for service interface and company network)

Address types

- DHCP
The values for the settable parameters IP address, subnet mask, DNS server 1, DNS server 2 and gateway are defined automatically by the DHCP server, in accordance with the entry in the "basesys.ini".
- Manual
The values for the set parameters IP address, subnet mask, DNS server 1, DNS server 2 and gateway can be specified manually. The values are stored in "basesys.ini".

Procedure



1. Select the "Diagnostics" operating area.



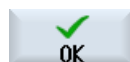
2. Press the menu forward key and the "Bus TCP/IP" softkey.



3. Press the "TCP/IP diagnostics" and TCP/IP config." softkeys. The "TCP/IP Configuration" window opens.



4. Press the "Modify" softkey to change specific parameters. The fields that can be edited have a white background.



5. Enter the desired changes and press the "OK" softkey to confirm the entries.



6. Press the "New display" softkey to refresh the display.

6.4.3 Ethernet network diagnostics

6.4.3.1 Diagnostics of the network and DHCP server

Press the "Network diagnostics" softkey to display data and availability of the network nodes. As result, the diagnostics provide, among other things, the following information:

- Component identification of the hardware
- TCP/IP address / MAC address
- DHCP server detection
- Status of the nodes

This gives you the option to test whether the current topology matches the reference topology.

Procedure



1. Select the "Diagnostics" operating area.



2. Press the menu forward key and the "Bus TCP/IP" softkey.



3. Press the "TCP/IP diagnostics" softkey.

The "TCP/IP Diagnostics" window opens and displays the current availability of the network connections.



4. Press the "Network diagnostics" softkey to browse your networks.

The "Network Status" window opens and displays the addresses of the PN system network and the company network (X130).

The corresponding fields in the window remain empty for non-accessible networks.



5. Press the "New display" softkey to refresh the display.

- OR -



Press the "DHCP server" softkey to fetch information and query the status of the DHCP server.

The "DHCP Server" window opens.

For each active DHCP server, the status, the role (priority), the MAC address, the TCU version and the lease version are displayed in tabular form in the upper part of the window. The required IP with the associated address area appears in the lower part of the window.

Any differing data is displayed in red.

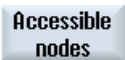
6.4.3.2 Accessible nodes

Pressing the "Accessible nodes" softkey creates and displays the list of nodes accessible via PROFINET. The list content can be compared as current topology with a reference topology.

Note

Although handheld terminals are not determined for the list of accessible nodes, they can be added as additional nodes to the topology.

Procedure

- | | | |
|---|----|---|
|  | 1. | The "Network Status" window opens. |
|  | 2. | Press the "Accessible nodes" softkey.
The "Accessible Nodes" window opens.
The name, IP address, HW ID/type, status and location are displayed in tabular form for each node. |
|  | 3. | Press the "Sort" softkey. |
|  | 4. | Press the "By name", "By IP", "By type" or "By location" softkeys to sort the display. |
|  | | |
|  | | |
|  | | |
| | | - OR - |
|  | 3. | If further information is provided for a selected node, press the "Details" softkey.
The "Details" window opens and the additional information is displayed. |
|  | 4. | Press the "Zoom +" or "Zoom -" softkey until the desired display size is reached. |
|  | | |

Additional nodes



3. Press the "Additional nodes" softkey to search for nodes that are not listed under "Accessible nodes".



The "Search for Additional Nodes" window opens. If necessary, change the address range and confirm with "OK".

The found nodes are appended in the table; the HW-Id/type, Location, Contact and Description fields can then be edited and, by activating the checkbox, added to the first column, and, with a subsequent save, added to the reference topology.

Comparing



3. Press the "Compare" softkey to compare the current list of nodes either with the previously saved manufacturer topology, a reference topology or a saved topology in the .xml format.



The "Compare With" window opens. Position the cursor on the desired option box and confirm with "OK".

Deviations during the comparison are identified by being color-highlighted:

- Red: Missing or non-accessible components compared with the reference topology.
- Pink: Components that deviate in their attribute values compared with the reference topology.
- Blue: New components not contained in the reference topology.

Saving



3. Press the "Save" softkey to save the current topology.

The "Save As" window opens.



4. Position the cursor on the desired option box, specify a file name and possibly a comment, and confirm with "OK".

Note

Manufacturer topology and reference topology can be created with access level 3 (user) or higher and are always saved on the active DHCP server.

Default storage locations:

- Current topology (.xml or .csv):
The following storage locations are offered depending on the configuration:
 - Network topology (storage location: HMI data / data backups / network topology)
 - User CF
 - Network drives
 - USB

In CSV format, ";" is preset as a separator; it can be changed in sldgconfig.xml

- Reference topology:
/user/common/tcu/\$soll/\$soll.xml
- Manufacturer topology:
/user/common/tcu/\$soll/\$oem_soll.xml

Customizing the "Machine" operating area

Automatic changeover

The circumstances under which an automatic changeover is made into the "Machine" operating area are set in MD51040 \$MNS_SWITCH_TO_MACHINE_MASK.

MD51040 \$MNS_SWITCH_TO_MACHINE_MASK		0 (default setting)
Bit 0 = 1	For program selection in the Program Manager, an automatic change is not made into the Machine operating area.	
Bit 1 = 1	For program selection in the Program Manager, an automatic change is not made into the Machine operating area. When changing over the operating mode via the machine control panel, an automatic change is not made into the Machine operating area.	
Bit 2 = 1	For program selection in the Program operating area, an automatic change is not made into the Machine operating area.	
Bit 3 = 1	When selecting a program/executing a program in the Program operating area, the block search is not automatically started.	

7.1 Setting the font size of the actual value display

Setting font size

The actual values for the machine coordinate system or workpiece coordinate system can be displayed in two different font sizes in the "Machine" operating area.

Using a channel machine data item, you can set how many axes should be displayed in a larger font.

MD52010 \$MCS_DISP_NUM_AXIS_BIG_FONT	Number of actual values with a large font
= 3	Number of actual values with a larger font (default setting)

If the number of axes changes, e.g. as a result of transformations, then the following channel-specific machine data is relevant for the representation pattern of the font sizes:

MD52011 \$MCS_ADJUST_NUM_AXIS_BIG_FONT	The number of actual values with large font is dynamically adapted to the number of geometry axes
= 0	Only MD52010 is valid. The number is therefore fixed.
= 1	Only the geometry axes are displayed using a large font. MD52010 is ignored.
= 2	The number of geometry axes and content MD52010 are displayed using a large font.

7.2 Inserting a user-specific logo

For the traditional SINUMERIK Operate display, the Siemens logo is visible in the "Machine" operating area in the header.

For the SINUMERIK Operate displays of the latest generation, the Siemens logo is permanently displayed in the header.

You can replace the Siemens logo with your own logo.

OEM-specific logo in traditional display

Procedure:

1. Name your own logo "logo.png".

Note

The file name must not contain any lower case letters.

2. Save your own logo depending on the bitmap size in one of the directories: **/oem/sinumerik/hmi/ico** or **/user/sinumerik/hmi/ico**

Directory	Bitmap size
/lco640	122 x 19
/lco800	160 x 24
/lco1024	199 x 30

OEM-specific logo of displays Skin 1 and 2

Procedure:

1. Depending on the screen resolution, name your own logo
"company_label_XXXXxYYYY.png".
Placeholder XXXXxYYYY corresponds to the screen resolution, for example
"company_label_1024x768.png".
2. Save your own logo in one of the directories:
/oem/sinumerik/hmi/skins/skin1/ico/ico640
/user/sinumerik/hmi/skins/skin1/ico/ico640
or
/oem/sinumerik/hmi/skins/skin2/ico/ico640
/user/sinumerik/hmi/skins/skin2/ico/ico640

Note

Display error

Remove your own logo "logo.png" from the directories /oem or /user when the displays of the latest generation are used.

Your own logo must not exceed the header height in relation to the screen resolution used.

7.2 Inserting a user-specific logo

The following is allowed:

Screen resolution	Maximum bitmap height
640 x 480	32
760 x 505	32
800 x 480	32
800 x 600	40
1080 x 810	53
1024 x 768	51
1024 x 1014	51
1280 x 768	51
1280 x 800	51
1280 x 1024	53
1300 x 768	51
1366 x 768	51
1440 x 1080	72
1454 x 1080	70
1854 x 1014	51
1854 x 1080	70
1920 x 1080	70

7.3 Configuring the display of the G-code groups

The "G-functions" pane is visible in the "Machine" operating area.

Up to a maximum of 16 G-code groups are displayed in this pane. Configure the "slmagcodeconfig.ini" file accordingly to determine which G-code groups should be displayed, and where.

Configuring G-code groups

Section	Meaning
Selected G groups	This area is intended for operation of the controller in Siemens mode.
Selected G groups ISO mode	This area is intended for operation of the controller in ISO mode.

Enter the G-code group and position as follows:

SelectedGGroupN = M	
N	Position where G-code group M will be displayed. If you want to leave a position empty, omit the number (M), e.g. "SelectedGGroup1 = "
M	The number of the G-code group to be displayed at position N

Structure of the file

```
[Selected G-Groups]
SelectedGGroup1 = 1
SelectedGGroup2 = 2
SelectedGGroup3 = 3
SelectedGGroup4 = 4
SelectedGGroup5 = 5
SelectedGGroup6 = 6
SelectedGGroup7 = 7
SelectedGGroup8 = 8
SelectedGGroup9 = 9
SelectedGGroup10 = 11
SelectedGGroup12 = 12
SelectedGGroup13 = 13
SelectedGGroup14 = 14
SelectedGGroup15 = 15
SelectedGGroup16 = 16
[Selected G-Groups ISO-Mode]
... (as with section for Siemens)
```

Procedure

1. Copy the "slmagcodeconfig.ini" file from the folder **/siemens/sinumerik/hmi/template/cfg**.
2. Place the file in the following directory: **/oem/sinumerik/hmi/cfg** or **/user/sinumerik/hmi/cfg**.
As soon as the file is located in the user-specific directory, the entries for this file take precedence over the Siemens file. If an entry is missing in a user-specific file, the corresponding entry from the Siemens file is used instead.
3. Open the file and enter the number of the G code in question at the required position
"SelectedGGroup1 = **5**".
I.e. the 5th G code group will be displayed at the 1st position.

7.4 Configuring the channel operational message

Overview

The channel operational messages are displayed in the third line of the header in the "Machine" operating area. There are two types of channel operational messages:



1. Channel operational messages that require an operator action to continue a program, e.g. feed enable missing, M01 active.

The channel operational messages are identified by exclamation marks.



2. Channel operational messages that do not require an operator action to continue a program. A stop condition is active for a length of time (>1 s) and is interrupted by the program processing, e.g. dwell time active, waiting for tool change.

The channel operational message is identified by a clock icon and disappears after a certain time.

File

The settings for this are specified in the "slmahdconfig.ini" file.

Configuring the channel operational message

Section	Meaning
CondStopIcons	Specification of an icon that is to be displayed with a certain channel operational message. If no icon is specified for a certain channel operational message, then the icon specified for "DefaultIcon" is displayed.
CondDelayTime	Specification of how long (in milliseconds) a certain message must be present before it is displayed. • If no time is specified for a certain channel operational message, then the time specified at "DefaultDelayTime" is used. • If the channel operational message is to be displayed immediately, "0" must be specified as time.

Procedure

1. Save your own icon files under the following path in one of the intended subdirectories:

/oem/sinumerik/hmi/ico or

/user/sinumerik/hmi/ico

Depending on the screen resolution, there are the following subdirectories:

Directory	PNG image size (pixels)
/lco640	16 x 16
/lco800	17 x 17
/lco1024	27 x 27

2. Copy the sample file "slmahdconfig.ini" from the directory **/siemens/sinumerik/hmi/template/cfg**.

7.4 Configuring the channel operational message

3. Place the file in the following directory:
 /oem/sinumerik/hmi/cfg or
 /user/sinumerik/hmi/cfg
4. Open the file and make the required settings:
 - Specify an individual icon by changing the file name of the icon for channel operational messages.
 - Specify a different period by changing the time value for the channel operational message.
5. Save the settings.
 The configurations are activated after booting.

Example of file "slmahdconfig.ini"

Configuration

The following default messages are intended when technical faults occur:

```
[CondStopIcons]
DefaultIcon = condwait.png
1 = condstop.png      ; No NC ready
2 = condstop.png      ; No mode group ready
3 = condstop.png      ; Emergency stop active
4 = condstop.png      ; Alarm with stop active
...
26 = condwait.png     ; Waiting for position control
...
30 = condwait.png     ; Waiting for safe operation
31 = condwait.png     ; No channel ready
...
46 = condwait.png     ; Rapid retraction started
...

[CondDelayTime]
DefaultDelayTime=1000
1 = 0                 ; No NC ready
2 = 0                 ; No mode group ready
3 = 0                 ; Emergency stop active
4 = 0                 ; Alarm with stop active
...
```

7.5 Deactivating program test

There are several program test options to test and run-in a part program that are executed using the "Program Test" function.

This function is active as standard. You can select the program test with the "PRT" check box in the "Program control" window in the "Machine" operating area.

Deactivating the program test

You can deactivate the program test using the following configuration machine data. "PRT" is no longer displayed in the status line.

MD51039 \$MNS_PROGRAM_CONTROL_MODE_MASK		Options for machine - program control
Bit 0 = 0	Function program test not available	

Additional information

Additional information about the test options is provided in the Basic Functions Function Manual.

7.6 Display articulated joint position STAT and rotary axis position TU

Display STAT and TU in the "MCS" window

If you want to display the joint position of the machine with the value STAT and the position of the rotary axes with the value TU in the "Machine" operating area for Cartesian point-to-point or PTP traversing, set the following channel-specific configuration machine data $\neq 0$:

MD52032 \$MCS_STAT_DISPLAY_BASE		Number base for display of the joint position STAT
= 0	No display, value from MD51032 effective (default)	
= 2	Display as binary value	
=10	Display as decimal value	
= 16	Display as hexadecimal value	

MD52033 \$MCS_TU_DISPLAY_BASE		Number base, display of the rotary axes TU position
= 0	No display, value from MD51033 effective (default)	
= 2	Display as binary value	
=10	Display as decimal value	
= 16	Display as hexadecimal value	

The settings in MD52032 and MD52033 also work for Teach In.

Additional information

You can find additional information about STAT and TU in:

- Chapter "Activating the "Teach In" function (Page 105)"
- Transformations Function Manual

7.7 Activating the "Teach In" function

Activate Teach In

The Teach In function is activated in part via the general configuration machine data.

Settings

MD51034 \$MNS_TEACH_MODE	
Bit 0 = 1	The "Progr. teach in" softkey is displayed. The approached position is accepted in the program with the "Accept" softkey. (default)
Bit 1 = 1	The acceptance of the teach-in block can be blocked by the PLC. DB1700.DBX1000.0 = 0 Block is accepted DB1700.DBX1000.0 = 1 Block is not accepted

Access to the function still depends on access rights.

MD51045 \$MNS_ACCESS_TEACH_IN		Protection level TEACH IN
= 7	The protection level for the execution of traversing movements to MDA buffer / part program for TEACH IN is specified. (default)	

MD51050 \$MNS_ACCESS_WRITE_PROGRAM		Protection level write part program
= 7	The protection level for the writing of part programs is specified. (default)	

Set the channel-specific machine data:

MD20700 \$MC_REFP_NC_START_LOCK		NC-Start disable without reference point
= 0		

MD52207 \$MCS_AXIS_USAGE_ATTRIB		Attributes of the axes [n] channel axis number
Bit 14	Release spindle in axis mode for Teach In	
=1	a spindle that is in axis mode should also be teachable	

Activating Teach In

1. Set the machine data and initiate a reset.
2. Now switch to the AUTO and MDA operating mode.
3. Select the TEACH IN machine function.
4. Press the "Teach prog." softkey.

Teach In not active

If the "Prog. teach in" softkey is not active, the reasons can be as follows:

- The program is not in the stop, reset or interrupt state.
- Not all of the axes that require referencing are referenced.

Accept STAT and TU

If you want to import the joint position with the value STAT of the machine and the position of the rotary axes with the value TU into the traversing block for Teach In, click on the checkboxes STAT and TU in the "Settings" window under "Prog. teach in".

In order that the checkboxes are displayed, set the following general configuration machine data $\neq 0$:

MD51032 \$MNS_STAT_DISPLAY_BASE		Number base for display of the joint position STAT
= 0	no display	
= 2	Display as binary value (default)	
= 10	Display as decimal value	
= 16	Display as hexadecimal value	

MD51033 \$MNS_TU_DISPLAY_BASE		Number base, display of the rotary axes TU position
= 0	no display	
= 2	Display as binary value (default)	
= 10	Display as decimal value	
= 16	Display as hexadecimal value	

7.8 Block search

7.8.1 Activating block search mode

Block search mode settings

The following machine data can be used to set the possible variants for the block search with G-code programming:

MD51028 \$MNS_BLOCK_SEARCH_MODE_MASK	Bit mask for the available search modes
Bit 0 = 1	Block search with calculation, without approach
Bit 1 = 1	Block search with calculation, with approach
Bit 4 = 1	Block search without calculation
Bit 5 = 1	Block search with test run

The following machine data can be used to set the possible variants for the block search in a JobShop program (ShopMill, ShopTurn):

MD51024 \$MNS_BLOCK_SEARCH_MODE_MASK_JS	Bit mask for the available search modes (ShopMill, ShopTurn, single-channel)
Bit 0 = 1	Block search with calculation, without approach (default setting)
Bit 1 = 1	Block search with calculation, with approach Notice: In a ShopMill/ShopTurn program, the search is performed only on G code-blocks.
Bit 3 = 1	Skip EXTCALL programs
Bit 5 = 1	Block search with test run

MD52212 \$MCS_FUNCTION_MASK_Tech	General function screen for all technologies
Bit 3	Block search cycle for ShopMill / ShopTurn
= 0	The E__ASUB or F__S__ASUB cycles are not called in the block search cycle PROG_EVENT.SPF
= 1	The E__ASUB (under ShopMill) or F__S__ASUB (under ShopTurn) cycles are called in the block search cycle PROG_EVENT.SPF
Bit 5	Call block search function for SERUPRO.
= 0	CYCLE207 is not called in the block search cycle PROG_EVENT.SPF.
= 1	The cycle for SERUPRO (CYCLE207) is called in the block search cycle PROG_EVENT.SPF (for block search with program test - SERUPRO). CYCLE207 provides for the output of the collected machine commands after the block search with program test.

Note

The "Block search mode" softkey is only displayed, if, in the machine data MD51024 or MD51028, more than one version for the block search has been defined; for example "Block search with calculation without approach" and "Block search with calculation with approach".

When so doing, take into account that the search versions set active in the "Block search mode" window are also referred to the program.

7.8.2 Accelerated block search for execution from external

The accelerated block search function allows you to run the part program forward until you reach the block you require.

Settings

If you activate the "Skip EXCALL" function, then you must set the following general configuration machine data:

MD51028 \$MNS_BLOCK_SEARCH_MODE_MASK	Bit mask for the available search modes
Bit 3 = 1 (33H to 3BH)	

For an external main program, a block search without calculation is always a fast block search. Only the blocks from the search position are loaded.

7.9 Manual machine

General information

"Manual machine" allows you to work with large cycles in manual mode without having to write programs especially for this purpose.



Software option

You require the following software option in order to use this function:
"ShopMill/ShopTurn"

Activation

MD52212 \$MCS_FUNCTION_MASK_TECH	General function mask for all technologies
Bit 8	Manual machine (ShopMill/ShopTurn)

Enables

MD52214 \$MCS_FUNCTION_MASK_MILL	Function mask for milling
Bit 5	Enable spindle control, tool spindle
= 0	via the NC/PLC interface
= 1	via the user interface
Bit 6	Enable spindle control, rotating spindle
= 0	via the NC/PLC interface
= 1	via the user interface

MD52218 \$MCS_FUNCTION_MASK_TURN	Turning function screen
Bit 3	Enable spindle control, main spindle
= 0	via the NC/PLC interface
= 1	via the user interface
Bit 4	Enable spindle control, counterspindle
= 0	via the NC/PLC interface
= 1	via the user interface
Bit 5	Enable spindle control, tool spindle
= 0	via the NC/PLC interface
= 1	via the user interface

SD43200 \$SA_SPIND_S	Speed for spindle start using NC/PLC interface signal; value outputs using the operating software
SD43202 \$SA_SPIND_CONSTCUT_S	Cutting speed for spindle start using NC/PLC interface signal; value outputs using the operating software
SD43206 \$SA_SPIND_SPEED_TYPE	Spindle speed pipe for spindle start using NC/PLC interface signal; value outputs using the operating software

Additional parameter assignments

We recommend the following additional parameter assignments:

Handwheel	
MD11346 \$MN_HANDWH_TRUE_DISTANCE = 3	Handwheel path or velocity specification
MD11602 \$MN_ASUP_START_MASK bit 0, 3 = 1	Ignore stop conditions for ASUB
MD20150 \$MC_G_CODE_RESET_VALUE [7] = 2	Delete position of the G groups; run-up G54 active
MD20624 \$MC_HANDWH_CHAN_STOP_COND = 0xFFFF	Definition of the behavior of traveling with handwheel, channel-specific
MD32084 HANDWH_STOP_COND = 0x7FF	Behavior, handwheel travel
MD52212 FUNCTION_MASK_TECH Bit 9 = 1	Function screen across technologies; work offset via softkey

Note

If, in machine data MD52212 FUNCTION_MASK_TECH bit 9 is set, the "WO selection" softkey is displayed in the "Work Offset – G54 ... G599" window.

Working area limits	
MD28600 \$MC_MM_NUM_WORKAREA_CS_GROUPS > 0	Number of coordinate system-specific working area limits
MD20150 \$MC_GCODE_RESET_VALUES[59] = 2	Delete position of the G groups; activate working area limits of the group

Note

If \$MC_GCODE_RESET_VALUES[59] = 2 is set in the MD20150 machine data, the "Limit stops" softkey is displayed in the "Manual Machine" window. This requires that \$MC_MM_NUM_WORKAREA_CS_GROUPS also has the value 1 for MD28600.

Feedrate	
SD42600 \$SC_JOG_FEED_PER_REV_SOURCE = -3	Revolutional feedrate control in JOG mode
SD43300 \$SA_ASSIGN_FEED_PER_REV_SOURCE [X axis] = -3	Revolutional feedrate for positioning axis X
SD43300 \$SA_ASSIGN_FEED_PER_REV_SOURCE [Z axis] = -3	Revolutional feedrate for positioning axis Z

Note

Ensure that the axis-specific setting data SD43300 SA_ASSIGN_FEED_PER_REV_SOURCE are set the same.

Additional information

Additional information about working area limitation and working with the manual machine is provided in the:

- Turning/Milling Operating Manual
- NC Programming Programming Manual

7.10 Multi-channel support

In the "Machine" operating area, a dual channel view is possible for any two channels (2 from n) of a mode group.

Using the multi-channel editor, for a multi-channel machine, you can open, change or generate programs simultaneously that are independent of one another from a time perspective. In this case, the programs involved are displayed next to one another in the editor.

Further, you have the option of simulating the programs before they are actually processed at your control.

In addition, you can align the programs according to synchronization points.

Requirement

- For multi-channel machine, the following data must be the same for all channels:
 - Measurement unit
 - Zero offset (e.g. G54)
 - Z value of the zero offset (optional)
 - Blank
 - Speed limitation
- Further, the following JobShop-specific data must be the same for all channels:
 - Retraction plane (one each for the main and counterspindle)
 - Tailstock (one each for the main and counterspindle)
 - Tool change point (one each for the main and counterspindle)
 - Safety clearance
 - Climbing and conventional

Creating a multi-channel program

For multi-channel machining, you require a job list with the following data:

- Program names
- Program type
- G code
- ShopTurn program
- Channel assignment



Software option

You require the "ShopMill/ShopTurn" option to generate and edit ShopTurn/ShopMill programs.

After you have created a job list, in the "multi-channel data" parameter screen, you can combine the data from the G code and JobShop, which is then saved as cycle call in the job list. The "multi-

channel data" parameter screen is displayed as soon as at least one ShopTurn program is contained in a job list.

Only programming G code programs

If the job list only contains G code programs, then you can deactivate the "multi-channel data" function. To do this, set the following machine data:

MD51228 \$MNS_FUNCTION_MASK_Tech		Cross-technology function screen
Bit 0	G code programming without multi-channel data	
= 1	Multi-channel data is not listed for job lists that only contain G code programs. The "multi-channel data" softkey is not displayed.	

7.11 User status display (OEM)

Function

User icons can be displayed in the Machine operating area in the second line in the header. In this case, the program name is displayed in the right-hand field of the third line in the header. Active hold conditions hide the program name.

SIEMENS					SINUMERIK OPERATE		
1	2	3	4	5			
Reset					MRD NC/MPF/CIRCULARITY_MEASURE_01		
Machine		Position [mm]				Feed/over	
MX1		0.000				0.000 mm	

The display of the user icons is controlled via PLC bits. The icons are provided in the form of PNG files, and stored in the file system under `/user/sinumerik/hmi/ico/ico640`. **Up to 16 positions are available in the header** to display the user icons.

Note

Display error

The user icons must not exceed the header height in relation to the screen resolution used. A PNG screen height of 30 pixels is appropriate.

Configuration

The user icons are configured in the `slmahdconfig.ini` file in Section `[UserIcons]`. This file is saved in the `/user/sinumerik/hmi/cfg/` directory.

The following data is required:

- The start address of the PLC double word with the bits to control the display of the user icons.
- For each bit used, the file name of the icon and the position at which the icon should be displayed.

Example of file "slmahdconfig.ini"

```
[UserIcons]
USER_ICON_BASE=DB9059.DBB0
; set the start byte of the plc double word that displays the icons
(DB9059.DBD0)
UI_0 = icon1.png,1
; show icon1.png on position 1 if bit 0 (DB9059.DBX0.0) is set
UI_1 = icon2.png,2
; show icon2.png on position 2 if bit 1 (DB9059.DBX0.1) is set
UI_2 = icon3.png,3
; show icon3.png on position 3 if bit 2 (DB9059.DBX0.2) is set
```

```
UI_3 = icon4.png, 4  
; show icon4.png on position 4 if bit 3 (DB9059.DBX0.3) is set  
UI_4 = icon5.png, 5  
; show icon5.png on position 5 if bit 4 (DB9059.DBX0.4) is set
```

Superimposition

With a **non-unique** selection (bits for multiple identifiers whose image is to be displayed at the same position):

The image with the lower identifier number is superimposed with the image with the higher identifier number for the same position. Non-unique selection can also be used with less than 16 active selection bits. The PLC can deliberately use this superimposition to overlay less important displays (with lower identifier numbers) with important displays (high identifier numbers).

Possible applications

With a **unique** selection, a maximum of 16 bits are set simultaneously in the 4 selection bytes for each of the different positions.

- 32 alternative images for a position, each selected with a different bit (currently only one active selection bit)
- 2 alternative images for each of the permissible 16 positions, each position requires 2 of the 32 selection bits (max. 16 active selection bits at any one time)
- Combination thereof with a maximum total of 32 images for the maximum 16 positions (max 16 active selection bits at any one time)

7.12 Configuring the offset display

Configuring the screen display

Overrides resulting from handwheel offsets (DRF) or tool length corrections from the value of system variable \$AA_TOFF can be directly displayed on the screen.

The screen display for the override is configured using the following channel-specific machine data:

MD52210 \$MCS_FUNCTION_MASK_DISP		Function screen display
Bit 14	Display DRF offset under Machine	
Bit 15	Display tool offset (\$AA_TOFF) under Machine	

7.13 Activate machining time recording

If you want to record and store the machining time in Automatic mode, you can make the settings on the user interface.

Additional information

Additional information about the setting is provided in the
Universal/Turning/Milling/Grinding Operating Manual

Activating the time recording

To use the function, the following channel machine data must be so set in all channels, even if the machining time recording is not activated for all channels:

MD18370 \$MN_MM_PROTOC_NUM_FILES[1]	Maximum number of log files
= 2	For OEM applications

MD18371 \$MN_MM_PROTOC_NUM_ETPD_STD_LIST[1]	Number of standard ETPD data lists
= 6	Number of standard data lists in the OPI module ETPD that are needed for Trace system functions.

MD28300 \$MC_MM_PROTOC_USER_ACTIVE[1]	Activate logging for one user
= 1	For system functions

MD28302 \$MC_MM_PROTOC_NUM_ETP_STD_TYP[1]	Number of ETP standard event types
= 5	Number of standard event types in the OPI module ETP that are needed

Note

Machine data MD18370, MD18371 and MD28302 are still valid for recording machining time when they are assigned higher values.

7.13 Activate machining time recording

Simulation and simultaneous recording

8.1 Simulation overview



Software option

You require the "3D simulation 1 (finished part)" option for the 3D representation of the simulation.

Simulation is supported with its own program interpreter (SNCK) and a separate simulation-data environment in SINUMERIK Operate. The SNCK considers the complete syntax of the SINUMERIK control family, including the possibility of incorporating special user options on the machine by comparing data with the NC environment. The simulation data can be matched statically as required with the NC environment (initialization data, macros, user data, tool data, machining cycles) or also dynamically when tool data or machining cycles are changed.

Machining simulations, with emphasis on the drilling, milling, turning and grinding technologies, can be performed in the workpiece coordinate system for certain machine kinematics on the user interface of the controller:

- The simulation of the finished part is performed with the real NC data.
- The NC data is automatically compared at each change of the part program.
- The simulation allows a dynamic representation of the machining, even for active 5-axis transformation (TRAORI) and swiveled planes.
- Turning on milling machines.
- Simulation of a machining channel for conventional milling machines.
- Simulation of up to four machining channels for turning machines with B axis.
- Intelligent determination of the block times and the program execution time.
- Very fast graphical representation through the continuous refinement of the workpiece.
- Optimum resolution for each selected picture area.
- Any sections are possible.
- In parallel to the machining of one workpiece, the machining of another workpiece can be simulated (as of NCU 720.x).
- 3D representation of simulation (option).

General conditions for the simulation

- Very fast simulation result, this means, however, low dynamic resolution.
- For complex representations, fine recording is too slow or does not run for very complex parts.
- "Quickview" for mold-making part programs.
- The simulation cannot be used for compile cycles with special kinematics.

- Compile cycles are supported only to a limited extent.
- Blank CAD data cannot be imported (for example, for castings).
- Couplings cannot be simulated. (coupled motion, axial master-value coupling, electronic gearbox):
 - Coupled motion: TRAILON(<following axis>, <leading axis>, <coupling factor>)
 - Axial master-value coupling: LEADON(Faxis, LAxis, curve table n) actual value and setpoint coupling
 - Electronic gearbox: EGDEF(FA, LA1, coupl.1, LA2, coupl.2, ... LA5, coupl.5), EGON, EGONSYN, EGONSYNE
- Reference point approach G74 is not operative during simulation.
- The following is not supported:
 - Axis container
 - Dependencies on part programs and PLC
 - Multiple clamping
 - CNC function Synchronized actions for safe programmable logic (SPL) in the NC
- Different technologies in different channels with different axis coordinate systems are not fully supported.
- A 3D simulation is not possible for the grinding technology.
- The software limit switches are not evaluated:
 In a manufacturer cycle for the tool replacement, the "Software limit switch + the Z axis" machine data is used as replacement position:
 GO Z=\$MA_POS_LIMIT_PLUS[Z]-0.1
 This position is not used in the simulation, however, but rather the default setting for the machine data = 1e8 = 100.000.000. This specified position is traversed in the simulation. This makes the simulation very slow and causes the machining time to explode, e.g. 555 hours rather than 2 minutes.
- Program start in the simulation:
 If MD22622 \$MCS_DISABLE_PLC_START is set in a channel, then the channel is not automatically started in the simulation. It can then only be started from another channel using the part program command "START". This function is not available for the SINUMERIK 828D.
- Alarm display in the simulation:
 Alarms are output in the simulation exactly the same way as alarms in the normal program execution, but no more than five alarms are displayed (above the graphic, to distinguish them from the machine alarms).
 If a different error signaling behavior (e.g. syntax error) occurs between simultaneous recording and simulation, then, for example, the reason can be due to the hardware access operations in the programming. The simulation does not access the hardware, however, internally it reads the input with 0 and is then exited. On the other hand, for simultaneous recording, an alarm is output.

8.2 Setting the technology for simulation

Turning technology

Examples of machine kinematics:

- Conventional turning with two geometry axes
- Three spindles: Main spindle, counterspindle, tool spindle
- Counterspindle slides, tailstock as NC axis
- B axis: Aligning turning tools in the tool spindle
- Milling with geometry axes: TRANSMIT, TRACYL, TRAANG

Machine data for the turning technology:

MD52200 \$MCS_TECHNOLOGY = 1	Turning technology
MD52000 \$MCS_DISP_COORDINATE_SYSTEM = 34	Position of the coordinate system (example)

Milling technology

Examples of machine kinematics:

- Milling with five axes: Swivel/TRAORI
- Swivel head change

Machine data for the milling technology:

MD52200 \$MCS_TECHNOLOGY = 2	Milling technology
MD52000 \$MCS_DISP_COORDINATE_SYSTEM = 0	Position of the coordinate system (example)

Grinding technology

Machine data for the grinding technology:

MD52200 \$MCS_TECHNOLOGY = 3	Cylindrical grinding technology
MD52200 \$MCS_TECHNOLOGY = 4	Surface grinding technology
MD52000 \$MCS_DISP_COORDINATE_SYSTEM = 0	Cylindrical grinding and surface grinding with moving column (example)
MD52000 \$MCS_DISP_COORDINATE_SYSTEM = 6	Surface grinding with table machine (example)

Meaning of the axes

MD52206 \$MCS_AXIS_USAGE[n]	Meaning of the axes in the channel [n] channel axis number
= 0	No special meaning
= 1	Tool spindle (rotating tool)

= 2	Auxiliary spindle (rotating tool)
= 3	Main spindle (turning)
= 4	C axis of the main spindle (turning)
= 5	Counterspindle (turning)
= 6	C axis of the counterspindle (turning)
= 7	Linear axis of the counterspindle (turning)
= 8	Tailstock (turning)
= 9	Back rest (turning)

Note

To prevent errors occurring, the same axis must be defined as main or counterspindle for all channels.

Enter the direction of rotation for the rotary axes that are not configured in a tool holder or a 5-axis transformation via the following channel-specific machine data.

MD52207 \$MCS_AXIS_USAGE_ATTRIB[n]		Attributes of the axes [n] channel axis number
Bit 0	Rotates around 1st Geometry axis (for rotation axes)	
Bit 1	Rotates around 2nd Geometry axis (for rotation axes)	
Bit 2	Rotates around 3rd Geometry axis (for rotation axes)	
Bit 3	Reversal of direction of rotation (for rotary axes)	

The MD52290 \$MCS_SIM_DISPLAY_CONFIG machine data acts only on the OP019. It has no significance for SINUMERIK 828D.

MD52290 \$MCS_SIM_DISPLAY_CONFIG		Position of the status display of the channel in the simulation.
Bit 0	Upper left-hand corner	
Bit 1	Upper right-hand corner	
Bit 2	Lower left-hand corner	
Bit 3	Lower right-hand corner	

In the channel-specific machine data, for at least one geometry axis per channel, enter a value other than 0.0, e.g. 0.001. If the value is 0.0, the system assumes that this parameter has still not been set.

MD53230 \$MCS_SIM_START_POSITION		Axis position when starting the simulation
Simulation is only possible if a value not equal to 0 has been set for at least one geometry axis.		

Deactivating simulation

The following machine data must be set to deactivate the simulation:

MD51226 \$MNS_FUNCTION_MASK_SIM		Simulation function screen
Bit 1 = 1	Deactivating simulation	
Bit 10 = 1	Hide zero point symbol	

Simulation from kinematic chain

The following machine data must be set to use the simulation from kinematic chains:

MD51226 \$MNS_FUNCTION_MASK_SIM	Simulation function screen
Bit 22	Machine model for simulation from kinematic chain (also without collision avoidance)

Tool change time

The following machine data defines the estimated time for a tool change in the simulation:

MD10190 \$MN_TOOL_CHANGE_TIME	Tool change times for simulation
= 0	Seconds (default value 0)

The value only replaces times that are principally not incurred for a simulated tool change, e.g. wait times for PLC commands (M206).

Traversing commands are added to the calculated time for the duration of the simulation if the commands are included in the tool change cycle.

Add the following example to the programming in the tool change cycle, which means that the value from MD10190 can be incorporated as total tool change time in the estimated simulation time:

```
IF $MN_SIM_ENVIRONMENT B_AND 'B100'  
  M206 ; see MD 22560 $MC_TOOL_CHANGE_M_CODE  
  RET  
ENDIF
```

8.3 Simultaneous recording overview



Software option

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



Software option

You require the "ShopMill/ShopTurn" option to display the tailstock



Software option

You require the "3D simulation 1 (finished part)" option for the 3D representation of the simultaneous recording function.

During machining, the tool paths can be simultaneously recorded on the display of the controller in a three-side view or 3D view. Workpiece graphics and views correspond to the graphic simulation.

Note

To prevent an incomplete representation, you must activate the simultaneous recording prior to the NC start.

Example: Activating the tailstock representation in the spindle chuck data

Procedure:

1. Make the tailstock visible by making the selection in the program.
2. Specify the tailstock dimensions:
Select in the operating area "Parameters" → "Setting data" → "Spindle chuck data", and enter the "ZR" and "XR" parameters for the tailstock length and tailstock diameter.
The tailstock dimensions are written into the following machine data:

MD52218 \$MCS_FUNCTION_MASK_TURN	Turning function screen
Bit 2 = 1	Tailstock enable: The tailstock becomes visible if selected in the "tailstock" program.

MD53240 \$MAS_SPINDLE_PARAMETER[i]	Spindle chuck data in mm
[0]: Chuck dimension [1]: Stop dimension [2]: Jaw dimensions	

MD53241 \$MAS_SPINDLE_CHUCK_TYPE	Spindle jaw type
0 = clamp from the outside 1 = clamp from the inside	

MD53242 \$MAS_TAILSTOCK_PARAMETER[i]	Tailstock data in mm
[0]: Tailstock dimension [1]: Tailstock length	

Activating the machining time display

Timers are provided as system variables in the Program runtime function. While the NC-specific timers are always activated (for time measurements since the last controller power-up), the channel-specific timers must be started via the following channel-specific machine data.

MD27860 \$MC_PROCESSTIMER_MODE	Activate the program runtime measurement.
Bit 0 = 1	The measurement of the total runtime for all part programs is active (\$AC_OPERATING_TIME).
Bit 1 = 1	The measurement of the actual program runtime is active (\$AC_CYCLE_TIME).
Bit 4 = 1	Measurement also during active dry run feedrate.
Bit 5 = 1	Measurement also during program test.

8.4 Clamping a blank

The blank clamping function allows you to select and specify in more detail the clamping location for the blank.

You require this function if your machine satisfies one or more of the following requirements:

Requirements

- The machine has several clamping options available for selection.
- The machine has a clamping option that does not point in the Z direction as standard (e.g. a boring mill).

Under the following circumstances, you can specify the position and orientation of the rotary axes in the machine data MD53220 and MD52207:

- The position and orientation of the rotary axis is offset against the MCS zero point, but not specified by a tool holder or a TRAORI.
- In a turning machine, the reference points of the main spindle or counterspindle are offset against the MCS zero point in X or Z.
- The counterspindle cannot be traversed with a saddle, but rather is at a fixed position.

In order that for simulation / simultaneous recording, the rotary axes are displayed at the correct location, their positions in the machine coordinate system must be specified using the following machine data:

MD53220 \$MAS_AXIS_MCS_POSITION[n]	Position of the axis in the MCS [n] 0 = X, 1 = Y, 2 = Z
------------------------------------	--

Additional information about the transformation of a rotary axis is provided in Chapter "Technologies and cycles (Page 129)".

Function

Set the following channel-specific machine data:

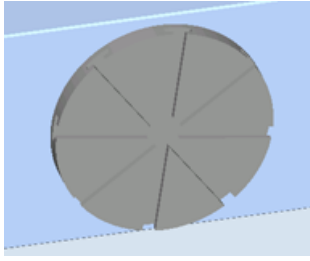
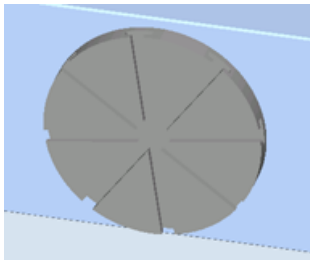
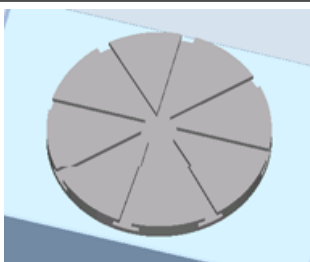
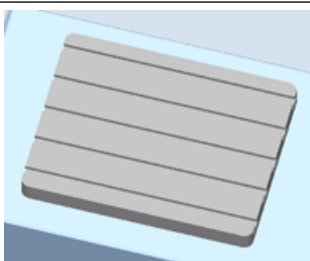
MD52207 \$MCS_AXIS_USAGE_ATTRIB[n]		Attribute of the axes in the channel [n] channel axis number
Bit 0	Rotates around 1st Geometry axis (for rotation axes)	
Bit 1	Rotates around 2nd Geometry axis (for rotation axes)	
Bit 2	Rotates around 3rd Geometry axis (for rotation axes)	
Bit 8	Lists rotary axis for blank clamping (on milling machines)	
Bit 13	Position of the axis for blank selection from active work offset (with rotary axes)	
= 0	When simulation or simultaneous recording are started, the mountable rotary axis is moved to the position from MD53220.	
= 1	With a blank command, The mountable rotary axis is moved into the zero point of the currently active work offset.	

MD52214 \$MCS_FUNCTION_MASK_MILL		Function mask for milling
Bit 1	Offer fixed table for blank clamping (on milling machines)	

Take the machine configuration for MD20080 from the example.

Function options

In the program header, a blank can be optionally set to A, B, C and table clamping:

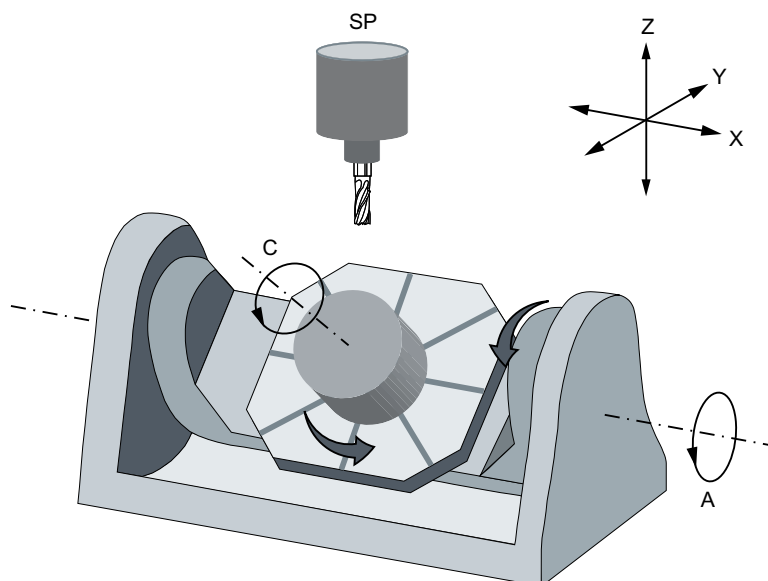
Clamping	View	Machine data
A axis		MD52207 \$MCS_AXIS_USAGE_ATTRIB bit 0 and bit 8 = 1
B axis		MD52207 \$MCS_AXIS_USAGE_ATTRIB bit 1 and bit 8 = 1
C axis		MD52207 \$MCS_AXIS_USAGE_ATTRIB bit 2 and bit 8 = 1
Table		MD52214 \$MCS_FUNCTION_MASK_MILL bit 1 = 1

Note

To obtain the selection option in the program header, two rotary axes must be defined for the raw part clamping.

Example for the machine configuration

Example for a machine configuration when the A axis should rotate at the X axis:



MD20080 \$MC_AXCONF_CHANAX_NAME_TAB[n]		Channel axis name in channel [n]
[0]	X	
[1]	Y	
[2]	Z	
[3]	Spindle	
[4]	A	
[5]	C	

MD52207 \$MCS_AXIS_USAGE_ATTRIB[4]		Attribute of rotary axis A
= 181H		
Bit 0 = 1	Rotates around 1st Geometry axis (for rotation axes)	
Bit 8 = 1	Lists rotary axis for blank clamping (on milling machines)	

Additional information

Additional information about setting ShopMill is provided in the
Milling Operating Manual

Technologies and cycles

9.1 Activating turning/milling/drilling/grinding technologies

The cycles are automatically loaded when the controller boots (Power On).

The configuration is realized using cycle machine data and cycle setting data. In so doing, a differentiation is made between the following data:

- General cycle machine data
- Channel-specific cycle machine data
- Axis-specific cycle machine data
- General cycle setting data
- Channel-specific cycle setting data
- Axis-specific cycle setting data

Note

Specifying the technology

For the SINUMERK 828D, the technology (turning, milling or grinding) is determined by the ordered software package. The technology cannot be parameterized with machine data.

Adapting the user interface

The softkeys display for selecting the cycle support in the program editor is set by the following channel-specific configuration machine data:

MD52200 \$MCS_TECHNOLOGY	Technology
= 1	Setting for turning
= 2	Setting for milling
= 3	Setting for the cylindrical grinding technology
= 4	Setting for the surface grinding technology

MD52201 \$MCS_TECHNOLOGY_EXTENSION	Technology extension for combined machines with several technologies
= 1	Turning technology
= 2	Milling technology
Example turning machine with milling technology MD52200 \$MCS_TECHNOLOGY = 1 MD52201 \$MCS_TECHNOLOGY_EXTENSION = 2	

Note

Cylindrical and surface grinding are not supported as a technology extension by SINUMERIK Operate.

Additional information

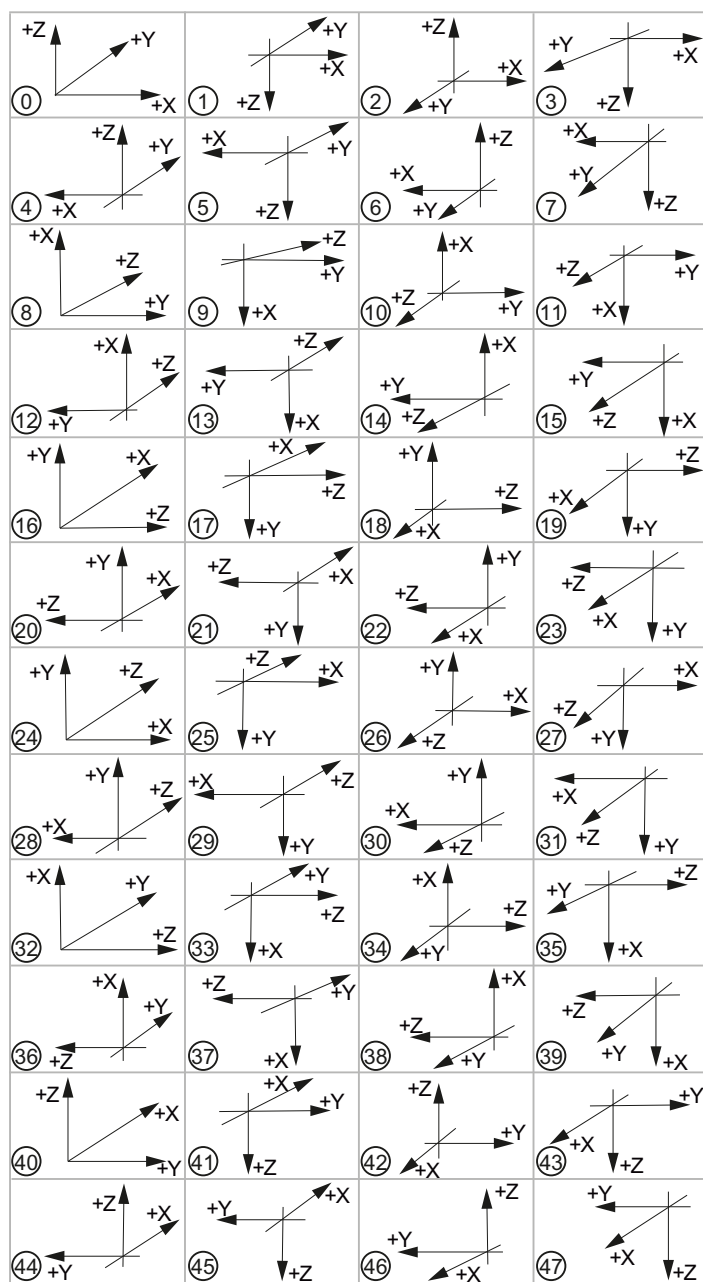
Information about grinding technology is available in:

- Chapter "Grinding (Page 187)"
- Grinding Operating Manual
- NC Programming Programming Manual
- Monitoring and Compensating Function Manual

Setting the coordinate system

MD52000 \$MCS_DISP_COORDINATE_SYSTEM		Position of the coordinate system
= 0	Default setting	
With this machine data you adapt the position of the coordinate system of the user interface to the coordinate system of the machine. In the user interface, all help screens, sequence graphics, simulations, and input fields with circle direction indication change automatically according to the selected position.		

The following settings can be selected for the coordinate system:



Meaning of the axes

MD52206 \$MCS_AXIS_USAGE[n]		Meaning of the axes in the channel [n] channel axis number
= 0	No special meaning	
= 1	Tool spindle (rotating tool)	
= 2	Auxiliary spindle (rotating tool)	
= 3	Main spindle (turning)	
= 4	Separate C axis of the main spindle (turning)	

9.1 Activating turning/milling/drilling/grinding technologies

= 5	Counterspindle (turning)
= 6	Separate C axis of the counterspindle (turning)
= 7	Linear axis of the counterspindle (turning)
= 8	Tailstock (turning)
= 9	Steady (turning)
=10	B axis (turning)
= 12	B axis in counterspindle (turning)
= 13	Transverse travel X of the counterspindle (turning)

Defining the direction of rotation

Enter the direction of rotation for the rotary axes that are not configured in a tool holder or a 5-axis transformation via the following channel-specific machine data.

MD52207 \$MCS_AXIS_USAGE_ATTRIB[n]		Attribute of the axes in the channel [n] channel axis number
Bit 0	Rotates around 1st geometry axis (for rotation axis)	
Bit 1	Rotates around 2nd geometry axis (for rotation axis)	
Bit 2	Rotates around 3rd geometry axis (for rotation axis)	
Bit 3	Direction of rotation is counter-clockwise (for rotary axis / C axis).	
Bit 4	Displayed direction of rotation of the spindle/C axis for the M function M3 is counter-clockwise	
Bit 5	Inverts M3 / M4 (for spindles) You must set this bit as for PLC bit DB38xx.DBX2001.6.	
Bit 6	Display rotary axis as offset target for measurement	
Bit 7	List rotary axis in the position pattern	
Bit 8	List rotary axis to accept blank (on milling machines)	
Bit 9	Spindle is not SPOS-capable	
Bit 10	Rotary axis rotates around 1st geometry axis (only for position pattern)	
Bit 11	Rotary axis rotates around 2nd geometry axis (only for position pattern)	
Bit 12	Rotary axis rotates around 3rd geometry axis (only for position pattern)	

Additional settings

MD52005 \$MCS_DISP_PLANE_MILL		PlaneSelectionMill
= 0	Plane selection in the operator interface (cycle support for programming under "program-GUIDE G-Code")	
= 17	G17 plane (default value)	
= 18	G18 plane	
= 19	G19 plane	

MD52006 \$MCS_DISP_PLANE_TURN		Plane selection turning
= 17	G17 plane	
= 18	G18 plane (permanently set)	
= 19	G19 plane	

MD52212 \$MCS_FUNCTION_MASK_Tech		General function mask for all technologies
Bit 0	Enable swivel with tool holder	
= 0	Swivel plane, swivel tool not enabled	
= 1	Swivel plane, swivel tool enabled	
Bit 1	Optimized travel along software limit switches	
= 0	No optimized travel along software limit switches	
= 1	Optimized travel along software limit switches	

MD52240 \$MCS_NAME_TOOL_CHANGE_PROG		Tool change program for G code steps
= Program name	The associated program is called for tool change.	

9.2 Technology cycles for drilling

Drilling technology

You can set drilling technology using the following channel-specific configuration machine data and channel-specific cycle setting data.

MD52216 \$MCS_FUNCTION_MASK_DRILL		Drilling function screen form
Bit 0	Tapping cycle CYCLE84, technology input fields	
= 0	Hide input fields	
= 1	Display input fields	
Bit 1	Tapping cycle CYCLE840, technology input fields	
= 0	Hide input fields	
= 1	Display input fields	

SD55216 \$SCS_FUNCTION_MASK_DRILL_SET		Drilling function screen form
Bit 1	Boring CYCLE86: Take into account the rotation of the tool plane when positioning the spindle Note: Spindle direction of rotation M3/ M4 and direction of rotation of the rotary axes must be set according to DIN. For SPOS=0, the tool cutting edge points in the + direction of the 1st axis of the plane (for G17, to X+).	
= 0	Do not take into account the rotation of the tool plane when positioning the spindle (SPOS).	
= 1	Take into account the rotation of the tool plane when positioning the spindle (SPOS).	
Bit 2	Boring CYCLE86: When positioning the spindle, take into account swiveled table kinematics (tool carrier)	
= 0	When positioning the spindle, do not take into account the components of the swiveled table kinematics (swiveling with CYCLE800, tool carrier). Note: Only the rotary axis of the machine kinematics that rotates the workpiece is taken into account. In the initial state of the kinematics, this rotary axis must be orientated parallel to the tool axis. e.g. rotary axis C rotation around Z (for G17) rotary axis vector of the swivel data set V2xyz = 0, 0, -1.	
= 1	When positioning the spindle, take into account the components of the swiveled table kinematics (swiveling with CYCLE800, tool carrier).	
Bit 3	Tapping CYCLE84: Monitoring of the machine data MD31050 \$MA_DRIVE_AX_RATIO_DE-NOM and MD31060 \$MA_DRIVE_AX_RATIO_NUMERA of the spindle	
Bit 4	Tapping CYCLE84, set monitoring of the machine data MD31050 \$MA_DRIVE_AX_RATIO_DE-NOM and MD31060 \$MA_DRIVE_AX_RATIO_NUMERA of the spindle	
= 0	No monitoring	
= 1	Monitoring	
Bit 6	Boring CYCLE86: Adapt the spindle position to the tool orientation Tool orientation can be realized using TOOLCARR or TRAORI before the cycle is called.	
= 1	Bits 1 and 2 are no longer effective.	
Bit 7	Boring CYCLE86: Adapt the direction of retraction travel in the plane to the active mirroring	

= 0	Retraction in the plane is realized as programmed.
= 1	When mirroring is active, the direction when retracting in the plane is adapted in the cycle to the active mirroring.

Tapping (CYCLE84 and CYCLE840)

If the technology screens are hidden using the channel-specific configuration machine data MD52216 \$MCS_FUNCTION_MASK_DRILL, then the settings in the following channel-specific cycle setting data are effective:

SD55481 \$SCS_DRILL_TAPPING_SET_GG12[0]		Setting tapping G group 12: Block change behavior for exact stop
= 0	Exact stop response as before the cycle call (default value).	
= 1	G601	
= 2	G602	
= 3	G603	

SD55482 \$SCS_DRILL_TAPPING_SET_GG21[0]		Setting tapping G group 21: Acceleration behavior
= 0	Acceleration behavior as before the cycle call (default value).	
= 1	SOFT	
= 2	BRISK	
= 3	DRIVE	

SD55483 \$SCS_DRILL_TAPPING_SET_GG24[0]		Setting tapping G group 24: Precontrol
= 0	Precontrol, the same as before the cycle call (default value).	
= 1	FFWON	
= 2	FFWOF	

Tapping (CYCLE84)

SD55484 \$SCS_DRILL_TAPPING_SET_MC[0]		Spindle operation for MCALL
= 0	For MCALL, reactivate spindle operation (default value).	
= 1	For MCALL, remain in position controlled spindle operation.	

9.3 Manufacturer cycles

9.3.1 Manufacturer cycles

Note

Protect your technological knowledge

To protect your know-how, you can protect your cycles with individual access rights and additional file encryption. Implement this cycle protection by means of the following measures:

- Encrypt your cycle data with the additional SIEMENS application SINUCOM Protector.
Further information about the SINUCOM Protector can be found on the Internet ([https://support.industry.siemens.com/cs/document/109474775/sinumerik-828d-840d-sl%3A-cycle-protection-with-protector-for-sinumerik-\(oem-system-integrator\)?dti=0&lc=en-WW](https://support.industry.siemens.com/cs/document/109474775/sinumerik-828d-840d-sl%3A-cycle-protection-with-protector-for-sinumerik-(oem-system-integrator)?dti=0&lc=en-WW)).
- Assign individual access rights to your cycle data and adapt the authorization levels for the user.

Further information on the individual assignment of access rights can be found in Chapter "Access levels for programs (Page 30)".

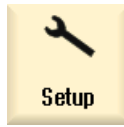
The following cycles are also available in the cycle package for individual adaptation:

CUST_TECHCYC.SPF	Manufacturer cycle for the adaptation of technology cycles.
CUST_782.SPF	Manufacturer cycle for adaptation to determine loading function (CYCLE782), see Chapter "Manufacturer cycle CUST_782.SPF (Page 228)".
CUST_800.SPF	Manufacturer cycle for the adaptation of the swivel plane and the swivel tool functions (CYCLE800), see Chapter "Manufacturer cycle CUST_800.SPF (Page 216)".
CUST_832.SPF	Manufacturer cycle for the adaptation of the high-speed settings function (CYCLE832), see Chapter "Manufacturer cycle CUST_832.SPF (Page 231)".
CUST_MEACYC.SPF	Manufacturer cycle for the adaptation of the measuring functions, see Chapter "Manufacturer and user cycle CUST_MEACYC.SPF (Page 238)".
CUST_T	Cycle is used to track the T preparation after SERUPRO.
CUST_M6	Cycle is used to track the tool change after SERUPRO.
CUST_MULTICHAN	Manufacturer cycle for lathes with several channels
CUST_CLAMP	Manufacturer's cycle for clamping elements

Copying manufacturer cycles

Copy the cycle from the NC data/Cycles/Standard cycles directory to the NC data/Cycles/

User cycles directory before making any changes. Changes for the cycle take effect only in this directory.



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



2. Press the "System data" softkey.

3. Open the directory NC data/Cycles/Standard cycles and select the manufacturer cycles.



4. Press the "Copy" softkey and open the NC data/Cycles/Manufacturer cycles directory.



5. Press the "Paste" softkey.
The cycles are available to you for individual adaptation.

9.3.2 PROG_EVENT.SPF standard cycle

Rules for PROG_EVENT

PROG_EVENT.SPF is a standard cycle and cannot be changed (not accessible).

- Storage of a PROG_EVENT.SPF cycle that you have written under manufacturer or user cycles is **not** switched active.
- Manufacturer-specific "prog_events" must be reproduced by the following cycles, which, when available, are automatically called by the PROG_EVENT.
 - CYCPE_MA.SPF is called at the end of the internal PROG_EVENT.SPF.
Note: CYCPE_MA.SPF should preferably be used.
 - CYCPE1MA.SPF is called at the beginning of the internal PROG_EVENT.SPF.
- Block search:
Default setting: The correction of tool change, spindle direction of rotation / speed, swivel axes is performed by the internal PROG_EVENT, the necessary machine data is appropriately preset. The manufacturer does not have to use a CYCPE_MA.SPF for the actions described above.

Special features of the block search in conjunction with PROG_EVENT:

Machine data		Bit
MD11450	\$MN_SEARCH_RUN_MODE	Bit 1=1
MD52212	\$MCS_FUNCTION_MASK_Tech	Bit 3=1

Note

If \$P_PROG_EVENT==5 is used in the CYCPE_MA.SPF, no REPOSA may be programmed as this is already present in the PROG_EVENT.SPF, otherwise an error occurs for SSL with calculation.

9.3.3 Manufacturer cycle for tool change CUST_T and CUST_M6

Function

The two cycles are used to track the T preparation (CUST_T) and the tool change (CUST_M6) according to SERUPRO. In so doing, depending on the specific tool holder, the following is output:

- Last programmed change
- Last programmed preparation

Sequence

The two cycles are called in the "SERUPRO-END-ASUB", which also provides the two transfer parameters.

Parameter	Meaning
_THnr	Tool holder or master spindle number that was programmed for the change or the preparation.
_MTHnr	Number of the active tool holder or master spindle at the time of programming.

The cycles themselves do not contain and "machine logic". Pure data operations are performed.

The active tool holder or the master spindle is saved.

Using the transfer parameter, the situation at the time of the T or M6 programming is determined. This situation is established, T and/or M06 output or a branch is made into the manufacturer cycles. After the output of T and M06 or after returning from the manufacturer cycles, the tool holder, saved at the beginning, is reactivated.

A change cycle can then only be called without any restriction, if the corresponding queries such as program test, block search,... exist.

T replacement cycle

If a T replacement cycle is being used, then the following note must be observed:

Note

The CUST_T cycle uses the language command TCA (ToolChangeAbsolut) for the T call, and therefore also has the possibility of inserting tools that have been disabled. The language command TCA is reconfigured to _TCA for 828D. TCA can be substituted by MD15710 \$MN_TCA_CYCLE_NAME.

The following must be observed if working at a machine with T replacement:

- The T preparation cycle must be explicitly called at the specified position.
 - If the call is realized using the CUST_T cycle, then the parameter evaluation (\$C_TS_PROG, ...) is skipped.
Alternatively, the original TCA language command can be reprogrammed.
-

Additional information

You can find additional information about "SERUPRO and ASUB" in the Basic Functions Function Manual.

9.3.4 CUST_TECHCYC.SPF manufacturer cycle

Function

For technological cycles, the CUST_TECHCYC cycle is called from ShopMill and ShopTurn cycles, if the machine builder must make certain adaptations to a specific machine (e.g. activate specific M commands). For instance, this may be necessary in order to retract or extend part catchers when cutting off parts on lathes or to position a spindle for clamping (only in JobShop).

The CUST_TECHCYC.SPF cycle can be used to program and execute the necessary sequences.

Adapt manufacturer cycle CUST_TECHCYC.SPF

You can execute the following actions with the aid of the `_MODE` parameter:

Marker	Action	
_M1	Main spindle:	Changeover to C axis mode
_M2		Changeover to spindle mode
_M3		Clamp C axis
_M4		Release C axis clamping Damping brake off
_M5		Rinse chuck
_M6		Close chuck
_M7		Open chuck when spindle is stationary
_M8		Open chuck when spindle is rotating
_M9		Flush chuck
_M11	Tool spindle:	Changeover to C axis mode
_M12		Changeover to spindle mode
_M13		Clamp C axis
_M14		Release C axis clamping
_M21	Counterspindle:	Changeover to C axis mode
_M22		Changeover to spindle mode
_M23		Clamp C axis
_M24		Release C axis clamping Damping brake off
_M25		Rinse chuck
_M26		Close chuck
_M27		Open chuck when spindle is stationary
_M28		Open chuck when spindle is rotating
_M29		Rinse chuck off
_M30	Position 4th axis after block search (e.g. counterspindle, tailstock, back rest)	
_M41	Engage driven tool	
_M42	Disengage driven tool	
_M61	Changeover to turning	
_M62	Changeover to peripheral surface machining C	
_M63	Changeover to face machining C	
_M64	Changeover to peripheral surface machining Y	
_M65	Changeover to face machining Y	
_M66	Changeover to face machining B (swivel)	
_M67	Deselection of the face machining B (swivel)	
M68	After block search to ST block before swiveling	

Marker	Action	
_M100	Drawer:	Position before cut-off
_M101		Open during cut-off
_M102		Close after cut-off
_M103	Cut-off completed	
_M110	Tool change	
_M111	After end of tool change cycle (with tool change)	
_M112	Before tool change on retraction plane	
_M113	Before cutting edge reselection	
_M114	After end of tool change cycle (without tool change)	
_M115	after tool preselection (ShopTurn)	
_M116	To tool change point (ShopTurn)	
_M117	Before tool change T command	
_M118	Output of M and H functions after tool change (ShopMill, ShopTurn)	
_M120	Definition: Coupling of counterspindle / main spindle	
_M121	After COUPOF switch off position control	
_M122	Definition: Coupling, main/counterspindle	
_M123	After COUPOF switch off position control	
_M125	Changeover to main spindle	
_M126	Changeover to counterspindle	
_M131	Start of program (ShopTurn program)	
_M132	End of program header (ShopTurn program)	
_M135	End of program loop (ShopTurn program)	
_M136	End of program (ShopTurn program)	
_M140	Block search	Before tool change
_M141		After tool change
_M142		After tool change (new tool from ShopTurn)
_M150	2-channel stock removal:	Leading channel before Balance Cutting
_M151		Leading channel after Balance Cutting
_M152		Following channel before Balance Cutting
_M153		Following channel after Balance Cutting
_M154		Start machining in the leading channel
_M155		End machining in the leading channel
_M156		Start machining in the following channel
_M157		Start machining in the following channel
_M160	Start multichannel programming	
M170	Start block with spindle	

Marker	Action
_M171	End block with spindle
_M231	Start of program (ShopMill program)
_M232	End of program header (ShopMill program)
_M235	End of program loop (ShopMill program)
_M236	End of program (ShopMill program)
_M300	Boring CYCLE86 Adapt the spindle to the tool position (SD55216 \$SCS_FUNCTION_MASK_DRILL_SET bit 6 = 1) For specific machine kinematics, it may be necessary to make an additional adjustment of the correction angle for the spindle positioning (e.g. intermediate gear at the spindle). The _M300 marker is run through after the correction angle is calculated. There the correction angles _MEA_CORR_ANGLE[0] (for the manufacturer) and _MEA_CORR_ANGLE[1] (system) can be changed, if applicable. For subsequent spindle positioning operations, both angles act additively in CYCLE86.
_M403	Main spindle: Damping brake on
_M423	Counterspindle: Damping brake on
_M900	Before blank output (only ShopMill + ShopTurn)

9.3.5 CUST_MULTICHAN user cycle

Function

The CUST_MULTICHAN cycle is used for multi-channel programming at lathes. It is called at the start of the block when programming with blocks.

In the cycle, e.g. the master spindle is set to the spindle specified at the start of the block. The return value (= 1) can be used to control whether the complete block is skipped.

CUST_MULTICHAN

Parameter	Meaning
_S_NR	Spindle number to which the master spindle is set.
_RET	Return value
= 0	The block is executed corresponding to the run-in mode.
= 1	The entire block is skipped.

9.3.6 Commissioning multi-channel machines with programSYNC

This section describes the special features that you must observe when commissioning a multi-channel machine with programSYNC (option) and ShopTurn.



Software option

To use function programSYNC, you require the following option: "programSYNC".

Additional information

Information about additional functions – such as help functions, block search or frames is provided in the

Basic Functions Function Manual.

Setting the multi-channel view

In automatic operation, select under "Settings" → "Multi-chan. function" which channels will be displayed adjacently.

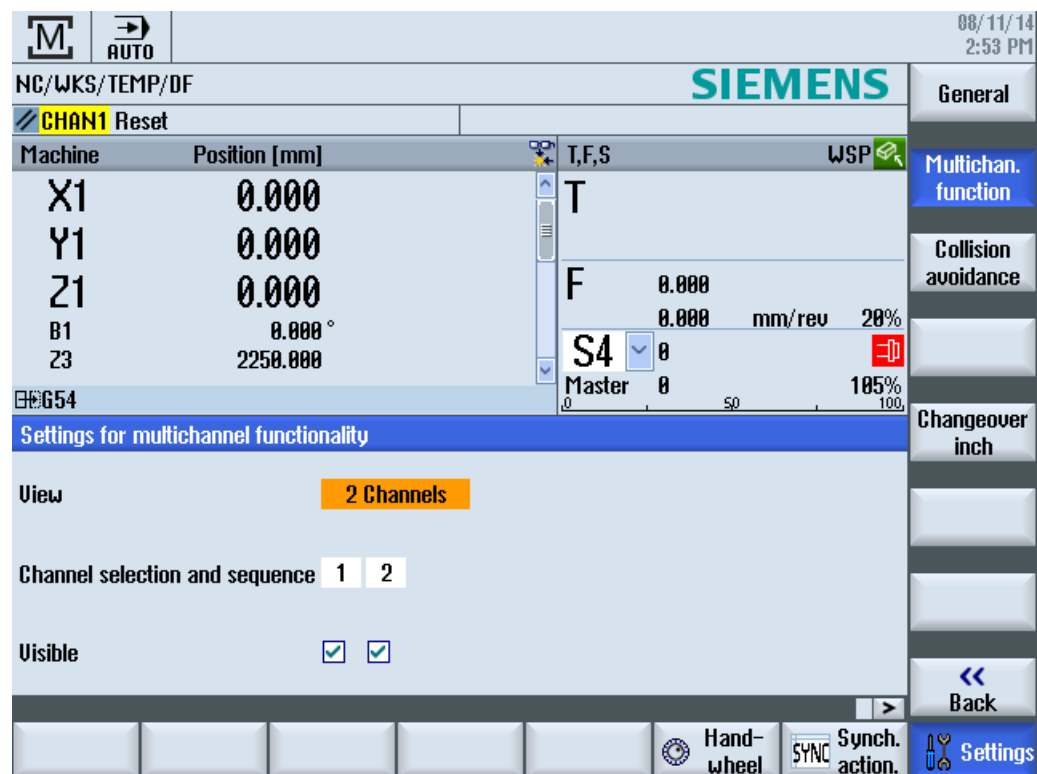


Figure 9-1 Dialog for viewing multiple channels

Defining work offsets

For multi-channel machines, it is desirable to maintain the same specific axes in certain work offsets over all channels.

Both channels work on the same workpiece, whose front edge normally has the value $Z = 0$ and which is defined using a settable work offset.

The position of the origin on the counterspindle is also defined using a settable work offset. However, this also depends on the position of the Z3 motion axis of the counterspindle, of which there is only one.

Normally, assign the following frames for a work offset:

MD10616 \$MN_MAPPED_FRAME_MASK	Enable frame mapping
Bit 0 = 1	System frame for actual value setting and scratching (\$P_SETFR)
Bit 12 = 1	Channel-specific base frames (\$P_CHBFR)
Bit 13 = 1	Settable frames, work offsets (\$P_UIFR)

With the following machine data, specify the connection of the axes that should have the same value in the work offset across all channels:

MD32075 \$MA_MAPPED_FRAME[AXn]	Mapping of an axial frame
--------------------------------	---------------------------

Example:

Axis AX2 = Z1 (Z-axis in the first channel)

Axis AX6 = Z3 (motion axis of the counterspindle)

Axis AX11 = Z2 (Z-axis in the second channel)

MD32075 \$MA_MAPPED_FRAME[AX6] = "AX6"

MD32075 \$MA_MAPPED_FRAME[AX11] = "AX2"

The motion axis of the counterspindle is connected with itself ($Z3 = Z3$). The two Z-axes of the channels are connected ($Z2 = Z1$). You only need to create the connection of the axes in one direction.

Program test run

To support the program test run, the following dialog is available under "Prog. ctrl." → "Run in":

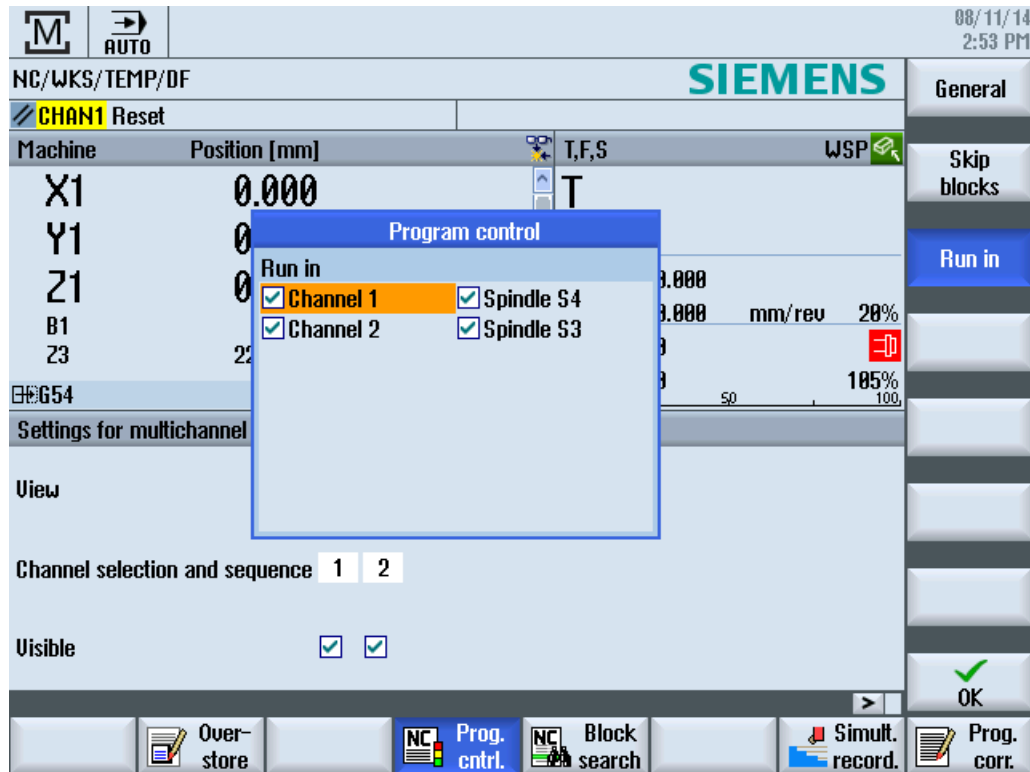


Figure 9-2 Program test run

In the dialog, specify which channel and which spindle should be used for the program test run.

The check marks for the program test run (dry run) are displayed as follows in the PLC:

(check mark = processing = no dry run = 0)

- Channel 1...n: DB330x.DBX1.7
- Spindle n: DB380x.DBX1.3

Note

An example for commissioning the positioning function is also supplied in the SINUMERIK 828D Toolbox.

All required M-commands must be grouped. Some groups are predefined for the standard M-commands (e.g. M3, M4, M5, ...), see following machine data item:

MD22040 \$MC_AUXFU_PREDEF_GROUP	Group assignment of predefined auxiliary functions.
---------------------------------	---

You must define user-defined M-commands in the following machine data item:

MD22000 \$MC_AUXFU_ASSIGN_GROUP	Auxiliary function group
---------------------------------	--------------------------

Bits 9, 10, 11 and 15 have a defined meaning in the following machine data:

MD22035 \$MC_AUXFU_ASSIGN_SPEC	Specification of the output behavior for the user-defined auxiliary functions.
MD22080 \$MC_AUXFU_PREDEF_SPEC	Specification of the output behavior for the predefined auxiliary functions.
Bit 9	Acquisition during block search type 5 (SERUPRO)
Bit 10	No output during block search type 5 (SERUPRO)
Bit 11	Cross-channel auxiliary function (SERUPRO)
Bit 15	No output during the test run of the program (dry running)

If a spindle is marked, the M-commands (direction of the spindle rotation, spindle chuck open/closed, ...) that relate to this spindle should be output. This must be initiated explicitly in the PLC (e.g. deactivate spindle via DB380x.DBX1.3).

If a channel is marked, M-commands (coolant, ...) that relate to this channel should be output. To do this, use bit 15. Alternatively, you can also select an initiation in the PLC.

Typical settings for cross-channel M-commands (e.g. M3, M4, M5) are:

Bit 9 = 1, bit 10 = 1, bit 11 = 1, bit 15 = 0

Typical settings for channel-specific M-commands (e.g. M7, M8, M9) are:

Bit 9 = 1, bit 10 = 1, bit 11 = 0, bit 15 = 1

Calculating the program runtime

To calculate the program runtime, you can assign an average time to the M-commands in the following machine data:

MD22037 \$MC_AUXFU_ASSIGN_SIM_TIME	Acknowledgment time of the auxiliary function in msec.
------------------------------------	--

MD22090 \$MC_AUXFU_PREDEF_SIM_TIME	Acknowledgment time of the auxiliary function in msec.
------------------------------------	--

You can increase the time with the following commands:

- \$AC_PRTIME_M_INC, operating time
- \$AC_PRTIME_A_INC, standstill time

You can consider individually known times.

Block search with SERUPRO

To perform a fast block search, you must specify the following values:

MD22600 \$MC_SERUPRO_SPEED_MODE	Velocity for block search type 5
= 0	Program test with block search / dry run velocity:

MD22601 \$MC_SERUPRO_SPEED_FACTOR	Velocity factor for block search type 5
= 10000	Factor which is multiplied by the dry run feedrate.

These values are also required for the simulation.

To prevent M-commands from being issued during the search with SERUPRO, you must set bit 10 to "1" in the following machine data:

MD22035 \$MC_AUXFU_ASSIGN_SPEC	Specification of the output behavior for the user-defined auxiliary functions.
Bit 10 = 1	No output during block search type 5 (SERUPRO)

MD22080 \$MC_AUXFU_PREDEF_SPEC	Specification of the output behavior for the pre-defined auxiliary functions.
Bit 10 = 1	No output during block search type 5 (SERUPRO)

To start the part program, set the following machine data:

MD10702 \$MN_IGNORE_SINGLEBLOCK_MASK	Prevent single-block stop
Bit 16 = 1	Single block has no effect with SERUPRO

MD10708 \$MN_SERUPRO_MASK	Block change modes
Bit 0 = 1	M0 has no effect with SERUPRO
Bit 4 = 1	Override has no effect with SERUPRO

To call the cycle for the SERUPRO process, set the following bit:

MD52212 \$MCS_FUNCTION_MASK_TECH	General function screen for all technologies
Bit 5 = 1	Call block search function for SERUPRO

You can set the selection options for the search modes separately in the following machine data for programGUIDE and ShopTurn (single channel):

MD51024 \$MNS_BLOCK_SEARCH_MODE_MASK_JS	Bit mask for the available search modes (ShopMill, ShopTurn single channel)
Bit 0 = 1	Block search with calculation, without approach

MD51028 \$MNS_BLOCK_SEARCH_MODE_MASK	Bit mask for the available search modes
= 33H	
Bit 0 = 1	Block search with calculation, without approach
Bit 1 = 1	Block search with calculation, with approach
Bit 3 = 0	Skip EXTCALL programs
Bit 4 = 1	Block search without calculation (only programGUIDE)
Bit 5 = 1	Block search with dry run

Block search: Position clamped axes

At the end of the PROGEVENT cycle, all axes are positioned automatically with REPOSA. Consequently, for clamped axes, the clamping must be opened automatically via the PLC or the axes must be positioned by the machine manufacturer previously in the CYCPE_MA cycle.

Example:

```
IF ($P_PROG_EVENT==5)AND($P_SEARCHL==5); Block search with SERUPRO
  IF($AC_RETPOINT[B]<>$AA_IW[B]); Rotary axis is not in position
    M... ; Open clamping
    G0 B=$AC_RETPOINT[B]
    M... ; Reclamp axis
  ENDIF
ENDIF
```

Block programming

It is preferable to use blocks during the multi-channel programming. To create a well structured program, you can combine machining blocks.

You can assign the blocks to a spindle. By default, the spindle becomes the master spindle at the start of the block. This behavior is configured in the CUST_MULTICHAN cycle and can be augmented by the machine manufacturer as required.

Coordinate systems

Normally, the tools are used with 180° offset on the workpiece. This produces different coordinate systems for the channels:

- Channel 1

MD52000 \$MCS_DISP_COORDINATE_SYSTEM	Position of the coordinate system
= 34	Z to the right, X upwards

- Channel 2

MD52000 \$MCS_DISP_COORDINATE_SYSTEM	Position of the coordinate system
= 33	Z to the right, X downwards

You can also set this 180° offset in a base work offset permanently for the C-axis of the second channel. Consequently, for example, holes at 0° are drilled in both channels at the same workpiece position.

Tool list

The tool positions are displayed in accordance with the associated channel in the tool list. If only one TOA is available, non-loaded tools are always displayed in the coordinate system of the first channel.

Main spindle and counterspindle

You must assign the spindles identically in both channels. If the main spindle in the first channel is the "left-hand" spindle, you must specify the main spindle in the second channel correspondingly.

You define the meaning of the axes in the channel in the following machine data, e.g. 3 = main spindle:

MD52206 \$MCS_AXIS_USAGE	Meaning of the axes in the channel
= 3	Main spindle (turning)

Multi-channel data

A task list is created for the programming with programSYNC. This task list contains the multi-channel data that includes cross-channel information. The multi-channel data informs the simulation, e.g. about the raw part. The multi-channel data contains implicitly a WAIT flag to ensure that all programs in the involved channels are synchronized at the start.

- In ShopTurn programs, the multi-channel data is called automatically in the program header.
- In G-Code programs, the relevant call of the multi-channel data is added automatically via CYCLE208 after generating the task list.

If this call should not be added automatically, you can suppress it with MD51228 \$MNS_FUNCTION_MASK_TECH Bit 0 = 1. The operator is then responsible for calling the CYCLE208 and for informing the simulation about the blank.

Program start

Two options are available to start programs in the channels:

- Press the "NC Start" key to start all channels automatically.
- Alternatively, the start can act only for the first channel that then starts all other involved channels.

For this alternative, in the manufacturer's cycle CUST_TECHCYC, you must appropriately adapt mark "_M160".

9.4 Milling

9.4.1 General

All settings for machine and setting data of the standard cycles are also applicable for milling under ShopMill and turning under ShopTurn.

9.4.2 Technology cycles for milling

Milling function screen

SD52214 \$SCS_function_MASK_MILL		Milling function screen
Bit 0	Enable cylinder surface transformation (ShopMill)	
Bit 1	List table to accept blank (on milling machines) (This bit must only be set if the machine has a rotation axis and a fixed table for the blank clamping.)	
Bit 2	Special treatment of disc mill/saw with multi-edge (For these tool types the first engagement is selected so that the top edge of the tool touches reference point Z0 exactly.)	
Bit 3	Enable machining inside/outside (ShopTurn)	
Bit 4	Enable clamp spindle (C-axis) (ShopTurn)	
Bit 5	Enable spindle control of the tool spindle via user interface	
Bit 6	Enable spindle control of the main spindle via user interface	

SD55214 \$SCS_FUNCTION_MASK_MILL_SET		Milling function screen
Bit 0	Basic setting milling cycles in sync.	
Bit 2	Depth calculation of the milling cycles, with or without safety clearance.	
= 0	Depth calculation of the milling cycles is performed between the reference plane + safety clearance and the depth.	
= 1	Depth calculation is performed without including the safety clearance. Bit 2 is effective in the following milling cycles: CYCLE61, CYCLE71, CYCLE76, CYCLE77, CYCLE79, CYCLE899, LONGHOLE, SLOT1, SLOT2, POCKET3, POCKET4.	

Contour milling (CYCLE63, CYCLE64)

SD55460 \$SCS_MILL_CONT_INITIAL_RAD_FIN		Finishing approach circle radius The radius of the approach circle during the finishing of contour pockets is affected.
= 0	The radius is selected so that at the starting point the safety clearance to the finishing allowance is maintained (default value).	
> 0	The radius is selected so that at the starting point the value of this channel-specific setting data to the finishing allowance is maintained.	

SD55212 \$SCS_FUNCTION_MASK_Tech_SET	General function screen for all technologies
= 6	
Bit 3	Delete programs generated by contour cycles (CYCLE63, CYCLE64, CYCLE952)
= 0	Generated programs are not deleted (compatibility as before)
= 1	Generated programs are deleted as soon as they have been executed by the calling cycle.

Engraving (CYCLE60)

SD55400 \$SCS_MILL_ENGRAVE_POINT_RAD	Engraving cycle CYCLE60: Circular arc radius for generating the "dot" character
= 0	
= 0	The "dot" character is produced by a simple linear insertion movement of the tool (default setting for conventional engraving tools, compatibility).
> 0	The "dot" character is executed as a circle with the radius value from this setting data (setting for some special tools).

Multiple edge (CYCLE79), circular position pattern (HOLES2), circumferential groove (SLOT2)

SD55230 \$SCS_CIRCLE_RAPID_FEED	Rapid feed in mm/min for positioning on a circular path between the circumferential grooves or the contour elements.
= 10000	

9.4.3 Setting-up ShopMill cycles for milling

Load ShopMill cycles for milling

When the control boots, the ShopMill cycles are automatically loaded into the following directory: ../siemens/sinumerik/cycles/sm/prog

Setting the coordinate system

MD52000 \$MCS_DISP_COORDINATE_SYSTEM	Position of the coordinate system
= 0	

Examples:

=0 for vertical milling machines

=16 for horizontal milling machines, boring mill

A detailed description of the MD52000 is provided in Section:

Activating turning/milling/drilling/grinding technologies (Page 129)

For plane selection when milling, appropriately set the following machine data:

MD52005 \$MCS_DISP_PLANE_MILL	Plane selection for milling
= 17	
= 0	In the cycle support
= 17	G17 (default value)
= 18	G18
= 19	G19

Simulation and simultaneous recording

In order that simulation and simultaneous recording are displayed without any errors, set the machine data as described in the following section:

Setting the technology for simulation (Page 121)

Defining the direction of rotation

In order that the direction of rotation is correctly displayed in the ShopMill user interface, and when programming ShopMill functions, the correct direction of rotation is executed, you must make some settings that are coordinated with one another. You must align these settings to the actual direction of rotation of the axis at the machine.

MD52207 \$MCS_AXIS_USAGE_ATTRIB[n]	Attribute of the axes in the channel [n] channel axis number
Bit 4	Displayed direction of rotation for M3 is counter-clockwise (for spindles)
Bit 5	Direction of rotation M3 corresponds to rotary axis minus (for spindles) This bit must be set analog to PLC bit DB38xx.DBX2001.6.

Defining other direction of rotation for position pattern

You can define a position pattern for the tool axis. Instead of bit 12 you have to set bit 10 to active in MD52207 so that, e.g. the position pattern can be machined on the rotary axis C with an appropriate swivel position.

MD52207 \$MCS_USAGE_ATTRIB[n]	Attribute of the axes in the channel [n] channel axis number
Bit 2	Rotates around 3rd geometry axis (for rotary axes)
Bit 4	Displayed direction of rotation for M3 is counter-clockwise (for spindles) The viewing direction can be selected. Either from the inside looking out or from the outside looking toward the spindle. However the selection must be the same for all spindles.
Bit 5	Direction of rotation for M3 corresponds to rotary axis minus (for spindles)
Bit 6	Display rotary axis as correction target for measuring
Bit 7	Offer rotary axis in the position pattern The orientation of the axis is defined by bits 0 to 2. If one of the bits 10 to 12 is set, then these bits define the orientation of the axis for position patterns.

Bit 8	Offer rotary axis for blank clamping (on milling machines) The orientation of the axis is defined by bits 0 to 2.
Bit 10	The rotary axis turns around the 1st geometry axis (only for position pattern)

Additional channel-specific machine data

MD52216 \$MCS_FUNCTION_MASK_DRILL	Drilling function screen form
Bit 2	Tapping also without encoder The setting depends on whether at the machine the tool spindle is equipped with an encoder.
= 0	Tapping only possible with encoder
= 1	Tapping also possible without encoder
Bit 3	Constant cutting speed referred to the diameter of the centering
= 0	Constant cutting speed referred to the diameter of the tool
= 1	Constant cutting speed referred to the diameter of the centering

MD52229 \$MCS_ENABLE_QUICK_M_CODES	Enable fast M commands
= 0	
Bit 0	Coolant OFF (default value)
Bit 1	Coolant 1 ON
Bit 2	Coolant 2 ON
Bit 3	Coolants 1 and 2 ON

MD52230 \$MCS_M_CODE_ALL_COOLANTS_OFF	M code for all coolants Off
= 9	You define the M function to switch off the coolant that is output when the tool is changed.

MD52231 \$MCS_M_CODE_COOLANT_1_ON	M code for coolant 1 On
= 8	You define the M function for coolant 1 that is output when the tool is changed.

MD52232 \$MCS_M_CODE_COOLANT_2_ON	M code for coolant 2 On
= 7	You define the M function for coolant 2 that is output when the tool is changed.

MD52233 \$MCS_M_CODE_COOLANT_1_AND_2_ON	M code for both coolants On
= -1	You define the M function for coolant 1 and 2 that is output when the tool is changed.

MD52281 \$MCS_TOOL_MCODE_FUNC_ON[]	M code for tool-specific function ON
= -1	M function is not output. If both M commands of a function "= -1", the corresponding field is not displayed on the interface.
[0]	M code for tool-specific function 1 ON
[1]	M code for tool-specific function 2 ON

[2]	M code for tool-specific function 3 ON
[3]	M code for tool-specific function 4 ON

MD52282 \$MCS_TOOL_MCODE_FUNC_OFF[]	M code for tool-specific function OFF
= -1	M function is output. If both M commands of a function "= -1", the corresponding field is displayed on the interface.
[0]	M code for tool-specific function 1 OFF
[1]	M code for tool-specific function 2 OFF
[2]	M code for tool-specific function 3 OFF
[3]	M code for tool-specific function 4 OFF

Channel-specific cycle setting data

SD55212 \$SCS_FUNCTION_MASK_TECH_SET	Cross-technology function screen
= 6	
Bit 0	Tool preselection active. The next tool is prepared directly after a tool change. Notice: For a revolver, the setting data must be set to "0".
Bit 1	Automatic calculation of the thread depth for metric threads.
Bit 2	Take the thread diameter and thread depth from the table.

9.4.4 Cylinder surface transformation (TRACYL_K/TRACYL)



Software option

You require the following software option in order to use this function:
"Transmit and peripheral surface transformation".

Precondition

- There must be at least one rotary axis at the machine.
- The milling tool must be radially oriented to the cylinder to be machined.

Function

The following groove machining operations can be performed with the Cylinder surface transformation functions:

- Longitudinal grooves on cylindrical bodies
- Transverse grooves on cylindrical objects
- Grooves with any path on cylindrical bodies

The path of the grooves is programmed with reference to the unwrapped, level surface of the cylinder. Programming can be realized using straight line/circle, drilling or milling cycles or contour milling (free contour programming).

There are two variants of cylinder surface transformation, i.e.

1. with groove side offset (ON)
2. without groove side offset (OFF)

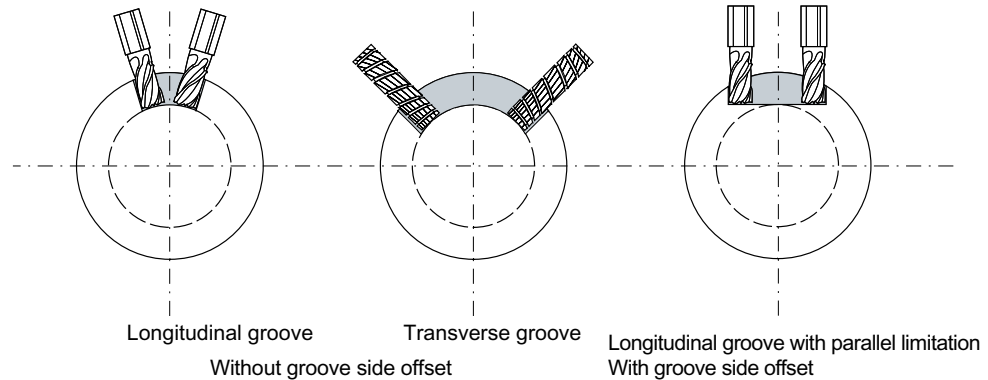
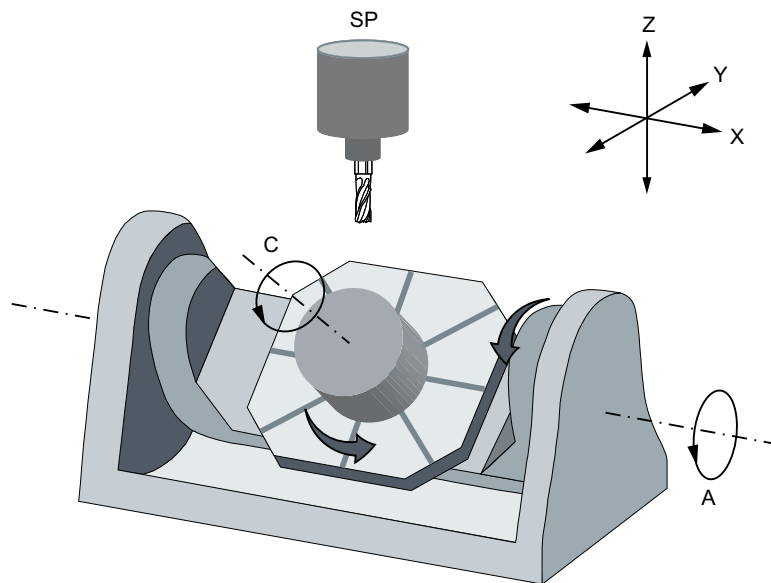


Figure 9-3 Grooves with and without groove side offset

9.4.5

Example: Milling machine with general axis configuration and TRACYL

Example of a general axis configuration and activation of TRACYL on the following machine:



Axis configuration

For the machine shown with rotary axis A and C, you must configure the following machine data:

MD10000 \$MN_AXCONF_MACHAX_NAME_TAB[]		Machine axis name
[0] = MX1	1st machine axis	
[1] = MY1	2nd machine axis	
[2] = MZ1	3rd machine axis	
[3] = MSP1	4th machine axis	
[4] = MA1	5th machine axis	
[5] = MC1	6th machine axis	

MD35000 \$MA_SPIND_ASSIGN_TO_MACHAX[]		Assignment of spindle to machine axis
[AX1] = 0	X is not a spindle	
[AX2] = 0	Y is not a spindle	
[AX3] = 0	Z is not a spindle	
[AX4] = 1	MSP1 spindle 1	
[AX5] = 0	A is not a spindle	
[AX6] = 0	C is not a spindle	

MD20050 \$MC_AXCONF_GEOAX_ASSIGN_TAB[]		Assignment of geometry axis to channel axis
[0] = 1	X is channel axis 1 if TRACYL is not active	
[1] = 2	Y is channel axis 2 if TRACYL is not active	
[2] = 3	Z is channel axis 3 if TRACYL is not active	

MD20060 \$MC_AXCONF_GEOAX_NAME_TAB[]		Geometry axis name in the channel
[0] = X	Geometry axis 1	
[1] = Y	Geometry axis 2	
[2] = Z	Geometry axis 3	

MD20070 \$MC_AXCONF_MACHAX_USED[]		Machine axis number valid in channel
[0] = 1	X as machine axis 1	
[1] = 2	Y as machine axis 2	
[2] = 3	Z as machine axis 3	
[3] = 4	SP1 as machine axis 4	
[4] = 5	A as machine axis 5	
[5] = 6	C as machine axis 6	

MD20080 \$MC_AXCONF_CHANAX_NAME_TAB[]		Name of channel axis in the channel
[0] = X1	Channel axis name	
[1] = Y1	Channel axis name	

[2] = Z1	Channel axis name
[3] = SP	Channel axis name
[4] = A	Channel axis name
[5] = C	Channel axis name

Preparation for TRACYL

MD24200 \$MC_TRAFO_TYPE_2[]	Definition of 2nd transformation in channel
= 513	Transformation TRACYL with groove wall offset

MD24210 \$MC_TRAFO_AXES_IN_2[]	Axis assignment for transformation 2
[0] = 3	Channel axis radial to rotary axis
[1] = 6	Channel axis in cylinder surface (rotary axis)
[2] = 2	Channel axis parallel to rotary axis
[3] = 1	Channel axis special axis to index [0]

MD24220 \$MC_TRAFO_GEOAX_ASSIGN_TAB_2[]	Assignment of geometry axes to channel axes for transformation 2
[0] = 6	6th channel axis becomes GEOAX X with TRACYL
[1] = 2	2nd channel axis becomes GEOAX X with TRACYL
[2] = 3	3rd channel axis becomes GEOAX X with TRACYL

MD24850 \$MC_TRACYL_ROT_AX_OFFSET_2	Offset of the rotary axis for the 2nd TRACYL transformation
= 0	Offset of rotary axis from home position

MD24860 \$MC_TRACYL_ROT_SIGN_IS_PLUS_2	Sign of the rotary axis for the 2nd TRACYL transformation
= 0	Sign of rotary axis

MD24870 \$MC_TRACYL_BASE_TOOL_2[]	Vector of the base tool for the 2nd TRACYL transformation
[0] = 0	Tool basic offset in X
[1] = 0	Tool basic offset in Y
[2] = 0	Tool basic offset in Z

Example part program

Program code	Comment
WORKPIECE(, "", "CYLIN- DER", 0, 0, -180, -80, 179)	;blank definition

Program code	Comment
N10 M3 S2000	;spindle speed defined
T="SLOTMILL" M6 D1	;tool selection
N20 G0 G54 X0 Y-20 Z105	;positioning
CYCLE800(0,"TA- BLE",100000,57,0,0,0,-90,0,0,0,0,-1,100 ,1)	;rotate A-axis with swivel cycle
N40 G17 G90	;machining plane defined
N70 G0 Y-10 Z100 G40	;positioning
N74 TRACYL(179, 2)	;selection of TRACYL data set 2 with groove wall offset
OFFN=20	;offset defined (half groove width)
N80 G1 F500 X0 Z75 G42	;starting point and milling radius offset defined
N90 G1 Y50	;groove center path
N112 X-60	;groove center path
N114 X0	;groove center path
N116 Y-10	;groove center path
N130 G1 Z105 G40	;retraction and deselection of milling radius offset
N150 TRAFOOF	;deselection of transformation
N160 G0 X0 Y-20 Z115	;positioning of retract movement
M5	;spindle STOP
CYCLE800(0,"TA- BLE",100000,57,0,0,0,0,0,0,0,0,-1,100,1)	;turn A axis back
N170 M30	;end of program

9.4.6 ShopMill cycles for multiple clamping



Software options

You require the following software option in order to use this function: "ShopMill/ShopTurn"

Function

The Multiple clamping function provides optimized tool change over several workpiece clampings. On the one hand, this reduces downtimes and, on the other, eliminates tool change times because a tool performs as many machining operations as possible in all clampings before the next tool change is initiated.

You can either execute the same program several times for the clampings or you can select different programs. If you are using large-area fixture plates on your machine, you do not have to set up anything else.

In the case of rotating clamping devices, on the other hand, you must adapt a cycle to match the features of the clamping device so as to ensure that the next workpiece can be turned to the

machining position after machining of the previous one (or for multiple clamping devices even while the current workpiece is being machined).

Procedure for setting up

1. Change the CLAMP.SPF cycle which is stored in the cycle/sc/prog/templates_deu or cycle/sc/prog/templates_eng directory.
2. Copy the cycle into the user or manufacturer directory.

Example

A rotating clamping device (reversible clamping device) is used with four clampings. The respective clampings can be machined by positioning rotary axis A:

Clamping 1:	$A = 0^\circ$
Clamping 2:	$A = 90^\circ$
Clamping 3:	$A = 180^\circ$
Clamping 4:	$A = 270^\circ$

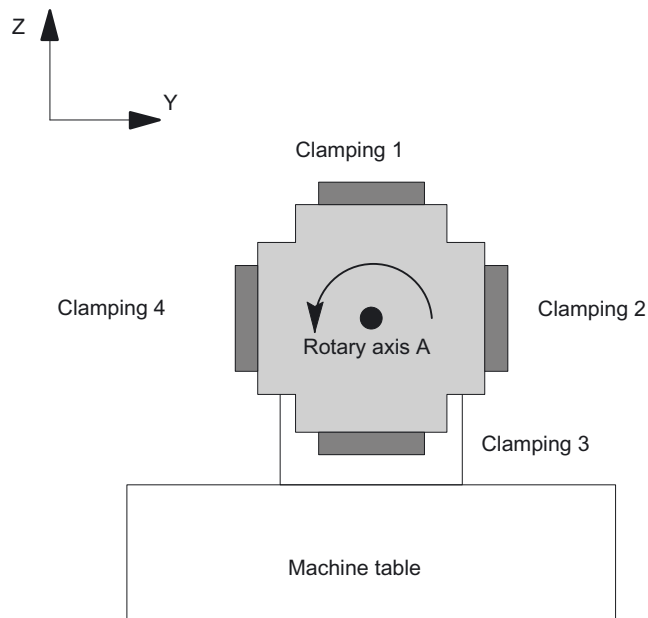


Figure 9-4 Rotating clamping device (reversible clamping device)

The cycle CLAMP.SPF must be adapted as follows:

```
...
DEF INT _NV ; Auxiliary variable
;-----
;Adaptation
;
IF _ACT==1
```

```

G0 A=DC (0)
ENDIF
;
IF _ACT==2
G0 A=DC (90)
ENDIF
;
IF _ACT==3
G0 A=DC (180)
ENDIF
;
IF _ACT==4
G0 A=DC (270)
ENDIF
;
; -----
_NV=_NPV+_ACT ; Calculate current zero offset
N10 G[8]=_NV ; There must not be a calculation here
RET

```

9.4.7 Activating/deactivating Surface Turning (CYCLE953)

Function

The "Surface Turning" function optimizes the spiral turning of freeform surfaces on a milling machine (e.g. for mobile phone cases). During this process, a typical turning pattern arises on the surface of the workpiece.



Software option

To use this function, you need the software option:
"Surface Turning"

You can enable or disable Surface Turning at startup of the control system in the startup menu. Only after this procedure can you set the software option in SINUMERIK Operate.

Note

Avoid functional errors on the controller

Always activate Surface Turning before you read in an archive with corresponding cycle data at the control.

Additional information

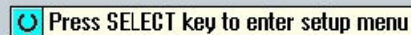
Additional information on using Surface Turning (CYCLE953) and its program structure is provided under:

- Operating Manual, Milling
- Programming Manual, NC Programming

Activating or deactivating Surface Turning

Procedure:

1. After the control is switched on, the following is displayed:



2. The default setting during startup is "Normal startup": Press the <SELECT> key. To call the "Startup menu", press the following keys in succession:
Menu back key, **HSK2** (horizontal softkey 2), **VSK2** (vertical softkey 2)

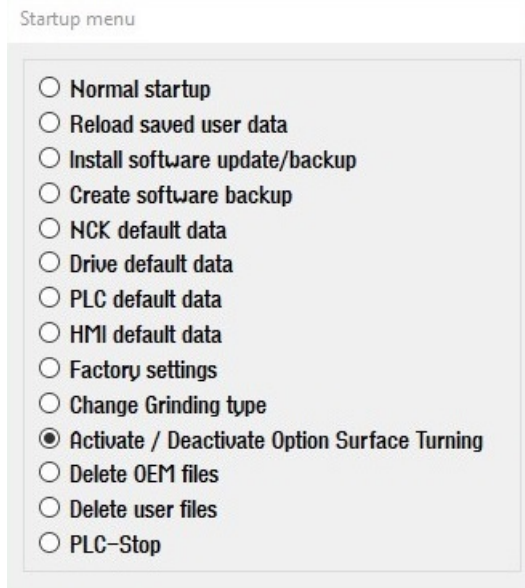


Note

PPU with touch operation

To call the Startup menu during run-up, there is an additional shortcut for all PPUs: **"8" → "2" → "8"**

3. Use the cursor keys to select the "Activate / Deactivate Option Surface Turning" menu command.



4. Press the <INPUT> key to confirm the selection.
A confirmation prompt is displayed before switching over.
5. Acknowledge the confirmation prompt.
Surface turning is now enabled or disabled on the control system.

Note

Re-commissioning of NC and drives after the changeover

Due to the cycle changeover after each activation or deactivation, the subsequent run-up is performed with standard data. Always carry out a new commissioning of the NC and drive data after a function change.

9.5 Turning

9.5.1 General

All settings for machine and setting data of the standard cycles are also applicable for milling under ShopMill and turning under ShopTurn.

9.5.2 Setting up cycles for turning

Setting the coordinate system

MD52000 \$MCS_DISP_COORDINATE_SYSTEM	Position of the coordinate system
= 0	Default setting

Examples:

0: Vertical turning machine (carousel-type turning machine)

19: Horizontal turning machine, machining in front of the turning center

34: Horizontal turning machine, machining behind the turning center (inclined bed turning machine)

A detailed description of MD52000 is given in Section: "Activating turning/milling/drilling/grinding technologies (Page 129)."

Turning function screen

Enable various functions under the Turning function screen in the following channel-specific configuration machine data.

MD52218 \$MCS_FUNCTION_MASK_TURN	Turning function screen
Bit 0	Enable zoom under manual for tool measurement
Bit 1	Enable part catcher for cut-off
Bit 2	Enable tailstock
Bit 3	Enable spindle control of main spindle via user interface
Bit 4	Enable spindle control of the counterspindle control via user interface
Bit 5	Enable spindle control of the tool spindle via user interface
Bit 6	Enable balance cutting for two-channel stock removal
Bit 7	Retraction when cutting along the contour with G1
Bit 8	Enter the spindle chuck data in the program
Bit 9	Additional input of the tailstock data in the program
Bit 12	Deactivate thread synchronization
Bit 13	Cutting along the contour with CYCLE95 (828D programGUIDE without Advanced Technology

Meaning of the axes

You set the meaning of the axes in machine data MD52206 \$MCS_AXIS_USAGE[n].

MD52206 \$MCS_AXIS_USAGE[n]	Meaning of the axes in the channel [n] channel axis number
-----------------------------	---

A detailed description of MD52206 is provided in Chapter "Activating turning/milling/drilling/grinding technologies (Page 129)

Traversing direction

In order that the user interface and cycles know in which direction the spindles and rotary axes rotate, the following settings must be made.

You set the direction of rotation of an NC rotary axis via MD32100 \$MA_AX_MOTION_DIR.

MD32100 \$MA_AX_MOTION_DIR	Traversing direction
-1	Direction reversal
0, 1	No direction reversal

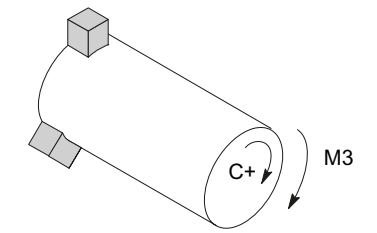
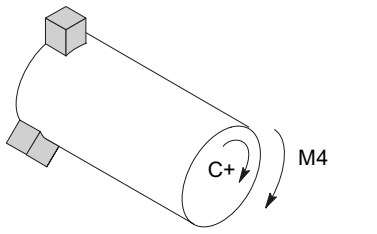
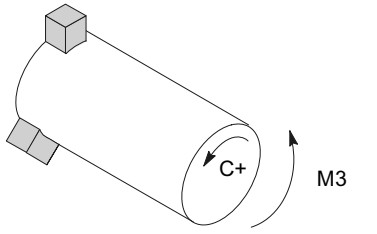
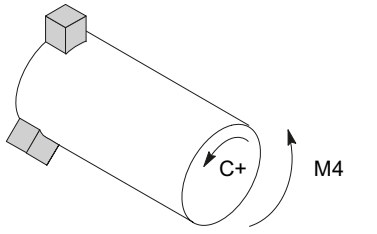
Defining the direction of rotation

In order to ensure that the rotational direction of the spindle and C axis is correctly displayed on the user interface and when programming, the correct direction of rotation is executed, you must make several settings that are coordinated with one another. You must base these settings on the actual direction of rotation of the spindle/C axis on the machine.

You specify the settings in the following machine data:

MD52207 \$MCS_AXIS_USAGE_ATTRIB[n]	Attributes of the axes [n] channel axis number
Bit 3	Displayed positive direction of rotation is counter-clockwise (for rotary axes) The view must be from the inside of the machine (operator's view) to the axis of rotation.
Bit 4	Displayed direction of rotation for M3 is counter-clockwise (for spindles) This bit can be set from the operator's view or according to DIN 66025.
Bit 5	Direction of rotation for M3 corresponds to minus rotary axis (for spindles) You must set this bit as for PLC bit DB38xx.DBX2001.6.

As a result, the following setting options are obtained for the main spindle:

Main spindle direction of rotation	52207[n] bit 3 =	52207[n] bit 4 = *)	52207[n] bit 5 =	DB38xx DBX2001.6
	0	0	0	0
	0	1	1	1
	1	1	0	0
	1	0	1	1

Note

MD52207[n] bit 4 = *)

Bit 4 can be set from the operator's view (values as specified) or according to DIN ("0" and "1" are then interchanged).

Dimensions

To define the reference point for moving the counterspindle, you must first communicate the dimensions of the counterspindle. You can either enter the dimensions in the following axis-specific cycle machine data or in the menu "Parameter" -> Setting data -> Spindle chuck data". Changes to the machine data are automatically accepted in the menu and vice versa.

MD53240 \$MAS_SPINDLE_PARAMETER[]	Spindle chuck data
[0]	Chuck dimensions

9.5 Turning

[1]	Stop dimensions
[2]	Jaw dimensions

MD53241 \$MAS_SPINDLE_CHUCK_TYPE[]	Spindle jaw type
= 0	Clamping, outer
= 1	Clamping, inner

Additional settings

The M code, e.g. M34 or M1 = 34 for the spindle chuck is defined in the following machine data. The manufacturer cycle CUST_TECHCYC.SPF also takes the M functions from the following machine data:

MD52250 \$MCS_M_CODE_CHUCK_OPEN[]	M code for open chuck with stationary spindle
[0]	Main spindle
[1]	Counterspindle

MD52251 \$MCS_M_CODE_CHUCK_CLOSE_OPEN_ROT[]	M code for open chuck with spindle rotating
[0]	Main spindle
[1]	Counterspindle

MD52252 \$MCS_M_CODE_CHUCK_CLOSE[]	M code for close chuck
[0]	Main spindle
[1]	Counterspindle

MD52253 \$MCS_M_CODE_TAILSTOCK_FORWARD[]	M code for quill forward
[0]	Quill opposite the main spindle
[1]	

MD52254 \$MCS_M_TAILSTOCK_BACKWARD[]	M code for quill back
[0]	Quill opposite the main spindle
[1]	Quill opposite the main spindle

Contour turning (CYCLE952)

SD55212 \$SCS_FUNCTION_MASK_TECH_SET	General function screen for all technologies
= 6	
Bit 3	Delete programs generated by contour cycles (CYCLE63, CYCLE64, CYCLE952)
= 0	Generated programs are not deleted (compatibility as before)
= 1	Generated programs are deleted as soon as they have been executed by the calling cycle.

Rounding the contour

SD55582 \$SCS_TURN_CONT_TRACE_ANGLE		Contour turning: Minimum angle for rounding on the contour
= 5	Specifies the angle between the cutting edge and contour, above which for contour turning, the contour is rounded in order to remove residual material (default value).	

9.5.3 Setting-up ShopTurn cycles for turning

MD52210 \$MCS_FUNCTION_MASK_DISP		Function mask for display
Bit 0 = 1	Measuring system for programs always in the base system (default setting)	
Bit 1 = 1	Face view when turning (default setting)	
Bit 4 = 1	Show follow-on tool in T, F, S window	

MD52212 \$MCS_FUNCTION_MASK_TECH		General function mask for all technologies
Bit 2	Approach logic for step drill	
Bit 3	Block search cycle for ShopMill / ShopTurn	
= 0	The E__ASUB or F__S_ASUB cycles are not called in the block search cycle PROG_EVENT.SPF	
= 1	The E__ASUP (under ShopMill) or F__S_ASUP (under ShopTurn) cycles are not called in the block search cycle PROG_EVENT.SPF	
Bit 4	Approach logic using the cycle	
Bit 5	The cycle for SERUPRO (CYCLE207) is called in the block search cycle PROG_EVENT.SPF.	
Bit 6	Work offset value WO cannot be entered	
Bit 7	Identification of the end of tool lives in the program (ShopMill / ShopTurn)	
Bit 8	Manual machine (ShopMill/ShopTurn)	
Bit 9	Selection/deselection of work offset via softkey	
Bit 11	Deactivate position check for drilling and milling tools	
Bit 12	Alarm at positive Z offset	

MD52214 \$MCS_FUNCTION_MASK_MILL		Function mask for milling
Bit 3	"Inner/rear" machining is enabled in the ShopTurn masks which themselves define the machining plane.	
Bit 4	If you have implemented the "Clamp/release spindle" function using the manufacturer cycle CUST_TECHCYC.SPF, then using this machine data, you can activate the "Clamp/release spindle" parameter in the drilling and milling masks.	
= 0	The "Clamp/release spindle" parameter is not displayed in the drilling and milling masks. ShopTurn automatically clamps the spindle if it makes sense for the particular machining operation.	
= 1	The "Clamp/release spindle" parameter is displayed in the drilling and milling masks. You decide for which machining operation the spindle should be clamped.	
Bit 8	Release damping brake (C axis)	

9.5 Turning

= 0	"Damping brake on/off" parameter is not displayed in the milling masks.
= 1	The "Damping brake on/off" parameter is displayed in the milling masks. You decide for which machining operation (Face C/Per.surf.C) the C axis is braked. If you have implemented the "Damping brake" function by means of the machine manufacturer cycle CUST_TECHCYC.SPF, you can activate the "Damping brake on/off" parameter in the milling masks with this machine data.

MD52216	\$MCS_FUNCTION_MASK_DRILL	Function mask for drilling
Bit 3	Constant cutting speed referred to the diameter of the centering	
= 0	Constant cutting speed referred to the diameter of the tool	
= 1	Constant cutting speed referred to the diameter of the centering	

MD52218	\$MCS_FUNCTION_MASK_TURN	Function mask for turning
Bit 2	Enable tailstock	
Bit 5	Enable spindle control of tool spindle via surface	
Bit 6	Enable balance cutting for dual-channel stock removal	

MD52229	\$MCS_ENABLE_QUICK_M_CODES	Enable fast M functions
Bit 0	Coolant OFF	
Bit 1	Coolant 1 ON	
Bit 2	Coolant 2 ON	
Bit 3	Coolants 1 and 2 ON	

MD52230	\$MCS_M_CODE_ALL_COOLANTS_OFF	M code for all coolants OFF
= 9	This machine data is used to define the M function for switching off the coolant; which is output when the tool is changed.	

MD52231	\$MCS_M_CODE_COOLANT_1_ON	M code for coolant 1 ON
= 8	This machine data is used to define the M function for coolant 1, which is output when the tool is changed.	

MD52232	\$MCS_M_CODE_COOLANT_2_ON	M code for coolant 2 ON
= 7	This machine data is used to define the M function for coolant 2, which is output when the tool is changed.	

MD52233	\$MCS_M_CODE_COOLANT_1_AND_2_ON	M code for both coolants ON
= -1	This machine data is used to define the M function for coolant 1 and 2, which is output when the tool is changed.	

SD55505 \$SCS_TURN_ROUGH_O_RELEASE_DIST		Return distance stock removal for external machining
= 1	Specifies the distance by which the tool is retracted from the contour when removing stock from an outer corner. This does not apply to stock removal at a contour (default value).	
= -1	The distance is internally defined.	

SD55506 \$SCS_TURN_ROUGH_I_RELEASE_DIST		Return distance stock removal for internal machining
= 0.5	Specifies the distance by which the tool is retracted from the contour when removing stock from an internal corner. This does not apply to stock removal at a contour (default value).	
= -1	The distance is internally defined.	

9.5.4 Setting up a counterspindle



Software option

You require the following software option in order to use the counterspindle on your machine:

"Generic coupling 'CP-STATIC'" or higher

Function

If your turning machine has a counterspindle, you can machine workpieces using turning, drilling and milling functions on the front and rear faces without reclamping the workpiece manually.

Before machining the rear face, the counterspindle must grip the workpiece, pull it out of the main spindle, and position it at the new machining position.

Meaning of the axes

MD52206 \$MCS_AXIS_USAGE[n]	Meaning of the axes in the channel [n] channel axis number
-----------------------------	---

A detailed description of MD52006 is provided in Chapter "Activating turning/milling/drilling/grinding technologies (Page 129)".

Traversing direction

In order that the user interface and cycles know in which direction the spindles and rotary axes rotate, the following settings must be made.

You set the direction of rotation of an NC rotary axis via MD32100 \$MA_AX_MOTION_DIR.

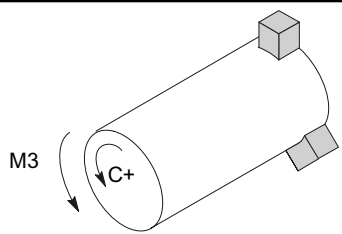
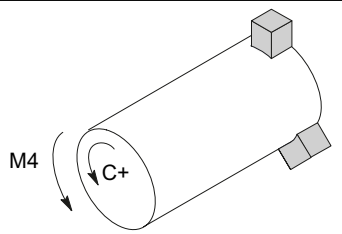
MD32100 \$MA_AX_MOTION_DIR		Traversing direction (not control direction)
-1	Direction reversal	
0, 1	No direction reversal	

Defining the direction of rotation

You set the directions of rotation for the user interface in the following machine data:

MD52207 \$MCS_AXIS_USAGE_ATTRIB[n]		Attributes of the axes in the channel [n] channel axis number
Bit 3	Displayed positive direction of rotation is counter-clockwise (for rotary axes) The view must be from the inside of the machine (operator's view) to the axis of rotation.	
Bit 4	Displayed direction of rotation for M3 is counter-clockwise (for spindles) This bit can be set from the operator's view or according to DIN.	
Bit 5	Direction of rotation for M3 corresponds to minus rotary axis (for spindles) You must set this bit as for PLC bit DB38xx.DBX2001.6.	

As a result, the following setting options are obtained for the counterspindle:

Counterspindle direction of rotation	52207[n] bit 3 =	52207[n] bit 4 = *)	52207[n] bit 5 =	DB38xx. DBX2001.6
	1	1	0	0
	1	0	1	1

Counterspindle direction of rotation	52207[n] bit 3 =	52207[n] bit 4 = *)	52207[n] bit 5 =	DB38xx. DBX2001.6
M3	0	0	0	0
M4	0	1	1	1

Note**MD52207[n] bit 4 = *)**

Bit 4 can be set from the operator's view (values as specified) or according to DIN ("0" and "1" are then interchanged).

The display must be the same for the main spindle and the counterspindle ("from operator's view" or "according to DIN").

Dimensions

To define the reference point for moving the counterspindle, you must first announce the dimensions of the counterspindle. You can either enter the dimensions in the following axis-specific cycle machine data or in menu "Parameter" → "Setting data" → "Spindle chuck data." Changes to the machine data are automatically accepted in the menu and vice versa.

MD53240 \$MAS_SPINDLE_PARAMETER[]		Spindle chuck data
[0]	Chuck dimensions	
[1]	Stop dimensions	
[2]	Jaw dimensions	

MD52241 \$MCS_SPINDLE_CHUCK_TYPE		Spindle jaw type
= 0	Clamping, outer	
= 1	Clamping, inner	

9.5.5 Setting up the counterspindle under ShopTurn



Software option

You require the following software option in order to use the counterspindle on your machine:

"Generic coupling 'CP-STATIC'" or higher

The position to which the counterspindle travels when the program starts is defined in the following channel-specific cycle setting data:

SD55232 \$SCS_SUB_SPINDLE_REL_POS	Retraction position Z for counterspindle
-----------------------------------	--

The following channel-specific setting data become effective when traveling to the fixed stop:

SD55550 \$SCS_TURN_FIXED_STOP_DIST	Distance for travel to fixed stop.
SD55551 \$SCS_TURN_FIXED_STOP_FEED	Feedrate for travel to fixed stop.
SD55552 \$SCS_TURN_FIXED_STOP_FORCE	Force for travel to fixed stop in %.

Between traveling to the fixed stop and gripping, the counterspindle can retract a short distance to counteract compressive stress in the workpiece.

SD55553 \$SCS_TURN_FIXED_STOP_RETRACTION	Retraction distance before clamping after fixed stop.
--	---

After gripping you can cut off the workpiece. Before doing so, the counterspindle can retract a short distance with the workpiece to exert tensile stress on the workpiece. This relieves pressure on the tool when cutting off.

SD55543 \$SCS_TURN_PART_OFF_RETRACTION	Retraction distance before cut-off.
--	-------------------------------------

After parting, you can carry out a cut-off check and for turning, use the "Travel to fixed stop" function. You can activate/deactivate the cut-off check using the channel-specific cycle setting data:

SD55540 \$SCS_TURN_PART_OFF_CTRL_DIST	Distance for cut-off check.
SD55541 \$SCS_TURN_PART_OFF_CTRL_FEED	Feedrate for cut-off check.
SD55542 \$SCS_TURN_PART_OFF_CTRL_FORCE	Force for cut-off check in %.

The cut-off is successful when travel to fixed stop fails. The following alarms are output:

Alarm	Alarm text
20091	Axis %1 has not reached the fixed stop.
20094	Axis %1 end stop has been aborted.

You can switch off the alarm display using the following machine data:

MD37050 \$MA_FIXED_STOP_ALARM_MASK	Enabling the fixed stop alarms.
= 2	Suppressing alarms 20091 and 20094.

You can set this machine data axis-specifically in the "Machine Data" window in the "Tool zero" operating area.

If, however, the specified force is reached during the cut-off check (i.e. travel to fixed stop is successful), alarm 61255 "Error during cut-off: Tool break?" is issued.

Note

The "Travel to fixed stop" function can also be used when gripping the spindle (see above). If travel to fixed stop does not succeed when gripping, an alarm will of course also be issued. Instead of alarms 20091 and 20094, the alarm 61254 "Error during travel to fixed stop" will be issued.

Machine manufacturer cycle

If you perform one of the following actions, you must adapt the machine manufacturer cycle CUST_TECHCYC.SPF.

- Switch between spindle and C axis mode for the main spindle or counterspindle.
- Open, close, flush chuck (main spindle / counterspindle).
- Change default settings for the coupling of the main spindle and counterspindle.

9.5.6 Technology cycles for turning

Thread-cutting (CYCLE99)

To correctly machine the thread, it is necessary to set the machine data MD52207 \$MCS_AXIS_USAGE_ATTRIB[n] with bit 3.

You can find **more information** in Chapter "Setting up a counterspindle (Page 169)".

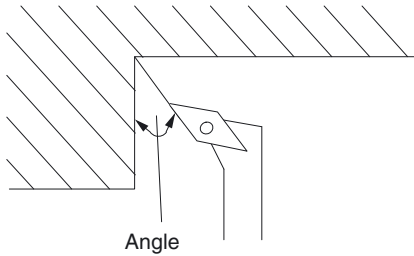
Contour grooving CYCLE930, contour turning CYCLE950, stock removal at corner CYCLE951, contour turning CYCLE952

SD55500 \$SCS_TURN_FIN_FEED_PERCENT	Enter the finishing feedrate for complete machining, roughing and finishing. The percentage of the value corresponds to that entered under parameter F (feedrate).
= 100	100% finishing feedrate

SD55510 \$SCS_TURN_GROOVE_DWELL_TIME	Dwell time, which is necessary between grooving and retracting for grooving technology. Tool clearance time when grooving at the base.
= > 0	Dwell time in seconds
= < 0	Dwell time in spindle revolutions

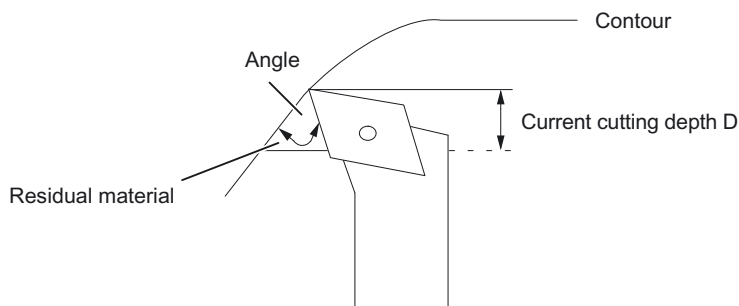
9.5 Turning

SD55580 \$SCS_TURN_CONT_RELEASE_ANGLE	Angle, through which the tool is lifted from the contour for contour turning, roughing.
= 45	Retraction angle of 45 degrees



SD55581 \$SCS_TURN_CONT_RELEASE_DIST	Amount, by which the tool is lifted when roughing a contour taking into account the retraction angle SD55580 \$SCS_TURN_CONT_RELEASE_ANGLE. This also applies to stock removal, grooving and plunge turning.
= 1	1 mm or 1 inch retraction distance

SD55582 \$SCS_TURN_CONT_TRACE_ANGLE	The angle between cutting edge and contour from which rounding is performed on the contour during contour turning in order to remove residual material. If the angle of the residual material is greater than that specified in the setting data, the tool will round the contour.
= 5	5 degree angle



SD55583 \$SCS_TURN_CONT_VARIABLE_DEPTH	Percentage of the changing cutting depth when turning a contour. You can select changing the cutting depth for stock removal and for removing residual material.
= 20	20% variable cutting depth

SD55584 \$SCS_TURN_CONT_BLANK_OFFSET	Safety clearance when avoiding obstacles, to avoid collisions during approach or retraction from the machining. This also applies to stock removal, grooving and plunge turning.
= 1	1 mm or 1 inch blank allowance

SD55585 \$SCS_TURN_CONT_INTERRUPT_TIME	Time for the feedrate interruption for contour turning. This also applies to stock removal, grooving and plunge turning.
= > 0	Interruption time in seconds
= < 0	Interruption time in revolutions
= 0	No interruption

Note

The channel-specific cycle setting data SD55585 is only evaluated if SD55586 \$SCS_TURN_CONT_INTER_RETRACTION = 0.

SD55586 \$SCS_TURN_CONT_INTER_RETRACTION	Retraction distance for contour turning for feedrate interruption. This also applies to stock removal, grooving and plunge turning.
= > 0	Retraction distance at feed interruption SD55585 \$SCS_TURN_CONT_INTERRUPT_TIME has no effect.
= 0	No retraction distance

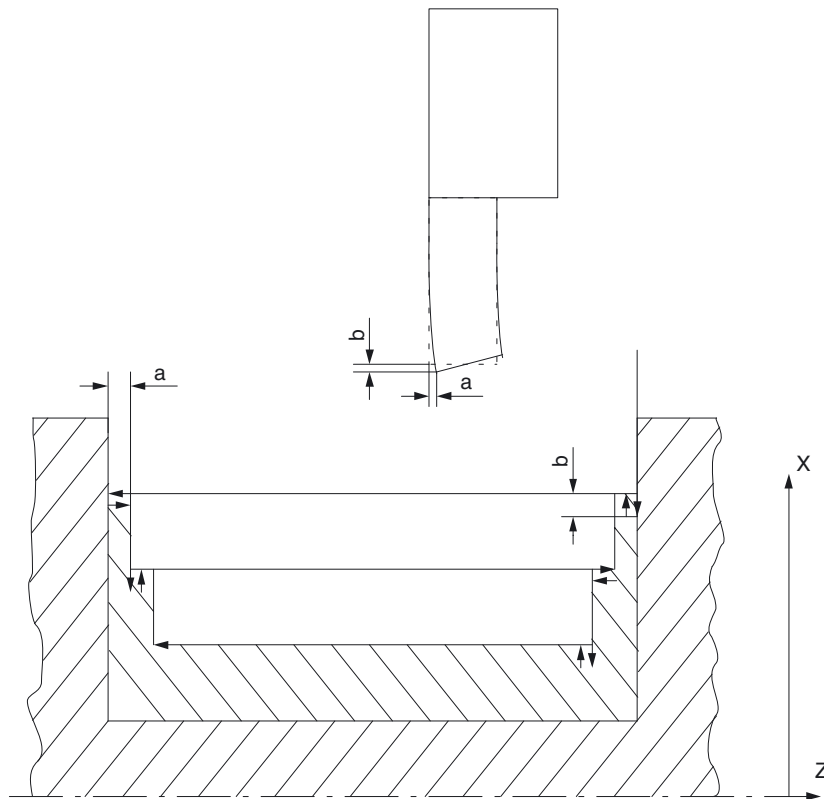
SD55587 \$SCS_TURN_CONT_MIN_REST_MAT_AX1	Limit value during contour turning for removal of residual material in the direction of axis 1 (for G18 Z). This also applies to residual machining – stock removal, grooving and plunge turning.
50	The roughing step for residual machining is not executed, if the material to be removed is less than 50% of the finishing allowance. The material is only removed when finishing.

Example: If the SD is set to 50% and the final machining allowance is 0.5 mm, any residual material less than 0.25 mm is not removed during residual machining – but is removed during finishing. If during a machining step, less residual material is present than defined in the SD, the error message "No material present" is issued. This means that this residual material roughing step can be omitted because no machining is performed.

SD55588 \$SCS_TURN_CONT_MIN_REST_MAT_AX2		Limit value during contour turning for removal of residual material in the direction of axis 2 (for G18 X). This also applies to residual machining – stock removal, grooving and plunge turning.
50	The roughing step for residual machining is not executed, if the material to be removed is less than 50% of the finishing allowance. The material is only removed when finishing.	

As the tool bends during plunge turning, the tool cannot travel right up to the contour during stock removal. The lateral distance to the last cut by which the next cut is shortened is specified in the following channel-specific cycle setting data.

SD55595 \$SCS_TURN_CONT_TOOL_BEND_RETR		Retraction distance because of tool bending
0.1	0.1 mm or 0.1 inch retraction distance	



- a Distance to the last cut, SD55595
b Retraction between plunge-cutting and stock removal, SD55596

As the tool bends during plunge turning, the tool would make an excessively deep cut during stock removal. The retraction distance of the tool between plunge-cutting and stock removal is specified in the following channel-specific cycle setting data:

SD55596 \$SCS_TURN_CONT_TURN_RETRACTION		Retraction depth before turning
= 0.1	0.1 mm or 0.1 inch retraction depth	

9.5.7 Single-channel and two-channel sensors

General configuration

If rotating milling tools are available on a turning machine, the following functions can also be set up on this machine:

- Cylinder surface transformation (TRACYL_K/TRACYL) (Page 180)
- End face machining (TRANSMIT_K/TRANSMIT) (Page 182)

For a single-channel or two-channel turning machine, you can configure the following channel-specific machine data, for example:

Single-channel turning machine with X1 and Z1 axis, main spindle and tool spindle, servo revolver (system 828D-TE42)

MD20080 \$MC_AXCONF_CHANAX_NAME_TAB[]		Channel axis name in the channel
[0] = X1	Channel axis X1	
[1] = Z1	Channel axis Z1	
[2] = C1	Main spindle C1	
[3] = SP1	Tool spindle SP1	
[4] = Q1	Turn tool turret	

Single-channel turning machine with X1, Z1, and Y1 axis, main spindle and tool spindle, servo turret (system 828D-TE62)

MD20080 \$MC_AXCONF_CHANAX_NAME_TAB[]		Channel axis name in the channel
[0] = X1	Channel axis X1	
[1] = Z1	Channel axis Z1	
[2] = C1	Main spindle C1	
[3] = SP1	Tool spindle SP1	
[4] = Y1	Channel axis Y1	
[5] = Q1	Turn tool turret	

Single-channel turning machine with X1, Z1, and Y1 axis, main spindle, counterspindle, and tool spindle, counterspindle axis, servo turret (system 828D-TE821)

MD20080 \$MC_AXCONF_CHANAX_NAME_TAB[]		Channel axis name in the channel
[0] = X1	Channel axis X1	
[1] = Z1	Channel axis Z1	
[2] = C1	Main spindle C1	
[3] = SP1	Tool spindle SP1	
[4] = Y1	Channel axis Y1	
[5] = Q1	Turn tool turret	

[6] = Z2	Counterspindle axis Z2
[7] = C2	Counterspindle C2

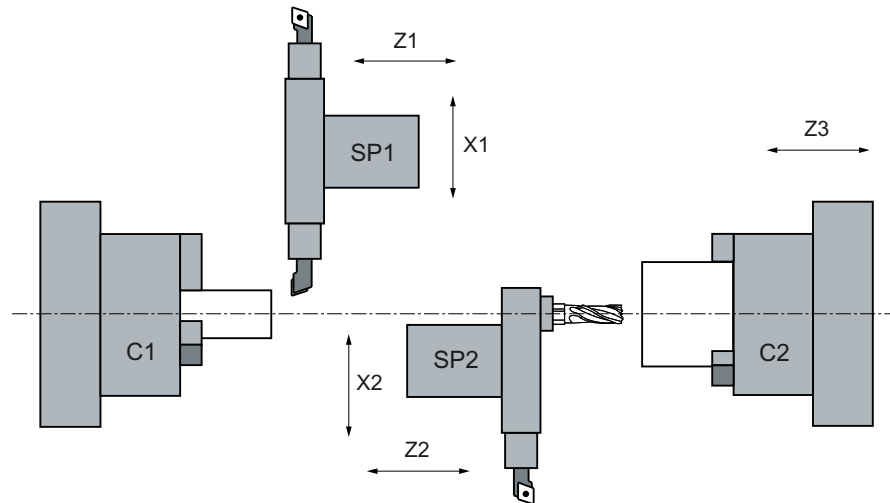
Two-channel turning machine with X1, Z1, and Y1 axis, main spindle and tool spindle, counterspindle axis Z3, X2 and Z2 axis, counterspindle and tool spindle (system 828D-TE822)

MD20080 \$MC_AXCONF_CHANAX_NAME_TAB[]	Channel axis name in channel 1
[0] = X1	Channel axis X1
[1] = Y1	Channel axis Y1
[2] = Z1	Channel axis Z1
[3] = C1	Main spindle C1
[4] = SP1	Tool spindle SP1
[5] = Z3	Counterspindle axis Z3
[6] = C2	Counterspindle C2
[7] = X2	Channel axis X2
[8] = Z2	Channel axis Z2

MD20080 \$MC_AXCONF_CHANAX_NAME_TAB[]	Channel axis name in channel 2
[0] = X2	Channel axis X2
[1] = Z2	Channel axis Z2
[2] = SP2	Tool spindle SP2
[3] = Z3	Counterspindle axis Z3 (default passive)
[4] = SP1	Tool spindle SP1
[5] = C1	Main spindle C1 (default passive)
[6] = C2	Counterspindle C2 (default passive)

9.5.8 Example: Two-channel turning machine with general axis configuration

Example of a general axis configuration on the following turning machine:



Axis configuration

For the two-channel machine shown, you must configure the following machine data:

MD10000 \$MN_AXCONF_MACHAX_NAME_TAB[]		Machine axis name
[0] = MX1	X1 axis in the first channel	
[1] = MY1	-	
[2] = MZ1	X1 axis in the first channel	
[3] = MC1	Main spindle C1	
[4] = MSP1	Tool spindle SP1	
[5] = MZ3	Linear axis toward counter spindle active in channel 1 and 2	
[6] = MC2	Counterspindle C2	
[7] = MX2	X2 axis in second channel	
[8] = MZ2	Z2 axis in second channel	
[9] = MSP2	Tool spindle SP2	

MD20050 \$MC_AXCONF_GEOAX_ASSIGN_TAB[]		Assignment of geometry axis to channel axis for channel 1
[0] = 1		
[1] = 0		
[2] = 3		

MD20050 \$MC_AXCONF_GEOAX_ASSIGN_TAB[]		Assignment of geometry axis to channel axis for channel 2
[0] = 1		

9.5 Turning

[1] = 0	
[2] = 2	

MD20070 \$MC_AXCONF_MACHAX_USED[]	Machine axis number valid in channel 1
[0] = 1	
[2] = 3	
[3] = 4	
[4] = 5	
[5] = 6	
[6] = 7	

MD20070 \$MC_AXCONF_MACHAX_USED[]	Machine axis number valid in channel 2
[0] = 8	
[1] = 9	
[2] = 10	
[3] = 6	
[4] = 4	
[5] = 7	

MD52206 \$MC_AXIS_USAGE[]	Meaning of the axes in channel 1
[3] = 3	Main spindle
[4] = 1	Tool spindle
[5] = 7	Linear axis of the counterspindle
[6] = 5	Counterspindle

MD52206 \$MC_AXIS_USAGE[]	Meaning of the axes in channel 2
[2] = 1	Tool spindle
[3] = 7	Linear axis of the counterspindle
[4] = 3	Main spindle
[5] = 5	Counterspindle

9.5.9 Cylinder surface transformation (TRACYL_K/TRACYL)

**Software option**

You require the following software option in order to use this function:

"Transmit and peripheral surface transformation"

Function

Using the cylinder surface transformation function (TRACYL), you can machine the peripheral surface of a turned part.

Setting up

General settings for cylinder transformation:

MD10602 \$MN_FRAME_GEOAX_CHANGE_MODE	Frames when changing geometry axes.
= 1	The actual total frame (work offsets) and when switching over geometry axes (selecting - deselecting TRACYL) is recalculated.

MD24040 \$MC_FRAME_ADAPT_MODE	Adapting the active frames
Bit 0 = 1	
Bit 1 = 1	
Bit 2 = 1	

MD28082 \$MC_MM_	System frame (SRAM)
SYS- TEM_FRAME_MASK	
Default, cannot be changed	

A cylinder surface transformation can be set up as TRACYL_K (transformation based on a kinematic chain) or as TRACYL (conventional transformation parameterized using machine data). A distinction can be made as to whether the work is done with a slot wall correction (a third linear axis is available) or without.

Additional information

A detailed description and corresponding examples are provided in the Transformations Function Manual.

Setting up cylinder surface transformation under ShopTurn

In addition, set the following channel-specific configuration machine data:

MD52214 \$MC_FUNCTION_MASK_MILL	Function mask for milling
Bit 3	Enable "inside/rear" machining in ShopTurn masks which define the machining plane themselves.
Bit 4	If you have implemented the "Clamp/release spindle" function by means of the machine manufacturer cycle CUST_TECHCYC.SPF, you can activate the "Clamp/release spindle" parameter in the drilling and milling masks with this machine data.
= 0	The "Clamp/release spindle" parameter is not displayed in the drilling and milling masks. ShopTurn automatically clamps the spindle if it makes sense for the particular machining operation.

=1	The "Clamp/release spindle" parameter is displayed in the drilling and milling masks. The operator decides for which machining operation the spindle should be clamped.
Bit 8	Release damping brake (C axis)
= 0	"Damping brake on/off" parameter is not displayed in the milling masks.
= 1	The "Damping brake on/off" parameter is displayed in the milling masks. You decide for which machining operation (Face C/Per.surf.C) the C axis is braked. If you have implemented the "Damping brake" function by means of the machine manufacturer cycle CUST_TECHCYC.SPF, you can activate the "Damping brake on/off" parameter in the milling masks with this machine data.

9.5.10 End face machining (TRANSMIT_K/TRANSMIT)



Software option

You require the following software option in order to use this function:
"Transmit and peripheral surface transformation"

Function

The end face machining function (TRANSMIT) is used to machine the end faces of a turned part.

Setting up

For general settings for transformations, please refer to Chapter "Cylinder surface transformation (TRACYL_K/TRACYL) (Page 180)".

End face machining can be set up as TRANSMIT_K or as TRANSMIT.

Using TRANSMIT_K, end face machining can be defined via a kinematic chain that has been set up; such a transformation can be set up in the operating area "Commissioning > NC > Transformations".

A TRANSMIT is conventionally parameterized using machine data. For end face machining, a distinction is made as to whether a Y axis is available or not.

Additional information

You can find a detailed description and examples in the
Transformations Function Manual.

Face machining under ShopTurn

If you wish to use the face machining function under ShopTurn, set the following additional channel-specific configuration machine data:

MD52214 \$MCS_FUNCTION_MASK_MILL		ShopTurn function screen
Bit 3	Enable "internal / rear" machining in ShopTurn screens that define the machining plane themselves.	
Bit 4	If you have implemented the "Clamp/release spindle" function using the machine manufacturer cycle CUST_TECHCYC.SPF, then using this machine data, you can activate the "Clamp/release spindle" parameter in the drilling and milling masks.	
= 0	The "Clamp/release spindle" parameter is not displayed in the drilling and milling masks. ShopTurn automatically clamps the spindle if it makes sense for the particular machining operation.	
= 1	The "Clamp/release spindle" parameter is displayed in the drilling and milling screens. The operator decides for which machining operation the spindle should be clamped.	

Note

The face end machining is automatically integrated in the cycles, with the exception of the straight line and circle.

You can select the functions for these two cycles in the "Program" operating area under "Straight line" and "Circle".

9.5.11 Inclined Y axis (TRAANG_K/TRAANG)



Software option

You require the following software option in order to use the inclined Y axis function on your machine:

"Inclined axis" or "Inclined axis, fixed angle"

Function

If your turning machine has an inclined Y axis (i.e. this axis is not perpendicular to axes X and Z), you can still completely program machining operations in Cartesian coordinates. The control uses the "Inclined axis function (TRAANG)" to transform the Cartesian coordinates to the motion of the inclined axis.

Setting up

An inclined axis can either be conventionally set up via machine data or parameterized via the kinematic chain.

Additional information

You can find a detailed description and examples in the

Transformations Function Manual.

Inclined axis under ShopTurn

Please also note the following:

Note

In the user interface, the inclined axis function is automatically integrated in the cycles after setting up. You can select the machining plane "Face Y1" or "Surface Y1" for machining with inclined axis in the screens and enter Cartesian coordinates.

Concatenating (TRACON) of end face machining on a main spindle (TRANSMIT) and inclined axis (TRAANG)

In transformations with kinematic chains, non-orthogonal axes can be defined via the kinematic chain. This means that transformations no longer have to be concatenated with TRACON to map non-orthogonal axes.

For conventional parameterization via machine data, two transformations can be connected one after the other via TRACOON.

9.5.12 X zero point not at the rotation center

General information

For some machines, the zero point of the X-axis in the Machine Coordinate System (MCS) is not located at the center of the rotating spindle.

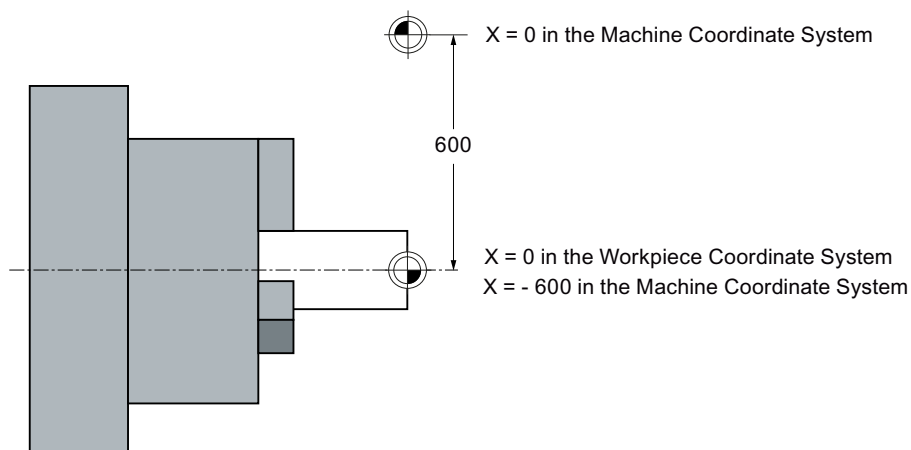


Figure 9-5 X zero point not at the rotation center

If the zero point of the X-axis in the Machine Coordinate System is not located in the center of the rotating spindle, a special commissioning is required.

You must store the position of the main spindle in the following machine data:

MD53220 \$MAS_AXIS_MCS_POSITION[0] [HSP]	Position of the axis in the MCS
= [MCS position]	e.g. - 600

The transformations must also be informed about the position of the rotary axis:

MD24906 \$MC_TRANSMIT_BASE_TOOL_COMP_1	Compensation of the BASE_TOOL in frame TRANSMIT 1
Bit 0 = 1	MD24920 \$MC_TRANSMIT_BASE_TOOL_1[0] compensated via \$P_TRAFRAME .

MD24920 \$MC_TRANSMIT_BASE_TOOL_1 [0]	Vector of the base tool for the 1st TRANSMIT transformation.
= - 600	Value in mm

If necessary, enter the same for further transformations (TRANSMIT, TRACYL).

Note also the following machine data:

MD20360 \$MC_TOOL_PARAMETER_DEF_MASK	Definition of the effect of the tool parameters.
Bit 3 = 0	The work offsets in frames in the transverse axis are not included as diameter value.

Offset in the Z position of the counterspindle

If the zero point of the counterspindle is not located at MCS = 0, you must enter the offset in the following machine data:

MD53220 \$MAS_AXIS_MCS_POSITION[2] [GSP]	Position of the axis in the MCS
= [MCS position]	e.g 1000

Fixed counterspindle

If the counterspindle does not have a traversing axis, you must enter the position of the counterspindle in the following machine data:

MD53220 \$MAS_AXIS_MCS_POSITION[2] [GSP]	Position of the axis in the MCS
= [MCS position]	e.g 800

Any rotary axis

If the machine has an arbitrary rotary axis that is not further defined (e.g. by the tool carrier), you must enter the position in the following machine data:

MD53220 \$MAS_AXIS_MCS_POSITION[n] [RAX]	Position of the axis in the MCS [n] 0 = X, 1 = Y, 2 = Z
--	--

9.5.13 Spindle speed limitation for current program

Speed limits are programmed in the part program. If the part program is applied, the spindle speed limits are entered in the setting data and saved there after the end of program.

However, if the modified speed limits must not apply after the end of the program, you can change the property with the following commands in the GUD block of the machine manufacturer:

Changing spindle speed limits for the current program only

```
REDEF $SA_SPIND_MAX_VELO_LIMS PRLOC ; maximum spindle speed limit for V  
                                         constant (G96)  
REDEF $SA_SPIND_MAX_VELO_G26 PRLOC  ; maximum spindle speed limit for rpm  
                                         (G26)
```

Additional information

You can find additional information about speed limiting in the NC Programming Programming Manual.

9.6 Grinding

General

Grinding tools are identified by tool type 400, 410; dressing tools are identified by tool type 490. The "New tool" function offers these types for selection. The tool list shows the "Grinding disk" and "Dressing tool" tool types listed according to their tool type. To generate the grinding disk correction, a tool setter creates a dressing program. Depending on the complexity, the tool setter makes use of external or internal code generators. This program will be assigned later to a grinding disk.

Oscillation cycles

These grinding cycles permit the machining of workpieces with grinding machines using two geometry axes.

The machines type does not exist because the cycles create only an oscillating movement that can be used as required on every machine.

This requires a SINUMERIK control as well as fast inputs/outputs for the program processing.

The cycles package provides the following cycles:

- CYCLE4071: Longitudinal grinding with infeed at the reversal point
- CYCLE4072: Longitudinal grinding with infeed at the reversal point and measurement control
- CYCLE4073: Longitudinal grinding with continuous infeed
- CYCLE4074: Longitudinal grinding with continuous infeed and measurement control
- CYCLE4075: Surface grinding with infeed at the reversal point
- CYCLE4077: Surface grinding, plunge grinding with measurement control
- CYCLE4078: Surface grinding with continuous infeed
- CYCLE4079: Surface grinding with intermittent infeed

Oscillation cycles: Activate GUD variables for synchronized actions

MD18661 \$MN_MM_NUM_SYNACT_GUD_INT[n]	Number of configurable integer-type GUD variables
> 0	With the machine data, the individual GUD blocks can be extended by additional channel-specific parameter areas of the type integer. [n] is equivalent to the access rights data block

Additional information

You can find additional information about oscillating cycles and GUD variables for synchronized actions in the:

- NC Programming Programming Manual
- Synchronized Actions Function Manual

Grinding function screen

MD51840 \$MNS_GRIND_FUNCTION_MASK		Grinding cycles function screen
Bit 0	Reference point for measuring the dresser/wheel is a work offset	
= 0	Deselection of the work offset as reference point	
= 1	Selection of the work offset as reference point	

MD52840 \$MCS_FUNCTION_MASK		Grinding cycles function screen
Bit 7	Retraction when form-truing with G1	
= 0	Retraction when form-truing with rapid traverse (G0)	
= 1	Retraction when form-truing with feedrate (G1)	

Activate and display fit-dependent corrections

Activate fit-dependent corrections

MD18603 \$MN_MM_NUM_GLOBAL_G_FRAMES		Number of global grinding frames (SRAM)
The value corresponds to the number of field elements for the pre-defined field \$P_GFR[]. If the value of the data is > 0, then all settable frames are only global. MD28079 is then ignored.		

MD28079 \$MC_MM_NUM_G_FRAMES		Number of grinding frames (SRAM)
Defines the number of predefined grinding frames.		

Display fit-dependent corrections in the operating area "Parameters" → "Work offsets" window:

MD52211 \$MCS_FUNCTION_MASK_DISP_ZOA		Function screen Display overview of work offsets
Bit 22	Display fit-dependent corrections (\$P_GFRAME)	

Details of disk size

MD52842 \$MCS_GRIND_DIAMETER_LENGTH		Number of the tool length for the disk diameter
= 1	Length 1 is disk diameter	
= 2	Length 2 is disk diameter	

MD52843	\$MCS_GRIND_WIDTH_LENGTH	Number of the tool length for the disk width
= 1	Length 1 is disk width	
= 2	Length 2 is disk width	

Form-truing (CYCLE495)

SD55880	\$SCS_GRIND_CONT_RELEASE_ANGLE	Angle of retraction
	This setting data is used to specify the angle of retraction from the contour during form-truing.	

SD55881	\$SCS_GRIND_CONT_RELEASE_DIST	Retraction distance
	This setting data is used to specify the distance retracted in both axes during form-truing.	

SD55884	\$SCS_GRIND_CONT_BLANK_OFFSET	Blank allowance
	This setting data is used to specify the distance to the grinding wheel as of which the switch-over is made from G0 to G1 during form-truing.	

9.7 Swiveling

9.7.1 Technology cycles for swiveling

Precondition

The commissioning of the machine kinematics is a mandatory requirement for correct swiveling functionality (CYCLE800). The vectors of the machine kinematics are stored in the tool parameters \$TC_CARR1 to \$TC_CARR65. The machine kinematics are stored in the kinematic chain and be accessed via the \$TC_CARR_KIN... tool parameters. The vectors of the machine kinematics can also be stored in the tool parameters \$TC_CARR1 to \$TC_CARR65 even without the kinematic chain.

Note

Only applies for milling technology:

The vectors of the machine kinematics can be determined with the "Measure kinematics" measuring function (CYCLE996).

A tool carrier is available in the 828D for activating the swivel function.

MD18088 \$MN_MM_NUM_TOOL_CARRIER	Maximum number of definable tool carriers
Default, cannot be changed.	

MD28082 \$MC_MM_SYSTEM_FRAME_MASK	System frames (SRAM)
Default, cannot be changed.	

More information

Additional information can be found in:

- Tools Function Manual
- Transformations Function Manual
- Measuring Cycles Programming Manual

Activating the swivel function

The swivel function is enabled on the user interface via the following channel-machine data:

MD52212 \$MCS_FUNCTION_MASK_TECH	General function mask for all technologies
Bit 0 = 1	Enable swivel

Configuring the input dialog

You can configure the input mask for swiveling using the following channel-specific cycle setting data. The setting data is effective for all of the declared swivel data sets.

SD55221 \$SCS_FUNKTION_MASK_SWIVEL_SET		Function mask for swivel CYCLE800
Bit 0	Input field "No swiveling"	
= 0	Hide	
= 1	Display	
Bit 1	Selection text in retraction "Z", "ZXY" or "Fixed point 1/2"	
= 0	Display text Z = "Z", display text ZXY = "ZXY"	
= 1	Selection text retraction "Fixed point 1" or retraction "Fixed point 2". If you modify the retraction variants "Z" or "ZXY" over the manufacturer cycle CUST_800.SPF, the neutral texts "Fixed point 1" and "Fixed point 2" are displayed in this case.	
Bit 2	Select, permit "deselection" of the swivel data set	
= 0	If deselection is not permitted, the "Swivel data set" (TC) selection field is not displayed in the "Swivel" input mask.	
= 1	Deselection permitted see also swivel data set parameter \$TC_CARR37 HUNDRED MILLIONS position	
Bit 3	Displays the active swivel plane under swivel in JOG. The setting of the Swivel function mask influences the swivel data set.	
Bit 4	Positioning in the basic position (pole position) of the kinematics	
= 0	Evaluation of the input values in pole position of the machine kinematics	
= 1	Compatibility (no evaluation of the input values when swiveling in pole position)	
Bit 5	Tool alignment absolute or via frame calculation	
= 0	Tool alignment via frame calculation (TCOFRY for G18)	
= 1	Tool alignment absolute (TCOABS) For B axis kinematics of a turning machine, the tool spindle can be set in the basic position of the kinematics, either parallel to Z or to X. Therefore, the function "Align milling tool" is required for rotary milling machine bit 5 = 1.	
Bit 6	Do not offer swivel mode "directly" under swivel in JOG	
Bit 7	Swivel plane: Direction selection in basic position of the kinematic	
= 0	With direction selection + or -, both calculated solutions for the rotary axes are approached	
= 1	With direction selection + or -, only one of the calculated solutions for the rotary axes is approached	
Bit 8	Traverse the work offset for the rotary axes of the swivel data set as offset	
= 0	Only take the work offset for the rotary axes of the swivel data set into account in the WCS (compatibility)	
= 1	Traverse the work offset for the rotary axes of the swivel data set as offset Bit 8 is set in conjunction with MD21186.	
Bit 9	Swivel data set TC permanently assigned to the tool	
= 0	Show number of the swivel data set TC	

9.7 Swiveling

= 1	Do not show number of the swivel data set TC The swivel data set is assigned to the tool with parameter SGUD_TC_GNO if technology MD52200 = 3 is set for cylindrical grinding. Example: The machine manufacturer can write the current tool _TC_GNO = \$P_TOOLNO into parameter _TC_GNO in the tool change program L6. As a result, the tool carrier will be updated whenever a tool is changed.
Bit 10	Swivel plane: Show input field "Positioning direction"
= 0	Input field "Positioning direction" is not displayed
= 1	Input field "Positioning direction" is displayed with swivel table and swivel head/table combinations The selected positioning direction is stored in GUD_TC_POS_DIR and can be evaluated in CUST_800: _TC_POS_DIR = 0: Automatic (shortest path) _TC_POS_DIR = -1: Direction - _TC_POS_DIR = +1: Direction + The function must be implemented by the manufacturer.
Bit 11	Positioning both solutions of AB kinematics
= 0	Positions only one solution of some swivel angles (compatibility)
= 1	Positions both solutions calculated by the NC Bit 11 should be set for kinematics = 1 where, in the kinematics basic setting, neither of the two rotary axes rotates about the tool axis (AB kinematics in tool axis in the Z direction)
Bit 12	Swivel plane, align tool: Reset behavior takes account of rotary axis positions
= 0	Reset condition refers to the final position of the rotary axes of the swivel data set (compatibility) Linear axis identifiers are written into parameters \$TC_CARR21/22. On reset (TCOABS), the tool carrier is initialized with the final positions of the rotary axes (\$TC_CARR13/14).
= 1	Reset condition refers to the current position of the rotary axes of the swivel data set The rotary axis names of the swivel data set are written into parameters \$TC_CARR21/22. On reset (TCOABS), the tool carrier is initialized with the actual values of the rotary axes.
Bit 13	Activates the extended evaluation of rotations for axis-by-axis swiveling in the pole position
= 0	The rotations when swiveling in the pole position (compatibility) are not evaluated, the settings of bits 4 and 11 apply
= 1	Extended evaluation of rotations when swiveling in the pole position

Additional settings

For the swivel function, set the following machine data as a minimum to the following - although this deviates from the default value:

MD10602 \$MN_FRAME_GEOAX_CHANGE_MODE	Frames when changing geometry axes
= 1	The current total frame (work offsets) is recalculated when switching geometry axes. Translations, scaling and mirroring of the new geometry axes become effective. The rotations of the old geometry axes are retained.

MD11450 \$MN_SEARCH_RUN_MODE		Parameterization block search
Bit 1 = 1	Activate PROG_EVENT.SPF after block search. This means that for a block search, the rotary axes of the active swivel data set are pre-positioned.	

MD20110 \$MC_RESET_MODE_MASK		Control basic position after RESET and end of part program
Bit 14 = 1	System behavior "Base frame"	

MD20112 \$MC_START_MODE_MASK		Control basic position after part program start
400H		

MD20126 \$MC_TOOL_CARRIER_RESET_VALUE		Active tool carrier at RESET.
= 0	No active tool carrier at RESET.	
= > 0 (n)	Tool carrier with number n active at RESET. MD20126 is written to in CYCLE800. CYCLE800() corresponds to deselect tool carrier (MD20126 = 0).	

MD20150 \$MC_GCODE_RESET_VALUES[]		Delete position, G group.
[41] = 1	Delete position, G group 42 on TCOABS	
[51] = 2	Delete position, G group 52 on PAROTOF	
[52] = 1	Delete position, G group 53 on TOROTOF	
[52] = > 1	Delete position, G group 53 on TOROT, TOROTY or TOROTX Used for machine kinematics, types "T" and "M". See parameter \$TC_CARR34	

Note

If, after a RESET from the NC, a frame must be calculated in the tool direction, then MD20150 \$MC_GCODE_RESET_VALUES[52] can be set to a value > 1.

Application: Machine kinematics with Hirth gearing

Note on kinematics (swivel head / mixed kinematics) with Hirth tooth system:

Depending on the active plane (G17, G18, G19), the TOROT command (or TOROTX, TOROTY) is programmed in the NCU (G group 53) to calculate the compensating frame for the Hirth gearing system in CYCLE800. If the Hirth gearing causes the programmed rotation to deviate from the possible positions of the rotary axes, a \$P_TOOLFRAME compensating frame is created for swivel head and mixed kinematics (see HMI active ZO/details tool reference).

9.7 Swiveling

If the compensating frame must be retained after RESET or end of part program, enter the following value in the channel-specific machine data:

MD20150 \$MC_GCODE_RESET_VALUES[52]	Reset behavior of the G groups
= 2	For G17 (TOROT)
= 3	
= 4	For G18 (TOROTY)
= 5	For G19 (TOROTX)

Only applies for milling technology:

MD20196 \$MC_TOCARR_ROTAX_MODE	Rotary axis mode for oriented tool carriers
Bit 0 = 1	Swivel data set with a rotary axis used for a rotary table with C axis
Bit 1 = 1	Swivel data set with two rotary axes used for standard swiveling
Bit 2	
= 1	In a singular position, for which there is an unambiguous solution due to the alignment of the coordinate axes that are vertical to the orientation, the value 1 is displayed in the system variable \$P_TCSOL as the number of solutions.
= 0	This version is required to ensure compatibility with older software versions. Two solutions are displayed; the angles about the orientation axis differ by 180 degrees.

MD20360 \$MC_TOOL_PARAMETER_DEF_MASK	Tool parameter setting
Bit 10 = 1	The tool portion of an active orientable tool carrier is retained at T0 or D0 (no tool). Used for machine kinematics, types "T" and "M". See parameter \$TC_CARR34
Bit 19 = 1	The change in orientation of a tool caused by an orientable tool carrier is maintained at T0 or D0 (no tool). Bit 19 acts only with bit 10 = 1.

Only applies for milling technology:

MD21186 \$MC_TOCARR_ROT_OFFSET_FROM_FR	Offset of the rotary axes for an oriented tool carrier from the work offset of the rotary axis.
= 0	In CYCLE800, the WCS is recalculated when there is a value in the work offset (WO) of the rotary axes. If values are entered in the work offset of the rotary axes of the active swivel data set, the basic position of the swivel data set and therefore also the basic position of the WCS can change.
= 1	A value in the ZO of the rotary axes acts as offset of the orientable tool carrier. The Work remains unchanged.

MD21186 may not be rewritten in a program with call CYCLE800.

If several swivel data sets per channel are agreed and machine functions are to be started when changing the swivel heads or swivel tables, an M-command can be activated in the PLC program when changing the swivel data set.

MD22530 \$MC_TOCARR_CHANGE_M_CODE	M code for swivel data set change
= 0	No swivel data set change
< 0	M code + number of the swivel data set for the swivel data set change

Example

The PLC can limit or invert the spindle speed or clamp or release the rotary axes, for example, by outputting the M-commands.

Number of swivel data sets in channel 1	= 2
MD22530 \$MC_TOCARR_CHANGE_M_CODE	= -800
Programming swivel data set 1 (TCARR=1)	= M801
Programming swivel data set 2 (TCARR=2)	= M802

Setting the workpiece, tool and rotary table reference

Using the following machine data, you can set the workpiece, tool and rotary table reference system frames or you can influence the behavior of the system frames.

Application: System frames can be active after Reset or Power On, in order, e.g. to retract a drill from a swiveled position without causing a collision.

MD24006 \$MC_CHSFRAME_RESET_MASK	Active system frames after RESET
Bit 4	System frame workpiece reference
= 0	Not active
= 1	Remains active

MD24007 \$MC_CHSFRAME_RESET_CLEAR_MASK	Clear system frames after RESET
Bit 4	System frame workpiece reference
= 0	Do not delete
= 1	Delete

When used for measuring or swiveling in JOG, the workpiece reference must be active at RESET and not cleared (cascaded measuring).

MD24006 \$MC_CHSFRAME_RESET_MASK	Active system frames after RESET
Bit 4 = 1	System frame for workpiece reference remains active after RESET

MD24007 \$MC_CHSFRAME_RESET_CLEAR_MASK	Clear system frames after RESET
Bit 2 = 0	Reserved, do not delete TCARR and PAROT
Bit 3 = 0	Reserved, do not delete TOROT and TOFRAME
Bit 4 = 0	Do not clear system frame workpiece reference after RESET

MD24008 \$MC_CHSFRAME_POWERON_MASK		Reset system frames after power on.
Bit 2	System frame rotary table reference (PAROT)	
= 0	Do not reset	
= 1	Reset	
Bit 3	System frame tool reference (TOROT,...)	
= 0	Do not reset	
= 1	Reset	
Bit 4	System frame workpiece reference	
= 0	Do not reset	
= 1	Reset	

MD24080 \$MC_USER_FRAME_POWERON_MASK		Settings for settable frames.
Bit 0		
= 0	Settable work offset via power on not active.	
= 1	Last active settable work offset remains active after power on if MD20152 \$MC_GCODE_RESET_MODE[7] = 1.	

Application: Work offset G5xx, including all rotations, should remain active after Power On.

MD28082 \$MC_MM_SYSTEM_FRAME_MASK		Setting-up system frames (SRAM)
	Default, cannot be changed	

MD28083 \$MC_MM_SYSTEM_DATAFRAME_MASK		Setting-up data management, system frames (SRAM)
	Default, cannot be changed	

MD28085 \$MC_MM_LINK_TOA_UNIT		Assignment of a TO unit to a channel (SRAM)
If several channels are set-up, then the following can be set in the machine data: The number of tool carriers (see MD18088) on the TO unit per channel. - OR - Assigning all of the set tool carriers to a channel.		

Axial machine data for the modulo rotary axes of the swivel data set

MD30455 \$MA_MISC_FUNCTION_MASK		Axis functions
Bit 0	Modulo rotary axis programming	
= 0	Programmed positions must be in the modulo range, otherwise an alarm is triggered (e.g. 0 to 359.999 degrees).	
= 1	When positions are programmed outside of the modulo range, no alarms are signaled. The position is modulo converted internally (e.g. -180 to +180 degrees).	
Bit 2	Positioning, rotary axis	

= 0	As programmed
= 1	Along the shortest path Application: With the setting bit 2=1, the rotary axis C travels along the shortest path for G90 with DC. For further information, refer to chapter "Manufacturer cycle CUST_800.SPF (Page 216)".

MD32010 \$MA_JOG_VELO_RAPID[AX] AX = axis name	Rapid traverse in JOG Rotary and machine axes that should travel for swiveling in JOG.
= 10000	Rapid traverse in JOG mode for swiveling in JOG

SD42980 \$SC_TOFRAME_MODE	Setting, frame definition for TOROT, PAROT
= 2000	Swivel (default value)

SD42974 \$SC_TOCARR_FINE_CORRECTION	Fine offset TCARR (swivel data set)
= 0	No fine offset of the swivel data set vectors.
= 1	Fine offset of the swivel data set vectors. The parameters of the swivel data set from \$TC_CARR41[n] n...number of the swivel data set and higher apply.

Swiveling in the JOG mode

Cycle alarms 62186 and 62187 can be hidden or displayed using the following cycle machine data:

MD55410 \$MC_MILL_SWIVEL_ALARM_MASK	Activate fault evaluation CYCLE800
Bit 0	Activates fault 61186
= 0	Hide fault 61186 "Active work offset G%4 and base (base reference) contains rotations" (default setting).
= 1	Display fault 61186
Bit 1	Activate fault 61187
= 0	Hide fault 61187 "Active base and base reference (G500) contain rotations" (default setting).
= 1	Display fault 61187

9.7.2 CYCLE800 checklist for the identification of the machine kinematics

Note

Identification of the machine kinematics according to DIN 66217 or ISO 841-2001

This checklist does not claim to be complete.

- **Do the 3 linear axes of the machine that are active for the transformation form an orthogonal coordinate system?** Geometry axes XYZ
- **How many swivel kinematics does the machine have?**
Combinations of two (or one) rotary axis and the three linear axes are always formed.
- **Which machine components are concerned?**
Swivel head, swivel table or swivel head and rotary table.
- **What are the names of the rotary axes of the kinematics?**
Manual rotary axes are permitted and do not have to be declared in the NC.
- **What is the 1st or 2nd d rotary axis of a swivel data set?**
- **Is the traversing direction of the linear axes and the rotary axes correct?** Right-hand rule
Rule: If the linear axis or the rotary axis moves the workpiece, the direction of motion of the axis and also the sign of the rotary axis vector change.
- **What is the initial setting of the kinematics?**
This defines the tool orientation and the plane G17, G18, G19.
- **Which rotary axis rotates around which axis of the coordinate system or the machine axis (axes)?**
This defines the rotary axis vectors of the kinematics.
Example 1:
Head kinematics. Rotary axis 2 rotates around axis Y → rotary axis vector $V2_{xyz} = 0, 1, 0$
Example 2:
Table kinematics. Rotary axis 1 rotates around axis X → rotary axis vector $V1_{xyz} = -1, 0, 0$

9.7.3 Setting up a tool carrier/swivel data set

Defining a tool carrier/swivel data set

You must create a tool carrier (tool holder) for every swivel head, swivel table or each swivel head/swivel table combination.

Tool carriers can be declared in several channels. A tool carrier can be defined as classic or via the kinematic chain.

A classic tool carrier comprises the parameters $\$TC_CARR1[n]$ to $\$TC_CARR65[n]$
n = number of the tool carrier.

The geometry of the tool carrier based on the kinematic chain comes from the chain. There are also other descriptive parameters $\$TC_CARR_KIN...[n]$ and $\$TC_CARRxx[n]$.

If all preconditions from Chapter "Technology cycles for swiveling (Page 190)" are met, you can assign the tool carrier parameters via the input screen form in the "Setup" > "NC" > "Transformation" operating area.

In the case of a milling technology, when creating a new tool carrier you can choose whether it should be parameterized conventionally or via the chain.

Programming with appropriate value assignment is also possible in an NC program. The parameters of the tool carrier are immediately effective after the program has started.

The following softkeys are assigned to the Swivel function in the "Program" > "Various" operating area:

Milling	Turning	
"Swivel plane"	"Swivel plane"	
"Advance milling tool"	"Swivel tool"	→ "Align turning tool"
		→ "Align milling tool"
		→ "Advance milling tool"

The "Align turning tool" and "Align milling tool" softkeys are only displayed if the "B-axis kinematics" function has been activated via \$TC_CARR37[n].

More information

More information on setting up the kinematic chain can be found in the Function Manual Basic Functions.

Setting the rotary axis direction using ideal vectors

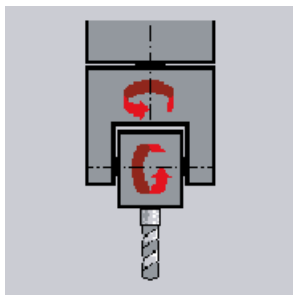
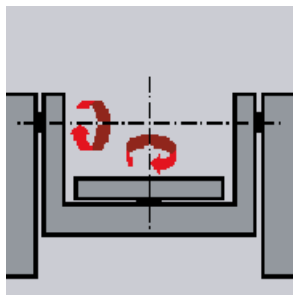
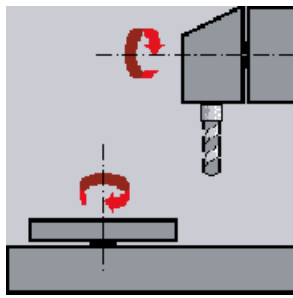
Note

Note that ideal vectors must be used for the direction of the rotary axes.

For example, if the rotary axis rotates around X, then the orientation (1,0,0) must be specified. Otherwise, when swiveling to 0°, then it is possible that the rotary axis is not positioned to this value.

It is recommended that the deviation is corrected using compensation if the rotary axis direction mechanically deviates from the ideal vector.

Kinematic types \$TC_CARR23[n]

Swivel head (type T)	Swivel table (type P)	Swivel head + Swivel table (type M)
		
Offset vector I1	Offset vector I2	Offset vector I1
Rotary axis vector V1	Rotary axis vector V1	Rotary axis vector V1
Offset vector I2	Offset vector I3	Offset vector I2
Rotary axis vector V2	Rotary axis vector V2	Offset vector I3

Swivel head (type T)	Swivel table (type P)	Swivel head + Swivel table (type M)
Offset vector I3	Offset vector I4	Rotary axis vector V2
		Offset vector I4

Name of swivel data set

If several swivel data sets are declared in each NC channel, then a name is assigned to each swivel data set: \$TC_CARR34[n]

The name of the swivel data set may only contain characters that are permissible for NC programming: A...Z, 0...9 and _ !

If multiple NC channels have been set up, note the settings in MD28085. If a TO unit is assigned to several NC channels, the channel display in the dialog corresponds to the TO unit.

You can find **more information** in Chapter "Technology cycles for swiveling (Page 190)".

System variable \$TC_CARR37[n]

In order that a value can be displayed in the input/selection boxes of the dialog for swiveling, the following display versions can be set:

Meaning of the decimal positions

\$TC_CARR37[n]	Display variants of the input masks for CYCLE800	
Decimal place	Meaning	
ONES	Select swivel mode	
	0 =	Axis-by-axis
	1 =	Axis-by-axis + projection angle
	2 =	Axis-by-axis + projection angle + solid angle
	3 =	Axis-by-axis + direct
	4 =	Axis-by-axis + projection angle + direct
	5 =	Axis-by-axis + projection angle + solid angle + direct
TENS	Rotary axis 1	
	0 =	Automatic
	1 =	Manual
	2 =	Semiautomatic
HUNDREDS	Rotary axis 2	
	0 =	Automatic
	1 =	Manual
	2 =	Semiautomatic
THOUSANDS	Selection field direction: Direction selection of the rotary axes	
	0 =	No display of the direction reference for kinematics that only have one solution. Direction selection (_DIR) Minus is generated in the cycle call CYCLE800.

\$TC_CARR37[n]	Display variants of the input masks for CYCLE800	
Decimal place	Meaning	
	3 =	Direction reference, rotary axis 1, direction selection Minus in the basic setting of the kinematics.
	4 =	Direction reference, rotary axis 2, direction selection Minus in the basic setting of the kinematics.
	5 =	No display of the direction reference for kinematics that only have one solution. Direction selection (_DIR) Plus is generated in the cycle call CYCLE800.
	8 =	Direction reference, rotary axis 1, direction selection Plus in the basic setting of the kinematics.
	9 =	Direction reference, rotary axis 2, direction selection Plus in the basic setting of the kinematics.
		The values 1, 2, 6 and 7 are not permitted.
TEN THOUSANDS	Selection field, tracking of the tool tip or B axis kinematics	
	0 =	No display of the tracking of the tool tip input field.
	1 =	Tracking of tool tip by means of TRAORI.
	2 =	No tracking of tool tip + B axis kinematics turning technology.
HUNDRED THOUSAND	Basic position of the kinematics ¹⁾	
	0 =	Corresponding to the working plane G17, G18 or G19 (compatibility)
	1 =	Tool axis in the Z direction
	3 =	Tool axis in the X direction
ONE MILLION TEN MILLION	Selection field, retraction	
	00 =	No retraction
	01 =	Retraction Z
	02 =	Retraction Z, XY
	03 =	Retraction Z or Z, XY
	04 =	Maximum retraction in tool direction
	...	
	08 =	Incremental retraction in tool direction
	...	
	15 =	Retraction Z or Z, XY or in maximum tool direction or in incremental tool direction
	\$TC_CARR38[n]	Retraction position X
	\$TC_CARR39[n]	Retraction position Y
	\$TC_CARR40[n]	Retraction position Z
HUNDRED MILLION	Swivel data set enabled Setting swivel data set change Automatic or manual tool change is only necessary under ShopMill or ShopTurn (→ see also: CUST_800.spf, marks _M2 to _M13). A swivel data set must be "enabled" in every case (value ≥ 4).	
	0 =	Tool carrier not enabled
	4 =	Tool carrier enabled Automatic swivel data set and tool change

\$TC_CARR37[n]	Display variants of the input masks for CYCLE800	
Decimal place	Meaning	
	5 =	Tool carrier enabled Automatic swivel data set change and manual tool change
	6 =	Tool carrier enabled Manual swivel data set change and automatic tool change
	7 =	Tool carrier enabled Manual swivel data set and tool change

Notes on the setting Basic position of the kinematics ¹⁾

For turning machines (1st technology is turning) with a B axis, you can set whether the tool is oriented toward "-Z" or toward "-X" in the basic position of the kinematics. This setting is taken into account for "Align tool" and for "Swivel plane".

Align tool

- For "Align tool", the programming of beta and gamma is machine-independent.
- With the basic position setting "Working plane" or "-Z", beta = 0 also corresponds to this basic position (compatibility).
- With the basic position setting "-X", the B axis is re-oriented through 90° for beta = 0.

Swivel plane (new swivel plane)

- With the basic position setting "Working plane" or "-Z" and active working plane G17, there is no new tool orientation for swivel plane to basic position (compatibility).
- With the basic position setting "-X" and active working plane G17, the tool orientation is rotated 90° around Y for swivel plane to basic position.

Therefore, with the basic position setting "-X", milling is possible in a swiveled plane in different working planes (G17 or G19).

The following settings and programming is not permitted and results in error messages:

- Swivel plane, basic position -Z and G18 or G19 active
- Swivel plane, basic position -X and G18 active

Retracting the geometry axes before swiveling

Properties (input screen)

Select retract

- Incremental in tool direction: Yes
- Maximal in tool direction: Yes
- Machine axis Z: Yes
- Machine axis Z and then XY: No
- Retract position Z: 300.000

Select swivel mode

- Axis by axis: Yes
- Projection angle: No
- Solid angle: No
- Rotary axes direct: Yes

Select prefer. direction

Reference axis pref. direction: Yes default=+

Select tracking

Rotation axis 1: No

Buttons: NC memory, PROFIBUS, Machine model, Transformations, Cancel, OK

Figure 9-6 Dialog to select the type of retraction

Select the retraction variant in the input field: This is defined in the ONE MILLION and TEN MILLION position of the system variable \$TC_CARR37[n]:

- Retract axis Z
- Retract axes Z, XY
- Retract in the tool direction, maximum or incremental

NOTICE

Collision avoidance

Make sure that the tool and the workpiece cannot collide during swiveling when the tool axes move.

Retraction of the axis Z or retraction of the axes Z, XY is implemented as an absolute machine position to the values of parameters \$TC_CARR38[n] to \$TC_CARR40[n].

\$TC_CARR38[n]	Retraction position X
\$TC_CARR39[n]	Retraction position Y
\$TC_CARR40[n]	Retraction position Z

The type of retraction is modified in the Manufacturer cycle CUST_800.SPF (Page 216).

Note

Retraction in the tool direction is useful for swivel data sets in which the tool is re-oriented (swivel head or mixed kinematics). This applies particularly for turning machines with a B axis. During retraction in "maximum tool direction", traversing is performed until one of the relevant linear axes reaches the software limit switch.

9.7.4 Setting up the tool carrier based on kinematic chain

You must create a tool carrier for every swivel head, swivel table or each swivel head/swivel table combination.

Precondition

A kinematic chain is already available or must be created for the machine.

More information

More information is provided in the
Function Manual Tools.

Creating the tool carrier

Create the tool carrier using the input screen form in the operating area
"Setup" > "NC" > "Transformation".

In the "New tool holder" dialog you can select the tool carrier on the basis of the chain.

More information on selection is provided in Chapter: "Setting up a tool carrier/swivel data set (Page 198)".

TOOLCARR - tool carrier (chain)

Name: **TABLE_CC**

Enable: **Yes**

Type: **Swivel plane**

Kinematic

Workpiece chain

Start of workpiece chain: **OFFSET_TABLE**

Correct. element: **B-OFFSET-CORR**

Rotation axis: **B-AXIS**

Angle area: **-90.000** **90.000** °

Hirth teeth: **No**

Correct. element: **C-OFFSET-CORR**

Rotation axis: **C-AXIS**

Angle area: **0.000** **360.000** °

Hirth teeth: **No**

End of workp. chain: **CLAMPING**

Close workpiece chain: **No**

Tool direction in initial setting: **-2**

Properties (input screen)

Select retract: **Yes**

Incremental in tool direction: **Yes**

Buttons: **NC memory**, **PROFIBUS**, **Machine model**, **Transformations**, **Cancel**, **OK**

Figure 9-7 Tool carrier dialog based on the chain

The following tool data is set using the input screen form for tool carrier setup:

Variable name	Meaning
\$TC_CARR_KIN_CNTRL[n]	Controls the acceptance of tool carrier geometry data from kinematic chain elements Bit 0: Tool carrier using kinematic chain Bit 1: Close part chain (calculation I4) Bit 2: Close tool chain (calculation I1)
\$TC_CARR_KIN_TOOL_START[n]	Name of the kinematic chain element that forms the starting point of the tool chain
\$TC_CARR_KIN_TOOL_END[n]	Name of the kinematic chain element that forms the end point of the tool chain
\$TC_CARR_KIN_PART_START[n]	Name of the kinematic chain element that forms the starting point of the tool chain
\$TC_CARR_KIN_PART_END[n]	Name of the kinematic chain element that forms the end point of the part chain
\$TC_CARR_CORRELEM[n, m] n = number of the tool carrier data set 0 ≤ m ≤ 3	Name of the kinematic chain element to which correction can be performed with CYCLE9960
\$TC_CARR_KIN_ROTAX_NAME[n, m]	String name of the kinematic chain element that contains the rotary axis
\$TC_CARR23[n]	Kinematic head (T), Table (P), Mixed (M)
\$TC_CARR26[n] \$TC_CARR27[n]	Offset Hirth teeth of the rotary axes

Variable name	Meaning
\$TC_CARR28[n] \$TC_CARR29[n]	Increment Hirth teeth of the rotary axes
\$TC_CARR30[n] \$TC_CARR31[n]	Software limit for minimum position of the rotary axes
\$TC_CARR32[n] \$TC_CARR33[n]	Software limits for maximum position of the rotary axes
\$TC_CARR34[n]	Name of the tool carrier
\$TC_CARR37[n]	
\$TC_CARR38[n] \$TC_CARR39[n] \$TC_CARR40[n]	Position for retraction

Rotary axes (axes of rotation)

The input screen form of the machine image is opened via the "Select element" softkey, and you can select the chain element that corresponds to the rotary axis.

The name for the chain element with rotary axis included is contained in the variable \$TC_CARR_KIN_ROTAX_NAME[n, m].

Correction element

The variable \$TC_CARR_CORRELEM[n, m] makes reference to the chain element that contains the kinematic correction if kinematics are measured and corrected via CYCLE9960.

9.7.5 Examples of machine kinematics for commissioning the tool carrier

The following examples are for a classic tool carrier. If the tool carrier is based on a kinematic chain, the kinematic chain must be set up accordingly:

Example 1: Swivel head 2 "HEAD_2" tool carrier based on kinematic chain

In order to map the kinematics in the machine, we recommend mapping the kinematics as they actually are with regard to the kinematic chain. The drawing on the left shows how the vectors are specified to map the kinematics in such a way that reflects how the machine actually is. The tool carrier starts at the pivot point of the B axis followed by the C_OFFSET up to the actual pivot point of the C axis with an orientation of (0,1,1). The offset then connects to the reference point of the tool. However, the tool carrier can also start at the pivot point of the C axis (as is usual with classic). If the swivel head is firmly attached to the machine and the kinematics are closed, the result of the description is the same. If there are multiple swivel heads on the machine and the kinematics are open, the first kinematics description is recommended.

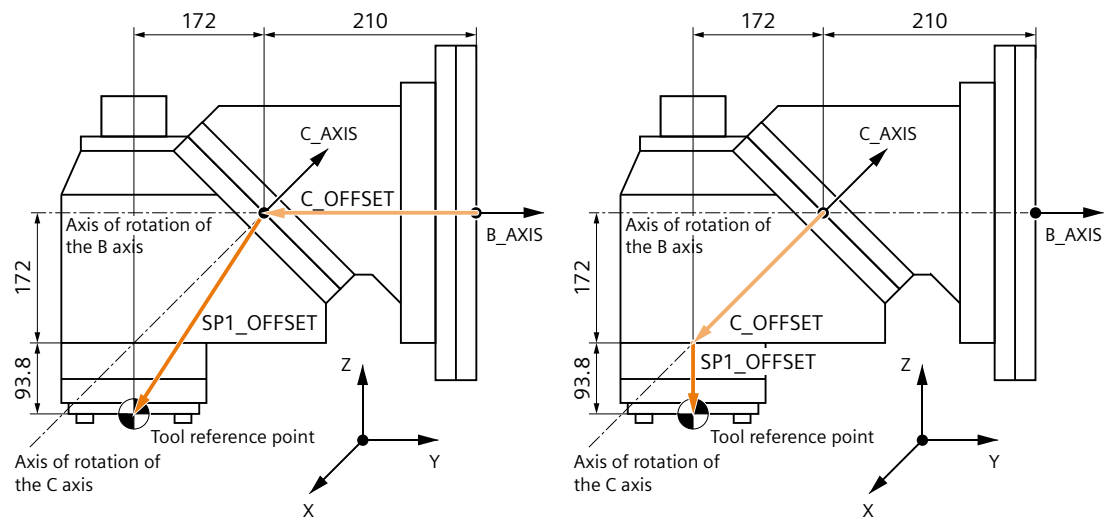
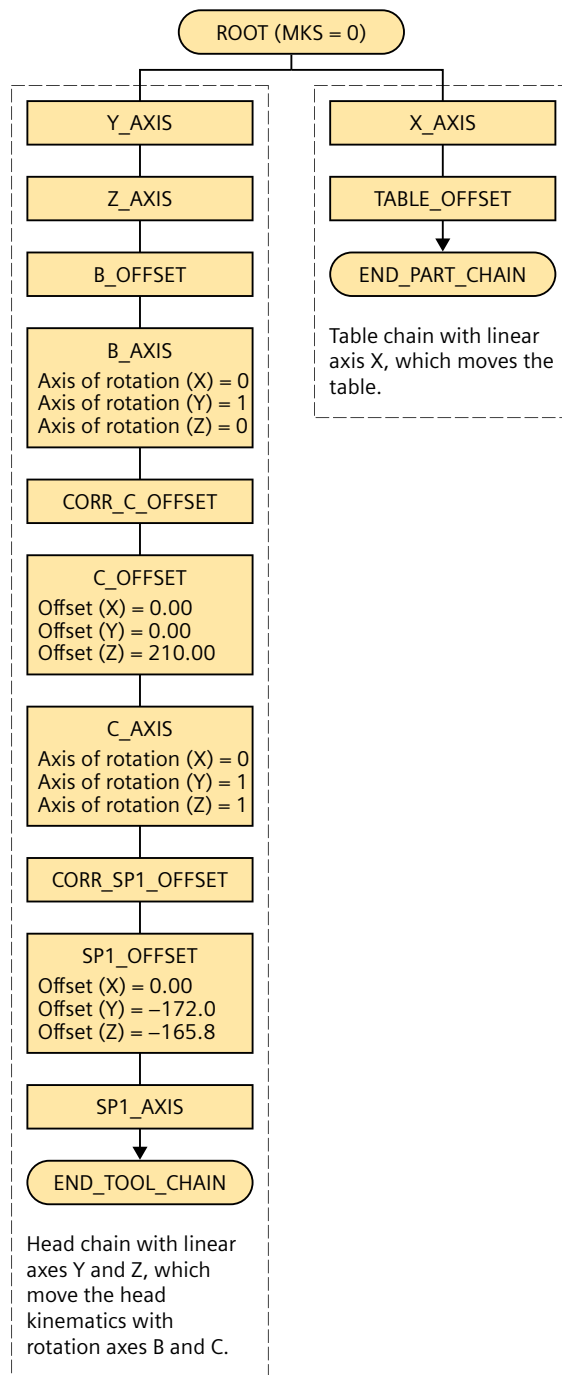


Figure 9-8 Swivel head with actual and offset kinematics description

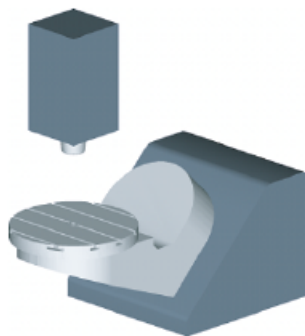
Schematic representation of the chain:



Name	HEAD_2		
Enable	Yes		
Type	Swivel plane		
Kinematics			
Tool chain			
Start of tool chain	Z_AXIS		
Axis of rotation	B_AXIS		

Angle range		0.000°	360.000°
Hirth teeth	No		
Correction element	CORR_C_OFFSET		
Axis of rotation	C_AXIS		
Angle range		-15.000°	100.000°
Hirth teeth	No		
Correction element	CORR_SP1_OFFSET		
End of tool chain	SP1_OFFSET		
Close tool chain	Yes		
Properties			
Select retract			
• Machine axis Z			Yes
• Retract position			200
Select swivel mode			
• Axis-by-axis			Yes
Reference axis pref. direction			Axis of rotation 2
Select tracking			No

Example 2: Cardanic table "TABLE_45" TCARR based on kinematic chain



The vectors in the drawing refer to the initial setting of the kinematics. The A_OFFSET vector can be offset anywhere on the axis of rotation of the A axis.

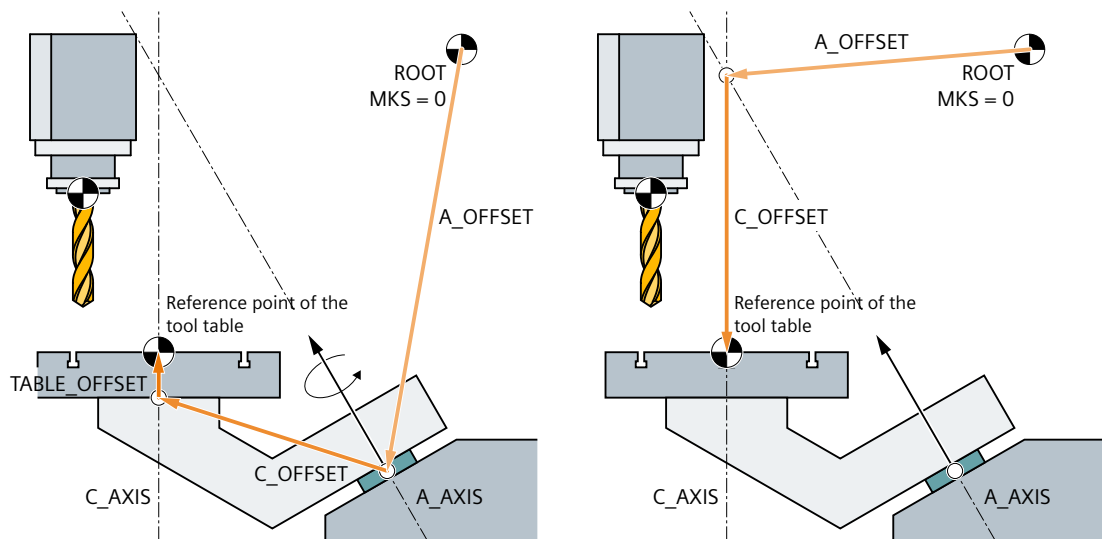
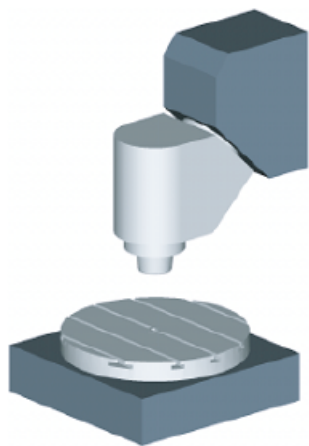


Figure 9-9 Vector offset A_OFFSET

Name	TABLE_45		
Enable	Yes		
Type	Swivel plane		
Kinematics			
Tool chain			
Start of tool chain	Z_AXIS		
	CORR_A_OFFSET		
Axis of rotation	A_AXIS		
Angle range		0.000°	360.000°
Hirth teeth	No		
Correction element	CORR_C_OFFSET		
Axis of rotation	C_AXIS		
Angle range		0.000°	360.000°
Hirth teeth	No		
End of tool chain	OFFSET_TABLE		
Close tool chain	Yes		
Properties			
Select retract			
• Machine axis Z			Yes
• Retract position			200
Select swivel mode			
• Axis-by-axis			Yes
Reference axis pref. direction			Axis of rotation 1
Select tracking			No

Example 3: Swivel head/rotary table "MIXED_45" based on kinematic chain



The vectors in the drawing refer to the initial setting of the kinematics. The vector SP1_OFFSET of the B axis to the reference point of the tool can be offset anywhere on the axis of rotation of the B axis.

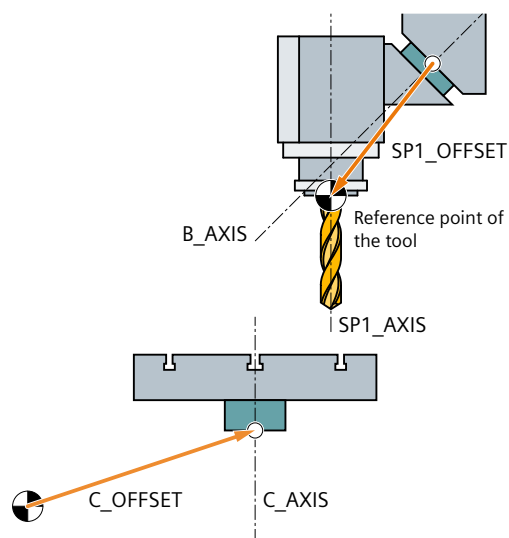


Figure 9-10 Vector offset SP1_OFFSET

Name	MIXED_45		
Enable	Yes		
Type	Swivel plane		
Kinematics			
Tool chain			
Start of tool chain	Z_AXIS		
Axis of rotation	B_AXIS		
Angle range		0.000°	180.000°
Hirth teeth	No		
Correction element	CORR_SP1_OFFSET		
End of tool chain	SP1_AXIS		

Close tool chain	Yes		
Workpiece chain			
Start of workpiece chain			
Correction element	CORR_C1_OFFSET		
Axis of rotation	C_AXIS		
Angle range		0.000°	360.000°
Hirth teeth	No		
End of workpiece chain	C_AXIS		
Close tool chain	Yes		
Properties			
Select retract			
• Machine axis Z			No
Select swivel mode			
• Axis-by-axis			Yes
Select prefer. direction			Yes, default=+
Reference axis pref. direction			Axis of rotation 1
Select tracking			No

Example 4: Setting up a turning machine with B axis (chain)

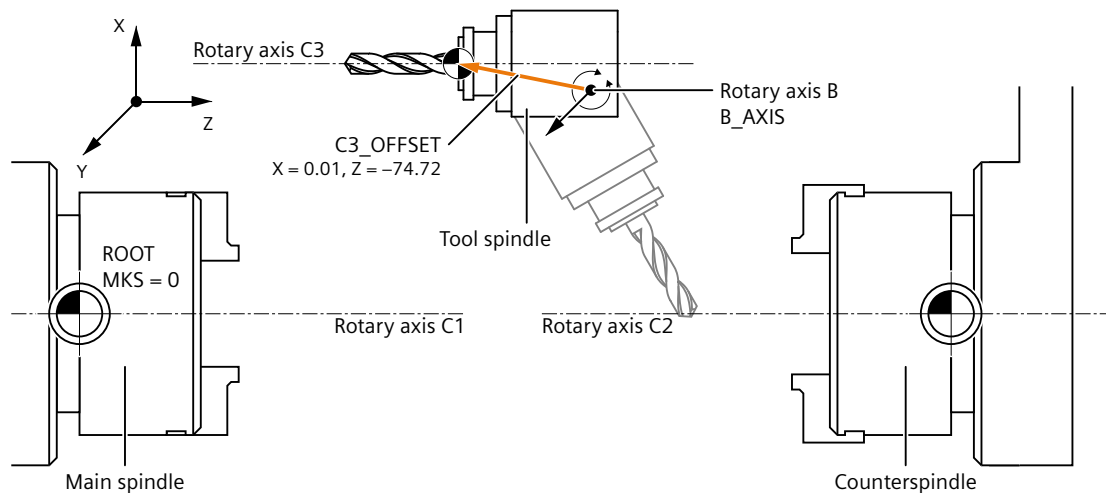


Figure 9-11 Turning machine with B axis

The example below relates to a turning machine with main spindle and counterspindle as well as B axis in the head.

The first step is to create the kinematic chain for this machine.

Schematic representation of the kinematic chain:

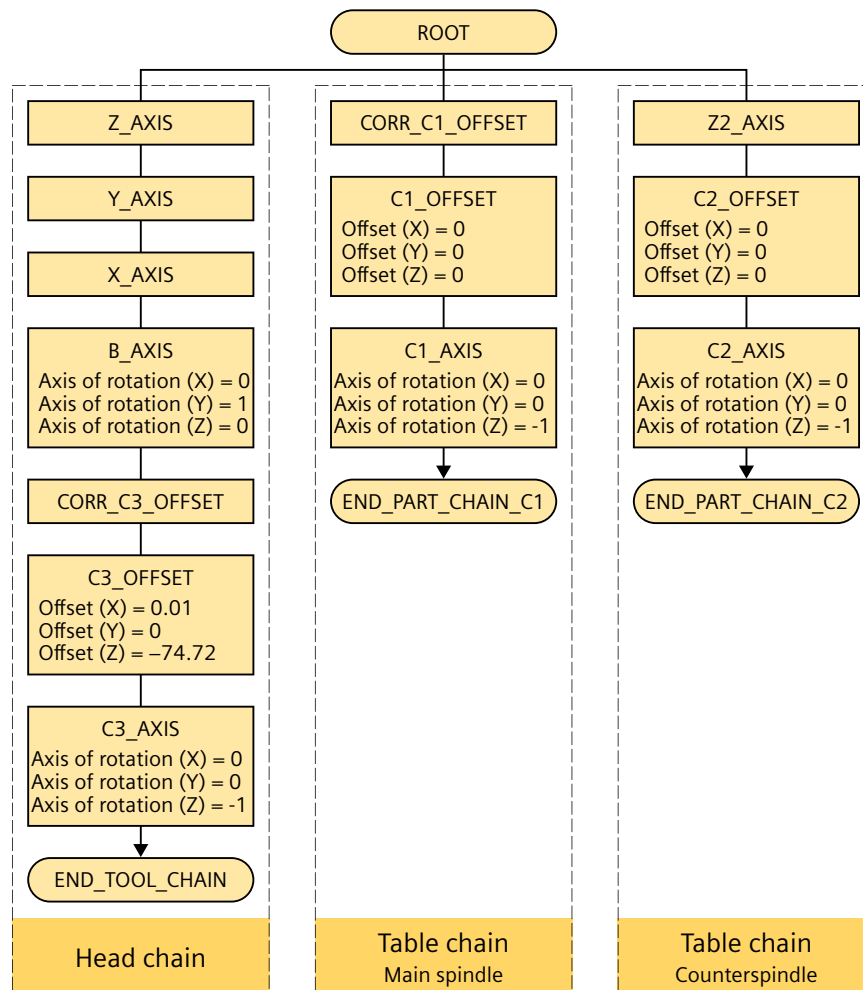


Figure 9-12 Schematic representation of the chain

Once the machine's kinematic chain has been created, various tool carriers can be created on this basis.

For turning machines with an additional rotary axis B, the tool can be aligned or swiveled on the X/Z plane. If the machine has a counterspindle, you can, for example, work with a tool alternating between the main spindle and counterspindle.

You can create the following tool carriers to use the "Align tool" and "Swivel plane" functions:

Tool carrier 1

For aligning and swiveling tools on the main spindle and counterspindle (Turning, Face Y/C, Surface Y/C):

Name			
Enable	Yes		
Type	Align turning tool		
Kinematics			
Tool chain			
Start of tool chain	Z_AXIS		

9.7 Swiveling

Axis of rotation	B_AXIS		
Angle range		-15.000°	195.000°
Hirth teeth	No		
Correction element			
Axis of rotation	C3_AXIS		
Angle range		0.000°	360.000°
Hirth teeth	No		
Correction element			
End of tool chain	SP1_OFFSET		
Close tool chain	Yes		
Properties			
Select retract			
• Machine axis Z			No
Select swivel mode			
• Axis-by-axis			Yes
Select prefer. direction			Yes, default=+
Reference axis pref. direction			Axis of rotation 1
Select tracking			No

When commissioning the "Align turning tool", the TEN THOUSANDS position in parameter \$TC_CARR37[n] must be set to 2 or 3 in swivel data set 1. This is the reason why in the program editor "Swivel tool → Align turning tool or milling tool" is listed for tool carrier 1.

For more information, see Chapter "Setting up a tool carrier/swivel data set (Page 198)"

Tool carrier 2

For machining inclined surfaces when milling on the main spindle (face B):

Name	TC_MAIN_SPINDEL		
Enable	Yes		
Type	Swivel plane		
Kinematics			
Tool chain			
Start of tool chain	Z_AXIS		
Axis of rotation	B_AXIS		
Angle range		-15.000°	195.000°
Hirth teeth	No		
Correction element	CORR_C3_OFFSET		
End of tool chain	C3_AXIS		
Close tool chain	Yes		
Workpiece chain			
Start of workpiece chain	CORR_C1_OFFSET		
Correction element	CORR_C1_OFFSET		
Axis of rotation	C1_AXIS		
Angle range		0.000°	360.000°
Hirth teeth	No		

End of workpiece chain	C1_AXIS		
Close workpiece chain	Yes		
Properties			
Select retract			
• Machine axis Z			No
Select swivel mode			
• Axis-by-axis			Yes
Select prefer. direction			Yes, default=+
Reference axis pref. direction			Axis of rotation 1
Select tracking			No

Tool carrier 3

For machining inclined surfaces when milling on the counterspindle (face B):

Name	TC_SUB_SPINDEL		
Enable	Yes		
Type	Swivel plane		
Kinematics			
Tool chain			
Start of tool chain	Z_AXIS		
Axis of rotation	B_AXIS		
Angle range		-15.000°	195.000°
Hirth teeth	No		
Correction element	CORR_C3_OFFSET		
End of tool chain	C3_AXIS		
Close tool chain	Yes		
Workpiece chain			
Start of workpiece chain	CORR_C2_OFFSET		
Correction element	CORR_C2_OFFSET		
Axis of rotation	C2_AXIS		
Angle range		0.000°	360.000°
Hirth teeth	No		
End of workpiece chain	C2_AXIS		
Close workpiece chain	Yes		
Properties			
Select retract			
• Machine axis Z			No
Select swivel mode			
• Axis-by-axis			Yes
Select prefer. direction			Yes, default=+
Reference axis pref. direction			Axis of rotation 1
Select tracking			No

The kinematic chain is created once and all requested tool carriers refer to the corresponding chain elements. If a value in the chain described in kinematic terms has to be changed, this is immediately effective for all tool carriers.

Parameter CUTMOD in CYCLE800

Distinction between calls:

- The CUTMOD NC function is activated when data set 2 is called in the NC program (CYCLE800 align turning tool). This means that the cutting edge position or the tool reference point is updated corresponding to the tool orientation:
- When calling a swivel data set for rotational operation in the NC program (CYCLE800 align tool), the CUTMOD NC function is activated. The cutting edge position, tool angle, cut direction and tool reference point are thus calculated in accordance with the current tool orientation following the "Align tool" function.

More information

A more detailed description of parameter CUTMOD is provided in the Function Manual Tools.

9.7.6 Manufacturer cycle CUST_800.SPF

Function

During swiveling, all axis positions are approached using the CUST_800.SPF cycle. The call is exclusively made from the swivel cycle CYCLE800 or from the cycles E_TCARR (ShopMill) or F_TCARR (ShopTurn).

In cycle CUST_800.SPF, the function markers (_M2: to _M59) are prepared and documented. Also refer to the following "Structogram CYCLE800".

Note

If you modify the CUST_800, ensure that the G commands of the G groups used are restored at the end of the CUST_800 (versions S_G_1 to S_G_5!).

Parameter

CUST_800 (INT _MODE, INT _TC1, REAL _A1, REAL _A2, INT _TC2, REAL _T_POS)

_MODE	A jump is made to markers _M2 to _M59
_TC1	Number of the swivel head/table
_A1	Angle of rotary axis 1
_A2	Angle of rotary axis 2

_TC2	Feed evaluation in percent (%) for swiveling in JOG mode and number of the new swivel head/table when replaced in ShopMill
_T_POS	Incremental position during retraction in the incremental tool direction (see marker _M44, _M45)

Retract prior to swiveling

If the CUST_800.SPF cycle is not modified, the Z axis (marker _M41) or the Z axis followed by the X, Y axes (marker _M42) are first traversed in the Machine to the positions when retracting prior to swiveling. The freely available position values are specified in the system variables \$TC_CARR38[n] to \$TC_CARR40[n]. When retracting, the active tool cutting edge is deselected (DO) and is reselected after retraction.

If retraction in the tool direction had been declared, the tool axis is retracted to the software end position (maximum in tool direction) or by an incremental distance away from the tool in the tool direction. The tool lengths are taken into account accordingly.

Structure (coarse) of swivel cycles

Input screen form under ShopMill (ShopTurn)

Input screen form for CYCLE800 standard cycle

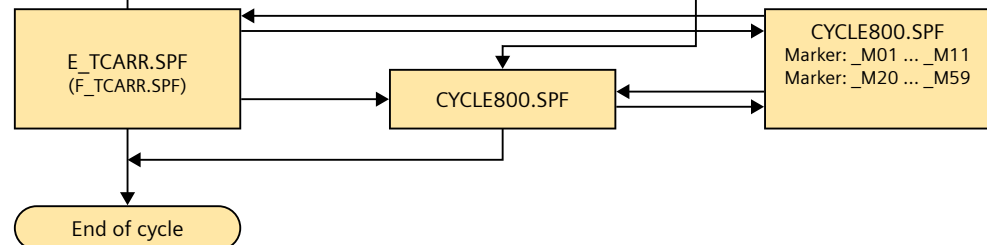


Figure 9-13 Structure (coarse) of swivel cycles

CYCLE800 structured chart

Sequence in AUTOMATIC mode

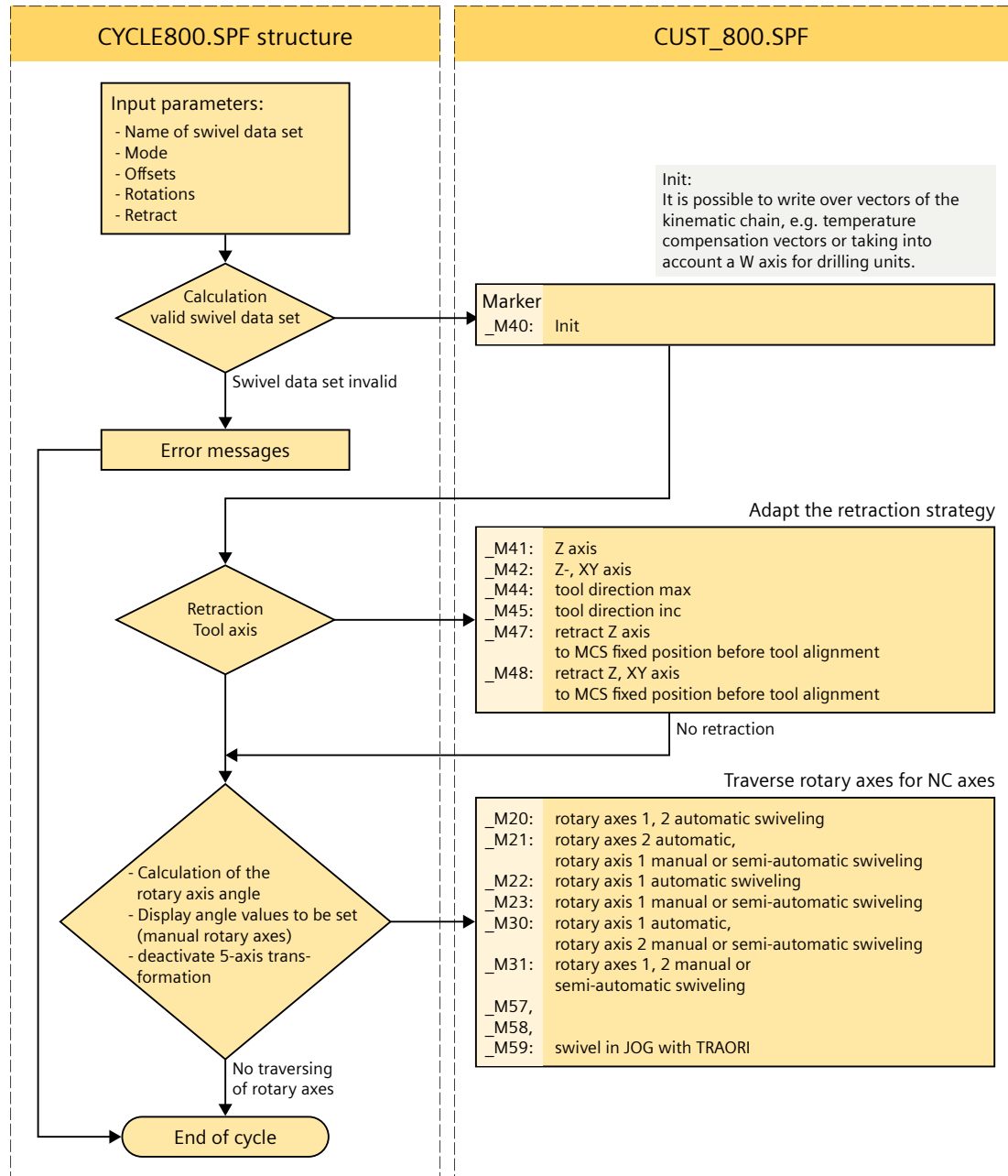


Figure 9-14 Structure: CYCLE800.SP / CUST_800.SP

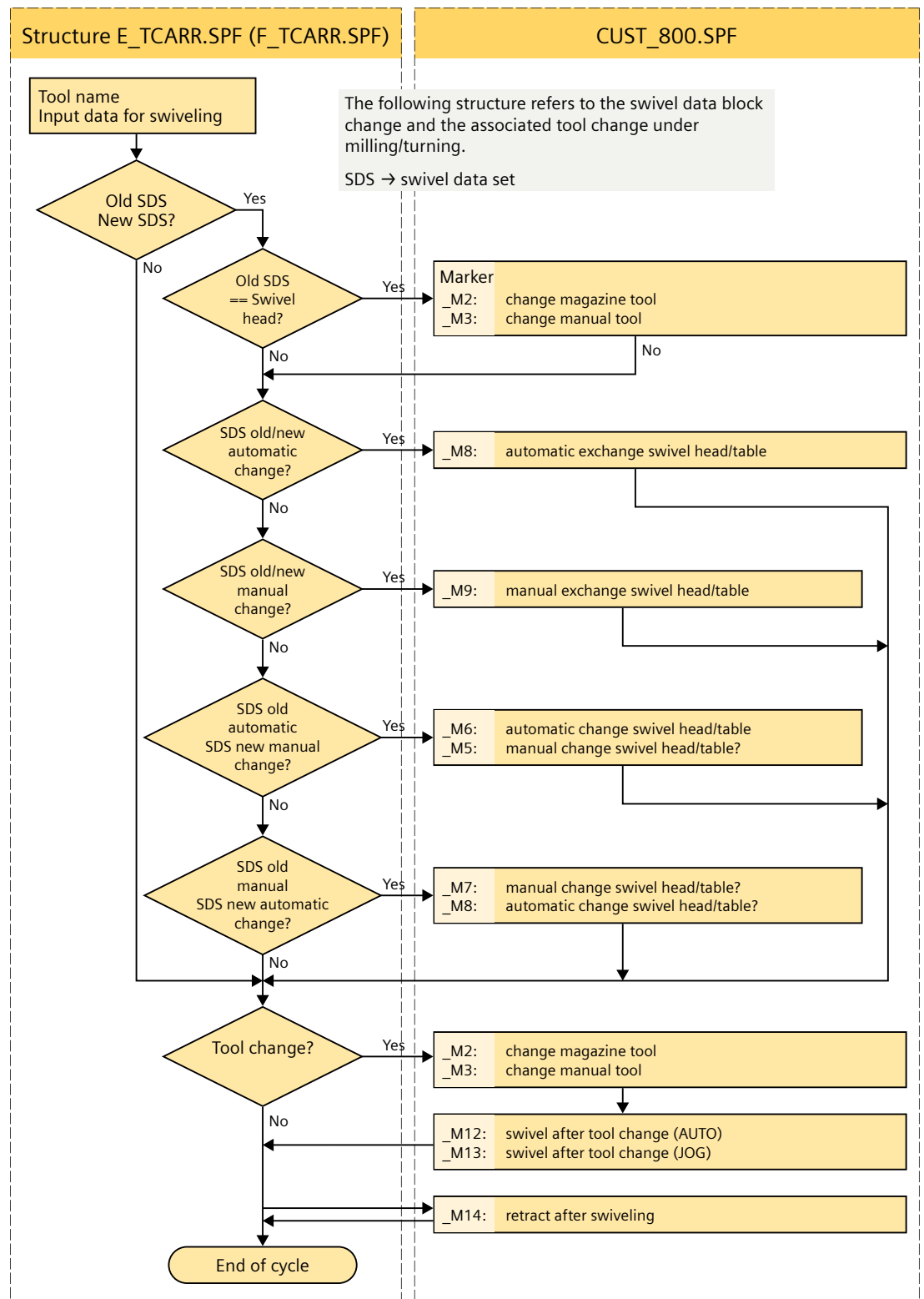


Figure 9-15 Structure: E_TCARR.SPF (F_TCARR.SPF) / CUST_800.SPF

Note on markers _M2 to _M13

When the swivel data set or tool is changed, the linear axes are recovered using the last retraction mode (modal).

If this behavior is not desired in milling/turning, the corresponding calls must be commented out with a semicolon (;). The E_SWIV_H or F_SWIV_H cycle is called in milling/turning (see markers _M2 to _M9) in the CUST_800.SPF manufacturer cycle.

Parameter E_SWIV_H (Par1, Par2, Par3)

- Par1: Number of swivel data set (_TC1)
- Par2: Angle 1st rotary axis
- Par3: Angle 2nd rotary axis

Modification examples

If the rotary axes (swivel head/table) are not to be positioned during swivel data change / tool change, the call of the E_SWIV_H cycle can be commented out at the relevant markers. If the rotary axes are to move to a certain position, an angle value can be transferred to parameters Par 2, Par 3.

Note on markers _M14, _M15

Depending on the values of the retraction plane and the programmed swivel plane, it is possible that the linear axes now also travel the swiveled retraction plane while running up from the current position to the software limit switches after a block search. To avoid this problem, marker _M14 in the CUST_800.SPF is called after swiveling. The E_SP_RP(30) cycle preset there runs up to the milling retraction plane, whereby travel may be along the software limit switches. An appropriate retraction after block search can be set at marker _M15.

Note on the marker _M16

If for a swivel data set change or when de-selecting the swivel data set, you wish to traverse the rotary axes to zero, the syntax of the _M16 marker can be modified accordingly, e.g. activate block N8006 E_SWIV_H(_TC1,0,0).

Note on markers _M20 to _M31

Markers _M20 to _M31 are distinguished by machine kinematics with two rotary axes or one rotary axis. A distinction is also made between automatic rotary axes (known to the NCU) and manual (semi-automatic) rotary axes. There is only ever one valid marker for swiveling with the active swivel data set.

Note on marker _M35

Run through _M35 for block search and a swivel data set with manual rotary axes.

Note on marker _M40

This marker is called before the calculation and activation of the tool holder function as user initialization. This makes it possible for special kinematics, to calculate or rewrite the current vectors of the active swivel data set.

Boring mill application example: For parallel axes, the vector in the Z direction can be updated with the actual value of the W axis.

A STOPRE is integrated for this marker.

Manufacturers can test as to whether this STOPRE for marker M40 in CUST_800.spf complies with their functional requirements. In this case, one must weigh up between velocity (without STOPRE) and functionality for interrupt conditions (RESET) for a swivel plane. This also depends on the type of machine kinematics.

Note on labels _M41 to _M45 and _M47, _M48

At marks _M41 to _M45, the retraction strategy can be modified before the swiveling of the rotary axes and at marks _M47 and _M48 before the alignment of the tool.

The retraction to the MCS positions Z or Z, XY is made considering the tool length components without tool cutter deselection.

Note on marker _M46

Retraction before swiveling after a block search can be set at marker _M46. Variable _E_VER is 1 if it is a milling technology program.

Note on markers _M57 to _M59

Markers _M57 to _M59 are used for swiveling in JOG mode and active 5-axis transformation (TRAORI).

Note on "Correct tool"

"Correct tool" requires that a 5-axis transformation is set up which is equivalent to the corresponding swivel data set. The programming section for "Correct tool" is integrated in the markers _M20, _M21, _M22 and _M30. The first 5-axis transformation is called with TRAORI(1).

Note on tool change + swivel

In general, the swivel (CYCLE800) and tool change functions for a machine are independent of each other. Thus, the swiveled work plane can be retained in a technological sequence with multiple tools, e.g. centering, drilling, tapping.

If the rotary axes of the active swivel data set are involved in the mechanical sequence of the tool change or have to be retracted, this must be taken into account in the tool change program. After the tool change, the rotary axis positions are approached as prior to the tool change. If linear axes (geometry axes) are also involved in the tool change, the rotations in the NC (swivel frame) must not be deleted. Rather, the linear axes can be positioned as machine axes using the G153 or SUPA commands.

Note on swiveling without active tool compensation

If swiveling the rotary axes without active tool cutting edge (D0) is not possible, then you can adapt this in cycle CUST_800.SPF:

```
_M40:
IF ((NOT $P_TOOL) AND _TC1)
  LOOP
  MSG ("No tool cutting edge active")
  M0
  STOPRE
  ENDLOOP
ENDIF
GOTOF _MEND
```

9.7.7 Indirectly programming the name of the swivel data set**Function**

The **_TC_CHANGE** GUD variable defined in the channel is a string with a length of 32. Using this variable, the user can determine before the CYCLE800 call which swivel data set is called in the CYCLE800, irrespective of what is programmed in the call itself.

The **_TC_CHANGE** GUD variable is re-initialized with each RESET. At the end of the CYCLE800, the variable is also deleted. This means that the variable must be re-described before each new CYCLE800 call.

The swivel data set is changed in the CYCLE800 following the CUST_800 jump to the **_M40** marker. This means that the variable can also be described in the CUST_800.

Programming example

```
N10 _TC_CHANGE="HEAD_2"
N20 CYCLE800(0,"HEAD_1",100000,57,0,0,0,0,0,0,0,0,0,0,-1,100,1)
N30 M0 ;* HEAD_2 is active

N40 _TC_CHANGE="HEAD_3"
N50 CYCLE800(0,"HEAD_1",100000,57,0,0,0,0,0,0,0,0,0,0,-1,100,1)
N60 M0 ;* HEAD_3 is active

N70 CYCLE800(0,"HEAD_1",100000,57,0,0,0,0,0,0,0,0,0,0,-1,100,1)
N80 M0 ;* HEAD_1 is active

N90 _TC_CHANGE="TABLE_2"
N100 CYCLE800(0,"TABLE_1",100000,57,0,0,0,0,0,0,0,0,0,0,-1,100,1)
N110 M0 ;* TABLE_2 is active
N120 M2
```

Note

The `_TC_CHANGE` variable must be re-described before each new CYCLE800 call.

Programmed restrictions in the CYCLE800

The swivel data set named in `_TC_CHANGE` is checked for the following properties:

- The swivel data set to be activated must be enabled.
- The following parameters must match the swivel data set selected on the screen:
 - `$TC_CARR23` (swivel data set type)
 - `$TC_CARR37` (swivel mode, units digit)
 - `$TC_CARR37` (selection of follow-up, ten-thousands digit)
 - The mode set (Bit 6, 7) on the screen
(axis-by-axis/solid angle/projection angle/direct) to `TC_CARR37`

9.7.8 CYCLE9960 workpiece - Measurement of machine kinematic**Function**

With the Measurement of machine kinematic function, it is possible to calculate the geometric vectors used to define the tool carrier by measuring the position of the ball in space.

Measurement essentially involves scanning three to twelve positions of a measuring ball for each rotary axis with a workpiece probe. The ball positions can be defined in accordance with user specifications so that they correspond to the geometric ratios on the machine. The ball positions are automatically positioned by CYCLE9960 by positioning the rotary and linear axes. Positioning can be intervened in the `CUST_800_M996`.

Application

The Measurement of machine kinematic function (CYCLE9960) can be used to determine the data that is relevant to transformations in the case of kinematic transformations that involve rotary axes TCARR.

Options:

- Redetermination of swivel data sets or kinematic chain after initial commissioning with drawing dimensions
 - Commissioning the machine
 - Use of swivel-mounted workholders as TCARR
- Checking of swivel data sets or kinematic chain
 - Service following collisions
 - Checking the kinematics during the machining process

9.7 Swiveling

Kinematics with manual axes (manually adjustable rotary tables, swivel-mounted workholders) can be measured in the same way as kinematics with NC-controlled rotary axes.

When CYCLE9960 is started, a swivel data set with basic data (kinematic type, drawing dimensions of the vectors) must be parameterized.

Preconditions

The following preconditions must be met in order to use CYCLE9960 (measure kinematics):

- SIEMENS measuring cycles package is installed
- Workpiece probe is calibrated
- Calibration ball is mounted
- Tool carrier with orientation capability ($MD18088 \$MN_MM_NUM_TOOL_CARRIER > 0$) is set up
- The basic geometry of the machine (X, Y, Z) is rectangular and referenced
The right angle refers to the workpiece spindle and should be preferably checked using a test mandrel.
- Defined position of the rotary axes involved in the transformation
- Defined traverse directions in compliance with the standard of all axes involved in the transformation according to ISO 841-2001 and/or DIN 66217 (righthand rule)
- Defined size of the log (protocol) file ($MD11420 \$MN_LEN_PROTOCOL_FILE \geq 20$)

More information

The precise procedure when measuring and programming CYCLE9960, including examples, is provided in the

Programming Manual Measuring Cycles.

9.8 Load-dependent controller setting

9.8.1 Adapt to load CYCLE782

Function

The workpiece and the clamping influence the dynamic response of the machine due to their moment of inertia. During axis movements, inaccuracies in workpiece machining can arise. Using cycle CYCLE782, you can automatically adapt the controller settings of the drive or the dynamic response parameters of the axes to a specific situation. In this way, you achieve faster and more accurate machining on the workpiece.



Software option

To use this function, you need the software option:
"Intelligent load adjustment"

Note

If the option "Intelligent load adjustment" is set, bit 20 is activated for MD19321 \$ON_TECHNO_FUNCTION_MASK_1 and the softkeys "Adapt to load" and "Adaptation" are displayed on the input mask.

Configuring the input mask form

Set the load response when opening the input mask using the following channel-specific machine data:

MD52212 \$MCS_FUNCTION_MASK_TECH		General function mask for all technologies
Bit 18	Enable determining the complete load: Axis empty + workpiece	
= 0	The selection separately determine load is enabled in cycle CYCLE782. The determined load value then only refers to the current workpiece. (default)	
= 1	The total load is determined in cycle CYCLE782. The determined load value corresponds to the sum of: Axis empty + workpiece.	

Further setting

MD53350 \$MAS_ILC_BASE_VALUE		Basic inertia of the axis
= 0	(default)	The result for ascertaining loading of the axis without workpiece is stored in this machine data. This value must be determined only once when commissioning the machine. It is required for support of the function Ascertain loading separately with CYCLE782.

Value output for linear axes

Loading can be determined or specified for linear and also for rotary axes. For linear axes, a mass, and for rotary axes, a moment of inertia is normally displayed in the measurement result display. For linear axes, however, the value display may differ depending on the drive type, e.g:

Drive of the linear axis	Value	Unit
Spindles or rotary axes	Moment of inertia	kgm ²
Rotary via ball screw, linear motor or gear	Loading	kg

Additional information

Additional information on using CYCLE782 and its program structure is provided under:

- Universal/Turning/Milling/Grinding Operating Manual
- NC Programming Programming Manual

9.8.2 CYCLE782: Help screens

For support, help screens are integrated in the input mask for cycle CYCLE782.

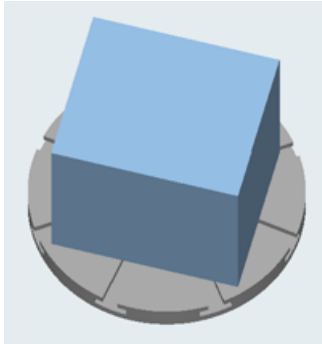
The help screens are correctly displayed in the input form if the following channel-specific machine data are set as appropriate for the machine configuration:

MD52206 \$MCS_AXIS_USAGE[n]	Meaning of the axes in the channel
MD52207 \$MCS_AXIS_USAGE_ATTRIB[n]	Attributes of the axes

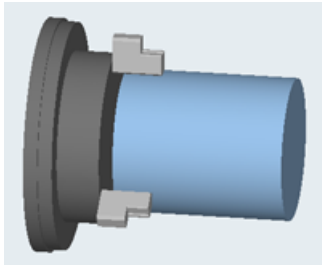
You will find a detailed description of MD52206 and MD52207 in Chapters "Setting the technology for simulation (Page 121)" and "Activating turning/milling/drilling/grinding technologies (Page 129)".

Auxiliary screens for axes

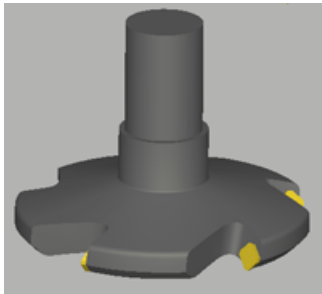
Depending on the type of axis, the following help screens are displayed, for example:



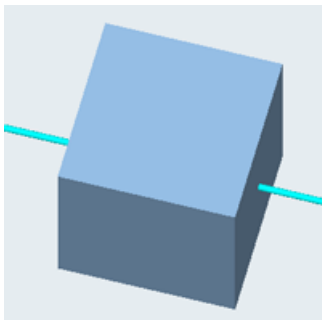
C axis



Main spindle



Tool spindle around Z



Linear axis

9.8.3 Manufacturer cycle CUST_782.SPF

Function

Cycle CUST_782 is called internally by CYCLE782 (determine the load). It is used to execute the required traverse distances and traverse conditions for determining the load. These are specific to the machine and must be set accordingly by the machine manufacturer.

Adapting manufacturer cycle CUST_782.SPF

You can execute the following actions with the aid of the _MODE parameter:

Marker	Action
_M1	Call at the start of CYCLE782
_M10	Go to starting position
_M11	Traverse movements for determining the load
_M12	Traverse movements for determining the friction
_M19	Go to end position
_M99	Call at the end of CYCLE782

Resetting value for the moment of inertia

Before each determination of the load, the moment of inertia last determined can be reset to a defined value in the CUST_782 movement profile by writing the following variables.

\$VA_RESET_INERTIA_TOTAL[ax]		Resetting the moment of inertia estimator
= -1	Reset moment of inertia and load	
= 0	No action (default when reading)	
= 1	Reset moment of inertia	
ax = axes in the system		

To ensure that the old moment of inertia does not have any influence on the new result, reset the last estimated moment of inertia prior to each new traversing with the parameterization = 1. This will allow the estimation results determined to be reproduced.

Should you wish to reset the estimated load torque at the same time (e.g. the friction force), use the parameterization = -1.

Note

System variable \$VA_RESET_INERTIA_TOTAL is only available for SINAMICS drives. It describes the SINAMICS parameter p1565.

Additional information

Additional information regarding the set parameters, commissioning and the boundary conditions for the axis functions is provided in:

- Function Manual SINAMICS S120 Drive Functions
- SINAMICS S120/S150 List Manual

9.9 High Speed Cutting (HSC)

9.9.1 High speed settings: Configuring CYCLE832

Function

The High Speed Settings function (CYCLE832) is used to preset data for the machining of free-form surfaces so that optimum machining is possible. The call of CYCLE832 contains three parameters:

- Tolerance
- Machining type (technology)
- Input of the orientation tolerance
The 3rd parameter is significant only for machines with multi-axis orientation transformation (TRAORI). By default, it is assigned the value 1.

Tolerance / orientation tolerance

The tolerance value for the linear axes (geometry axes) is transferred to the CNC with the NC command CTOL. $CTOL = \sqrt{3} \cdot \text{tolerance value}$.

Tolerance value = 0: Tolerance value (CTOL) is loaded from the following SDs:

SD55446 \$SCS_MILL_TOL_VALUE_ROUGH	Tolerance value for roughing CYCLE832
SD55447 \$SCS_MILL_TOL_VALUE_SEMIFIN	Tolerance value for semi-finishing CYCLE832
SD55448 \$SCS_MILL_TOL_VALUE_FINISH	Tolerance value for finishing CYCLE832

Example: CYCLE832(0,_ROUGH,1) → tolerance value from SD55446 is used.

Machining type and technology G group 59

The machining types of the technology G group 59 are permanently assigned in CYCLE832 or in CUST_832.SPF:

Machining type	Plain text	Technology G group 59	Array index
Deselection	_OFF	DYNNORM	0
Roughing	_ROUGH	DYNROUGH	2
Semi-finishing	_SEMIFIN	DYNSEMIFIN	3
Finishing	_FINISH	DYNFINISH	4
Roughing *)	_ORI_ROUGH	DYNROUGH	2
Semi-finishing *)	_ORI_SEMIFIN	DYNSEMIFIN	3
Finishing *)	_ORI_FINISH	DYNFINISH	4

*) With entry of an orientation tolerance

Dynamic parameters can be adapted to the respective machining operation with the technology G groups. Using the commands of technology G group 59, the value of the following channel and axis-specific machine data is activated using the corresponding **array index**:

MD20600 \$MC_MAX_PATH_JERK[0..4]	Path dependent maximum jerk.
MD20602 \$MC_CURV_EFFECT_ON_PATH_ACCEL[0..4]	Influence of path curvature on path dynamic response.
MD20603 \$MC_CURV_EFFECT_ON_PATH_JERK[0..4]	Influence of path curvature on path jerk.
MD32300 \$MA_MAX_AX_ACCEL[0..4]	Maximum axis acceleration.
MD32310 \$MA_MAX_ACCEL_OVL_FACTOR[0..4]	Overload factor for axial velocity jumps.
MD32431 \$MA_MAX_AX_JERK[0..4]	Maximum axial jerk for path motion.
MD32432 \$MA_PATH_TRANS_JERK_LIM[0..4]	Maximum axial jerk at the block transition in continuous-path mode.

Note

During the optimization of the machine axes, the values (note array index) of the specified machine data must be set correctly.

Machining type, deselection

When CYCLE832 is deselected, the G groups are programmed for the settings during the program run time; these settings are declared in machine data MD20150 \$MC_GCODE_RESET_VALUES[]. This involves the G groups 1, 4, 10, 12, 21, 24, 30, 45, 59, 61.

The contour and orientation tolerance is deactivated (CTOL=-1, OTOL=-1).

See also

Check program for Advanced Surface settings (<https://support.industry.siemens.com/cs/ww/en/view/78956392>)

9.9.2 Manufacturer cycle CUST_832.SPF

Function

In contrast to the settings (G functions) by CYCLE832.SPF, these settings can be modified in the manufacturer cycle CUST_832.SPF.

The following markers are prepared in CUST_832.SPF:

<code>_M_NORM:</code>	For deselection <code>_OFF CYCLE832</code>
<code>_M_FINISH:</code>	For finishing <code>_FINISH</code> and <code>_ORI_FINISH</code>

`_M_SEMIFINISH:` For semi-finishing `_SEMIFIN` and `_ORI_SEMIFIN`
`_M_ROUGH:` For roughing `_ROUGH` and `_ORI_ROUGH`

The following G functions are programmed in `CUST_832.SPF`:

```
SOFT
COMPCAD
G645
FIFOCTRL
UPATH
;FFWON
```

Dependent on the machining type:

```
DYNNORM
DYNFINISH
DYNSEMIFIN
DYNROUGH
```

9.9.3 Top Surface: CYCLE832 extension

Function

Besides Advanced Surface, the High Speed Settings (CYCLE832) function also offers the extension Top Surface. You can use Top Surface to improve the quality of free-form surfaces. If set actively, the continuous-path control is optimized within contour tolerances. The CYCLE832 call also contains the smoothing parameter.



Software option

To use this function, you need the software option:
 "Top Surface"

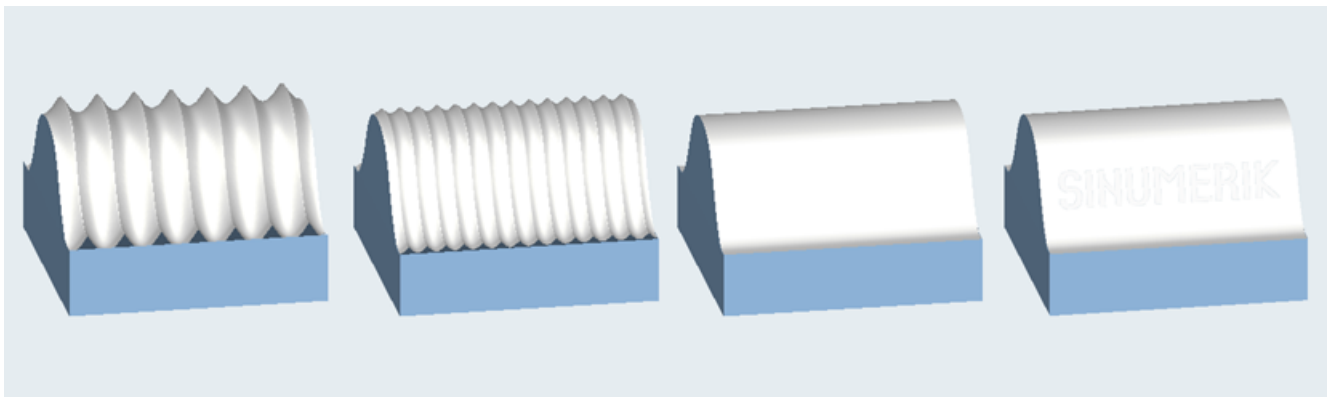


Figure 9-16 Roughing, rough-finishing, finishing or smooth-finishing

Tolerance

The tolerance value for Top Surface is transferred as a contour tolerance to the NC with command `CTOL`.

Setting up parameters

Set the following machine and setting data to set up the Top Surface extension:

MD28071 \$MC_MM_NUM_SURF_LEVELS	Dimensioning the COMPSURF function (DRAM)
= 6	Defines the size of the internal data structures of the COMPSURF function. With larger values, the attainable surface quality increases, but so do the computing time and memory requirements.

SD55220 \$SCS_FUNCTION_MASK_MILL_TOL_SET	Function mask for high-speed settings CYCLE832
Bit 2	Mold-making function can be selected in High Speed Settings mask
= 0	The best available mold making function is automatically used (default setting): - Top Surface not active → Advanced Surface - Top Surface active → Top Surface
= 1	The mold making function (Advanced Surface or Top Surface) can be selected in the input mask.
Bit 4	Display selection "Smoothing"
= 0	Do not display selection "Smoothing" in the mask (corresponds to Smoothing = yes)
= 1	Display selection "Smoothing" in the mask

MD52441 \$MCS_LIMIT_RES_PATH_SPEED	Limitation of the resulting speed of the path movement
mm/min	

MD52441 \$MCS_LIMIT_RES_PATH_SPEED can be used to limit or reduce the speed of the path movement resulting from the axial limitation values in critical program sections.

The resulting path speed is limited to the value entered in the machine data. No limitation is effective with the value 0.

In addition, when CYCLE832 is deselected or on reset or part program end, the limitation is deactivated.

Extension in manufacturer cycle CUST_832.SPF

The following labels are provided in manufacturing cycle CUST_832.spf for Top Surface:

31 _M_TOP_SURFACE_FINISH:	Finishing with COMPSURF
32 _M_TOP_SURFACE_SEMIFINISH:	Semi-finishing with COMPSURF
33 _M_TOP_SURFACE_ROUGH:	Roughing with COMPSURF
34 _M_TOP_SURFACE_PRECISION:	Smooth finishing with COMPSURF

The following G commands are programmed in labels 31 to 33:

```
SOFT
COMPSURF
G645
FIFOCTRL
UPATH
```

9.9 High Speed Cutting (HSC)

; FFWON

Depending on the machining type used, the following G commands of G group 59 are programmed:

DYNFINISH

DYNSEMIFIN

DYNROUGH

DYNPREC

9.10 Measuring cycles and measurement functions

9.10.1 Measuring cycles and measurement functions, general

Requirement

Two types of electronic probes are used for measuring:

- Probe to measure the workpiece
- Probe to measure the tool

The electronic probe must be connected on the PPU or the NX module.

The electronic probe is called a measuring input in the following document.

Setting the switching behavior

You set the electrical polarity of the connected probe using the following general machine data:

MD13200 \$MN_MEAS_PROBE_LOW_ACTIVE		Polarity change of the probe
= 0	Probe in the non-deflected state, 0 V default value Probe in the deflected state, 24 V	
= 1	Probe in the non-deflected state, 24 V Probe in the deflected state, 0 V	

MD13210 \$MN_MEAS_TYPE		Measuring type for distributed drives
= 1	default value, cannot be changed	

Testing the probe function

You can test the switching function of the probe by manually deflecting it and checking the following PLC interface signals:

DB2700	General signals from the NC NC -> PLC interface							
Byte	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
DBB0	-	-	-	--	-	-	EMERGENCY OFF active	-
DBB1	System inch meas- uring system	-	-	-	-	-	Probe actuated	
							Probe 2	Probe 1

9.10 Measuring cycles and measurement functions

To test the switching behavior and the measured value transfer, use an NC test program with, for example, the following NC commands:

MEAS	Measurement with deletion of distance-to-go
\$AC_MEA[n]	Check the switching operation, n = measurement input number
\$AA_MW[axis name]	Measured value of the axes in workpiece coordinates
\$AA_MM[axis name]	Measured value of the axes in machine coordinates

Example of a test program

Program code	Comment
%_N_PRUEF_MESSTASTER_MPF	
\$PATH=/_N_MPF_DIR	;Testing program probe connection
N00 DEF INT MTSIGNAL	;Bit memory to check the switching state
N05 G17 G54 T="3D_Taster" D1	;Select tool geometry for probe
N10 M06	;Activate tool
N15 G0 G90 X0 F150	;Starting position and meas. velocity
N20 MEAS=1 G1 X100	;Measurement at measuring input 1 in the X axis
N30 MTSIGNAL=\$AC_MEA[1]	;Switching operation at the 1st measuring input completed, ;YES / NO
N35 IF MTSIGNAL == 0 GOTOF _FEHL1	;Evaluation of the signal
N40 R1=\$AA_MM[X]	;Save measured value in machine coordinates at R1
N45 R2=\$AA_MW[X]	;Save measured value in workpiece coordinates at R2
N50 M0	;Check measured value in R1/R2
N55 M30	
N60 _FEHL1: MSG ("Probe does not switch!")	
N65 M0	
N70 M30	

General function settings for measuring cycles

MD51740 \$MNS_MEA_FUNCTION_MASK	Measuring cycles function mask
Bit 0	Workpiece, monitoring the calibration status
Bit 1	Workpiece, length reference of the probe in the infeed axis with relation to the sphere circumference
= 0	Tool length L1, referred to the center of the probe sphere
= 1	Tool length L1, referred to the sphere circumference of the probe sphere
Bit 2	Workpiece, offset into tool, tool carrier taken into account
Bit 4	Workpiece, measuring probe permanently affixed to the machine (not in a spindle)
Bit 15	Workpiece, traverse measuring block with path behavior G60
Bit 16	Tool, turning technology, consideration of tool carriers

Note

After changing this machine data, the probe must be re-calibrated.

MD52740 \$MCS_MEA_FUNCTION_MASK		Measuring cycles function mask
Bit 0	Workpiece probe connected to measurement sensing input 2	
= 0	Workpiece probe at the CNC measuring input 1, active - default	
= 1	Workpiece probe at the CNC measuring input 2, active	
Bit 1	Workpiece, turning technology, measuring in the third geometry axis (Y)	
Bit 2	Calculation of the correction angle with/without active orientation transformation	
Bit 16	Tool probe connected to measurement sensing input 2	
= 0	Tool probe at the CNC measuring input 1, active	
= 1	Workpiece probe at the CNC measuring input 2, active - default	
Bit 17	Tool measurement turning, recoding of cutting edge position	

SD54740 \$SNS_MEA_FUNCTION_MASK		Measuring cycles function mask
Bit 0	Workpiece, repeat measurement if TSA, TDIF exceeded	
Bit 1	Workpiece, repeat measurement with M0 if TSA, TDIF exceeded	
Bit 2	Workpiece, stop at M0 if TUL, TLL exceeded	
Bit 3	Workpiece, accept calibrated probe sphere radius in tool data	
Bit 4	Workpiece, plane can vary between calibration and measurement	
Bit 16	Tool, repeat measurement if TDIF, TSA exceeded	
Bit 17	Tool, repeat measurement with stop at M0 if TDIF, TSA exceeded	
Bit 19	Tool, milling, reduction of spindle speed during last probing	

SD55740 \$SCS_MEA_FUNCTION_MASK		Measuring cycles function mask
Bit 0	Workpiece, collision monitoring for intermediate positioning	
Bit 1	Workpiece, coupling of spindle position with coordinate rotation around the infeed axis	
Bit 2	Workpiece, reverse positioning direction if spindle position is coupled	
Bit 3	Workpiece, no repetition if probe does not switch	
Bit 4	Workpiece, 1st probing for every measurement with rapid measuring feed	
Bit 5	Workpiece, rapid retraction following measurement	
Bit 6	Workpiece, deactivate probe during spindle positioning	
Bit 7	Kinematics, scaling on the basis of the input values of the rotary axis vectors	
Bit 8	Kinematics, measuring with active swiveling (TCARR) or active TRAORI	
Bit 15	Workpiece, calibrating radius with starting point in center of ring, JOG mode only	
Bit 16	Tool, collision monitoring for intermediate positioning	
Bit 17	Tool, no repetition if probe does not switch	
Bit 19	Tool, rapid retraction following measurement	

9.10.2 Manufacturer and user cycle CUST_MEACYC.SPF

Function

The CUST_MEACYC.SPF is part of the measuring cycle functionality. It is called in every measuring cycle before and after executing the measurement task. The CUST_MEACYC.SPF acts in the same way when measuring in the JOG mode and measuring in the AUTOMATIC mode.

You can use the CUST_MEACYC.SPF to program and execute sequences that are necessary before and/or after a measurement (e.g. activating/deactivating a probe).

Note

Copy the CUST_MEACYC cycle before performing the changes from the directory NC data/Cycles/Standard cycles to the directory NC data/Cycles/Manufacture cycles or NC data/Cycles/User cycles. Changes for CUST_MEACYC only take effect in these directories.

CUST_MEACYC structogram

Sequence in the AUTOMATIC mode, using measure bore as example, CYCLE977 with probe type 712.

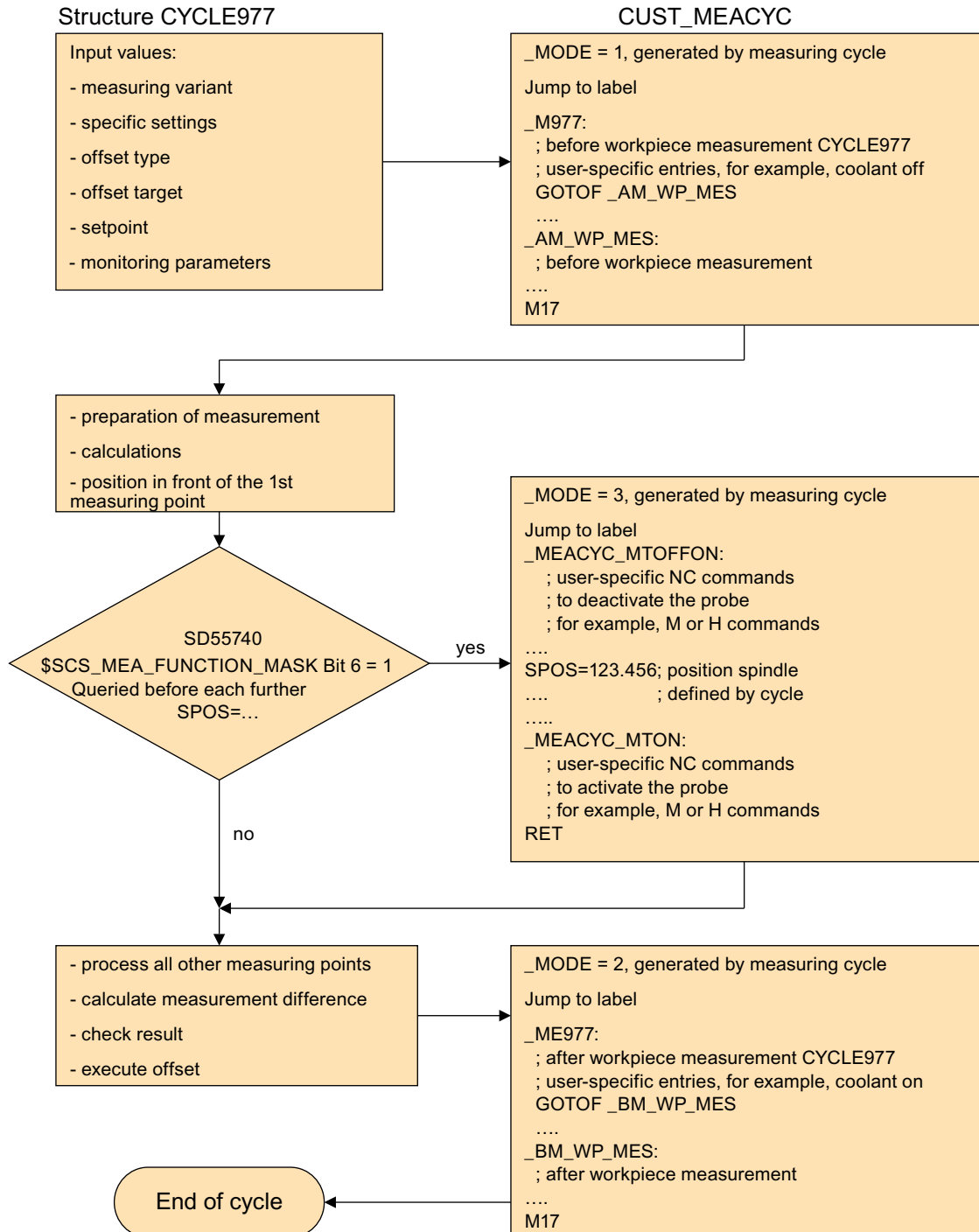


Figure 9-17 Structure: CYCLE977 / CUST_MEACYC

Note**Commenting out spindle SPOS commands**

If the spindle SPOS command is to be commented out in the CUST_MEACYC.SPF cycle, set the cycle setting in SD55740 \$SCS_MEA_FUNCTION_MASK to active with Bit 1 = 1.

You can see how to comment out a spindle SPOS command in the CUST_MEACYC.SPF cycle in the following example:

Example

```
;
; *#1 Example application for spindle pre-positioning if there is no coupling
; *##1 between spindle and the coordinate system active!
; *#1 This example can be removed if necessary!
; *#2 User example for spindle positioning, if you do not have a coupling
; *##2 between the spindle and the coordinate system!
; *#2 If this lines are not necessary, the user can delete this example!
;
; cycle is calling from the AUTOMATIC-mode AND the spindle is positionable
IF NOT(_OVI[14]) AND NOT(_SPOS_MOD)
  ; no coupling between the spindle and the coordinate system!
  ; SD55740, Bit 1 = 0
  IF ($SCS_MEA_FUNCTION_MASK B_AND 'B10')==0
    ; SPOS=0 ; <===== commented out
  ENDIF
ENDIF
M17
;
```

9.10.3 Measuring in the JOG mode**Preconditions**

You have already made the settings from the previous section "Measuring cycles and measurement functions, general (Page 235)".

Workpiece measurement

For milling:

- The probe has been inserted in the tool spindle.
- The probe has been selected in the tool list as type 7xx (probe).
- The probe is activated as tool in the current NC channel.

Tool measurement

To measure tools, an appropriate probe must be located in the machine space so that this can be reliably and safely reached with a tool in the spindle.

The following tool types are supported with the tool measurement:

- Milling technology: Tool types 1xx and 2xx
- Turning technology: Tool type 5xx, 1xx, 2xx

For the specified tool types, the tool lengths and the tool radii can be measured.

Setting general data

SD54782 \$SNS_J_MEA_FUNCTION_MASK_TOOL		Settings for the Tool measurement in JOG input mask
Bit 2	Activates the "Enable automatic tool measuring" function.	
Bit 3	Selects the tool probe calibration data field, enable.	
Bit 10	Selects the individual cutting edge measurement	
Bit 11	Selects spindle reversal when calibrating in the plane	

MD51784 \$MNS_J_MEA_T_PROBE_APPR_AX_DIR [i]		Measuring in JOG: Approach direction in the plane to the tool probe
-2	Negative direction= 2nd measuring axis	
-1	Negative direction= 1st measuring axis (default setting)	
0	Positive direction 1st measuring axis	
1		
2	Positive direction 2nd measuring axis	

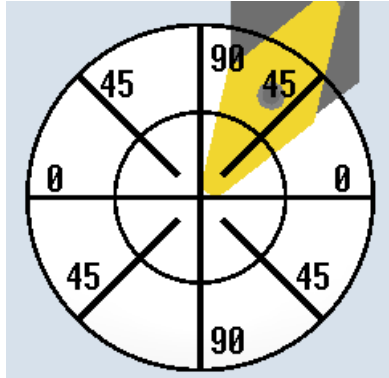
MD52750 \$MCS_J_MEA_FIXPOINT		Z value for measurement with respect to a fixed point
mm		

With MD52750, the Z position for "Measuring: fixed point" is saved. In milling technology, the fixed point serves as the reference point for manual tool measurement in JOG. The value can be set via the input mask "Calibration: fixed point".

MD52751 \$MCS_J_MEA_MAGN_GLAS_POS		Zoom-in position for tool measurement
mm		

9.10 Measuring cycles and measurement functions

[0]	Position in the 1st axis
[1]	Position in the 2nd axis



If a magnifying glass is available on the machine, it can be used to determine the tool geometry. In this case, SINUMERIK Operate calculates the tool offset data from the known positions of the tool carrier reference point and the cross-hairs of the magnifying glass.

Checking and setting the general machine data

MD11450 \$MN_SEARCH_RUN_MODE	Parameterization for search run
Bit 1 = 1	Automatic ASUP start after output of action sets. Alarm 10208 is not output until the ASUP is ended (see also MD11620 \$MN_PROG_EVENT_NAME).

MD11604 \$MN_ASUP_START_PRIO_LEVEL	Priorities from which ASUP_START_MASK is effective.
= 1 - 64	Priorities for ASUP_START_MASK.

Setting channel-specific machine data

MD20050 \$MC_AXCONF_GEOAX_ASSIGN_TAB[]	Assignment, geometry axis to channel axis.
[0]	For measuring in JOG mode, all of the geometry axes must be available. Preferably XYZ.
[1]	
[2]	

MD20110 \$MC_RESET_MODE_MASK	Defines the basic control settings after reset / TP end,
= 4045H	Minimum value
Bit 0 = 1	Basic control setting after power on and reset.
Bit 2 = 1	
Bit 6 = 1	
Bit 14 = 1	

MD20112 \$MC_START_MODE_MASK	Definition of the basic control setting after part program start.
= 400H	Minimum value
Bit 6 = 0	Definition of the basic control setting after NC start.

MD20310 \$MC_TOOL_MANAGEMENT_MASK	Activates tool manager functions
= 4002H	Minimum value
Bit 1 = 1	Tool manager and monitoring functions active
Bit 14 = 1	Automatic tool change for RESET and Start.

MD28082 \$MC_SYSTEM_FRAME_MASK	System frames (SRAM).
1	Default, cannot be changed.

The following channel-specific machine data acts only if the system frame actual value setting and scratching is set-up using the above mentioned machine data

MD28082 \$MC_SYSTEM_FRAME_MASK, bit 0 = 1.

MD24006 \$MC_CHSFRAME_RESET_MASK	Active system frames after reset.
Bit 0 = 1	Reset response of the system frames or actual value setting.

MD24007 \$MC_CHSFRAME_RESET_CLEAR_MASK	Clear system frames on reset.
Bit 0 = 0	Clear response of the system frames on reset.

Note

If you create the conditions described in this section and you have set and checked the machine setting data, then in JOG mode, you can measure a workpiece using a workpiece probe at a milling machine!

In JOG mode, you can measure a tool using a tool probe at a milling machine or lathe!

A description is provided in the following chapters as to which settings you can make in order to adapt measuring to the specific requirements of your particular machine.

9.10.3.1 Measuring workpieces at the milling machines

Measuring in the "Machine" operating area can be appropriately adapted to the specific requirements using the following channel-specific general machine data and channel-specific cycle setting data.

General configuration machine data

SD55630 \$SCS_MEA_FEED_MEASURE	Measuring feedrate for workpiece measurement in AUTOMATIC and JOG
300	Default value

Note**Measuring feedrate for workpiece measuring**

All measuring cycles use the value saved in SD54611 as the measuring feedrate value following calibration of the workpiece probe. A different measuring feedrate can be assigned for each calibration field [n].

When calibrating the probe, either the measuring feedrate from SD55630 \$SCS_MEA_FEED_MEASURE is used, or the measuring feedrate can be overwritten in the input screen when calibrating. To do this, bit 4 must be set to 1 in SD54760 \$SNS_MEA_FUNCTION_MASK_PIECE.

MD52207 \$MCS_AXIS_USAGE_ATTRIB[n]		Attributes of the axes [n] channel axis number
Bit 6	Request rotary axis as correction target for measuring	
= 0	Default value	
= 1	Rotary axis is displayed as correction target in the "Measuring in JOG" window	
Bit 9	Influence of the SPOS response of the main spindle in connection with measuring cycles	
= 0	The positioning response of the tool spindle is not affected	
= 1	The measuring cycles do not position the tool spindle (no SPOS)	

Note

If the tool spindle is fundamentally not SPOS-capable, bit 9 must be set to 1.

MD51751 \$MNS_J_MEA_M_DIST_MANUELL		Measurement path in mm, before and after the measuring point
= 10	Default value	

MD51757 \$MNS_J_MEA_COLL_MONIT_FEED		Position feedrate in mm/min, in the working plane with active collision monitoring
= 1000	Default value	

MD51758 \$MNS_J_MEA_COLL_MONIT_POS_FEED		Position feedrate in mm/min, in the infeed axis with active collision monitoring
= 1000	Default value	

MD51770 \$MNS_J_MEA_CAL_RING_DIAM[n]		Pre-assignment of the calibration ring diameter in mm specifically for the calibration data sets
= -1	Default value	

MD51772 \$MNS_J_MEA_CAL_HEIGHT_FEEDAX[n]	Pre-assignment of the calibration height in mm in the infeed axis, specifically for the calibration data sets
= -99999	Default value

General cycle setting data

SD54780 \$SNS_J_MEA_FUNCTION_MASK_PIECE	Configuration of the input screens for measuring in the "Machine" operating area in JOG mode.
= 512	Default value
Bit 2 = 1	Activates measurement with an electronic probe.
Bit 3 = 1	Selects the probe calibration data, enable.
Bit 6 = 1	Selects work offset correction in the basic reference (SETFRAME), enable.
Bit 7 = 1	Selects work offset correction in the channel-specific basic frames, enable.
Bit 8 = 1	Selects work offset correction in the global basic frames, enable.
Bit 9 = 1	Selects work offset correction in adjustable frames, enable
Bit 16 = 1	Selects workpiece measurement with spindle reversal
Bit 17 = 1	Selects Align workpiece probe in the switching direction
Bit 28 = 1	Selects adjust tool length, enable

9.10.3.2 Measuring tools at the milling machines

Measuring feed for tool measurement in JOG and AUTOMATIC

SD55628 \$SCS_MEA_TP_FEED_MEASURE	Calibrate measuring feed for tool probe and measure tool with stationary spindle.
= 300	Default value

Note

Measuring feed for tool measuring

All measuring cycles use the value saved in SD54636 or SD54651 as the measuring feed after the tool probe has been calibrated. A different measuring feed can be assigned for each calibration field [n].

When calibrating the probe, either the measuring feed from SD55628 SCS_MEA_TP_FEED_MEASURE is used, or the measuring feed can be overwritten in the input mask when calibrating. To do this, SD54762 \$SNS_MEA_FUNCTION_MASK_TOOL bit 4 must be set to 1.

MD51757 \$MNS_J_MEA_COLL_MONIT_FEED	Position feedrate in mm/min, in the working plane for active collision monitoring
= 1000	Default value

MD51758 \$MNS_J_MEA_COLL_MONIT_POS_FEED	Position feedrate in mm/min, in the infeed axis for active collision monitoring
= 1000	Default value

In the following setting data, index [k] stands for the number of the current data field (probe number -1) of the probe.

SD54633 \$SNS_MEA_TP_TYPE[k]	Probe type, cube/disk
= 0	Compatibility (measuring cycles: cube, turning surface shows probe, milling surface shows disk)
= 101	Disk in XY, working plane G17
= 201	Disk in ZX, working plane G18
= 301	Disk in YZ, working plane G19
= 2	Probe head
= 3	Cube

The following setting data is used to define in which axes and directions it is possible to calibrate a tool probe.

SD54632 \$SNS_MEA_T_PROBE_ALLOW_AX_DIR[k]	Axes and directions for "calibration"
= 133	Default value

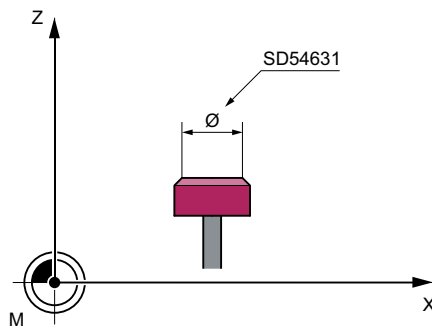
Decimal place		
ONES	1st axis	
	= 0	axis not possible
	= 1	only minus direction
	= 2	only plus direction
	= 3	both directions
TENS	2nd axis	
	= 0	axis not possible
	= 1	only minus direction
	= 2	only plus direction
	= 3	both directions
HUNDREDS	3rd axis	
	= 0	axis not possible
	= 1	only minus direction
	= 2	only plus direction
	= 3	both directions

Example

If the general setting data SD54632 \$SNS_MEA_T_PROBE_ALLOW_AX_DIR[k] has the value 123, the tool probe is calibrated as follows in the G17 plane:

- X in both directions
- Y only in plus direction
- Z only in minus direction

SD54631 \$SNS_MEA_TP_EDGE_DISK_SIZE[k]	Effective diameter of the tool probe for length measurement.
= 0	Default value

**General cycle machine/setting data for tool probe**

MD51780 \$MNS_J_MEA_T_PROBE_DIAM_RAD[k]	Effective diameter of the tool probe for radius measurement.
= 0	Default value

SD54634 \$SNS_MEA_TP_CAL_MEASURE_DEPTH[n]	Distance between the tool probe upper edge and the milling tool lower edge.
= 2	Default value

MD51752 \$MNS_J_MEA_M_DIST_TOOL_LENGTH	Measurement path for tool length measurement with rotating spindle for the JOG mode.
= 2	Default value

MD51753 \$MNS_J_MEA_M_DIST_TOOL_RADIUS	Measurement path for tool radius measurement with rotating spindle for JOG mode.
= 1	Default value

MD51786 \$MNS_J_MEA_T_PROBE_MEASURE_DIST	Measurement path to calibrate the probe or for measurements with stationary spindle for JOB mode.
= 10	Default value

General cycle setting data when measuring with rotating spindle

SD54670 \$SNS_MEA_CM_MAX_PERI_SPEED[0]	Maximum permissible peripheral speed of the tool to be measured.
= 100	Default value

SD54671 \$SNS_MEA_CM_MAX_REVOLUTIONS[0]	Maximum permissible tool speed of the tool to be measured. The speed is automatically reduced when exceeded.
= 1000	Default value

SD54672 \$SNS_MEA_CM_MAX_FEEDRATE[0]	Maximum permissible feedrate to probe the tool to be measured at the probe.
= 20	Default value

SD54673 \$SNS_MEA_CM_MIN_FEEDRATE[0]	Minimum feedrate for the first probing of the tool to be measured at the probe. This avoids excessively small feedrates for large tool radii.
= 1	Default value

SD54674 \$SNS_MEA_CM_SPIND_ROT_DIR[0]	Spindle direction of rotation to measure tools.
= 4	Spindle rotation the same as M4 (default value)

Note

If the spindle is already rotating when the measuring cycle is called, this direction of rotation remains independent of the setting of this data.

SD54675 \$SNS_MEA_CM_FEEDFACTOR_1[0]	Feedrate factor 1
= 10	Default value
= 0	Only single probing with the feedrate calculated by the cycle. However, as a minimum, the value from SD54673[0] \$SNS_MEA_CM_MIN_FEEDRATE.
>= 1	First probing with feedrate. However, as a minimum with the value from SD54673[0] \$SNS_MEA_CM_MIN_FEEDRATE) SD54675[0] \$SNS_MEA_CM_FEEDFACTOR_1

SD54676 \$SNS_MEA_CM_FEEDFACTOR_2[0]	Feedrate factor 2
= 0	Second probing with the feedrate calculated by the measuring cycle. This is only effective for SD54673[0] \$SNS_MEA_CM_FEEDFACTOR_1 > 0, default value
>= 1	Second probing with the calculated feedrate from SD54673[0] \$SNS_MEA_CM_MIN_FEE- DRATE feedrate factor 2. Third probing with the calculated feedrate.

Note

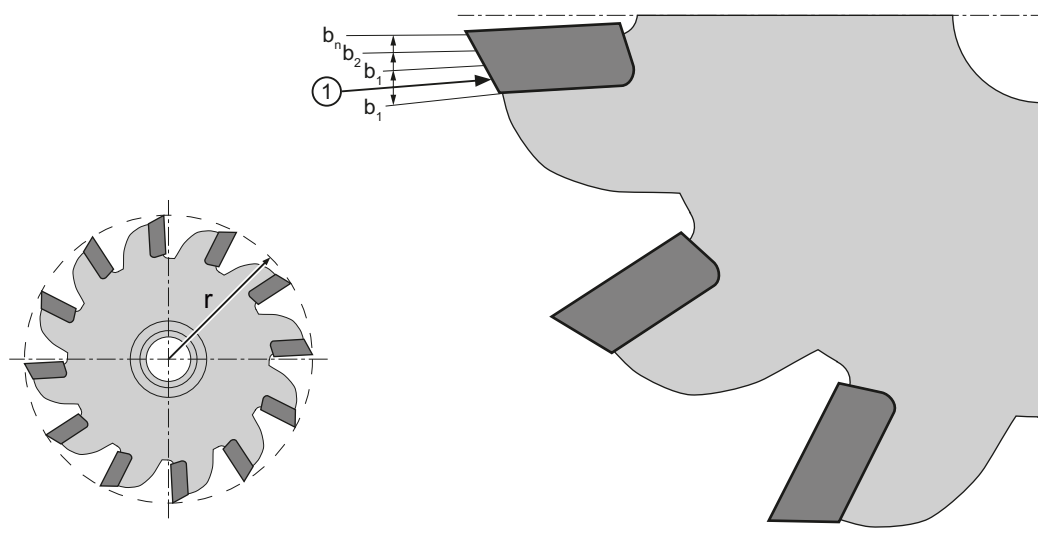
Feedrate factor 2 should be less than feedrate factor 1.

SD54677 \$SNS_MEA_CM_MEASURING_ACCURACY[0]	Specified measuring accuracy. The value of this parameter always refers to the last probing of the tool at the probe!
= 0.005	Default value

SD54692 \$SNS_MEA_T_CIRCULAR_ARC_DIST	Arc distance for the single cutting edge measurement of the radius.
= 0.25 *)	Default value

SD54693 \$SNS_MEA_T_MAX_STEPS	Maximum number of probe contacts for single cutting edge measurement of the radius.
= 10 *)	Default value

*) If the value = 0, then the "highest point on the longest cutting edge" is not sought, but the measured value of the contact with the rotating spindle taken.



r Effective radius of the tool

b Arc distance = SD54692

n Maximum number of probe contacts = SD54693

① 1st measuring point.

Figure 9-18 Single cutting edge measurement

General cycle setting data for correction using the correction tables when forming measurements with rotating spindle.

SD54691 \$SNS_MEA_T_PROBE_OFFSET		Activate the measuring result compensation
= 0	No data, default value	
= 1	Correction in the cycle. This is only effective if SD54689 \$SNS_MEA_T_PROBE_MANUFACTURER>0.	
= 2	Correction using user-defined correction table	

SD54689 \$SNS_MEA_T_PROBE_MANUFACTURER		Activates pre-configured compensation tables for several tool probe models (customer-specific)
= 0	No data, default value	
= 1	TT130 (Heidenhain)	
= 2	TS27R (Renishaw)	

9.10.3.3 Measuring tools at the turning machines

Measuring feed for tool measurement in turning operation

SD55628 \$SCS_MEA_TP_FEED_MEASURE	Calibrate measuring feed for tool probe and measure tool with stationary spindle.
= 300	Default value

Note

Measuring feed for tool measuring

All measuring cycles use the value saved in SD54636 or SD54651 as the measuring feed after the tool probe has been calibrated. A different measuring feed can be assigned for each calibration field [n].

When calibrating the probe, either the measuring feed from SD55628 is used, or the measuring feed can be overwritten in the input mask when calibrating. To do this, SD54762 \$SNS_MEA_FUNCTION_MASK_TOOL bit 4 must be set to 1.

General cycle machine/setting data for measurement path/feed

SD42950 \$SC_TOOL_LENGTH_TYPE	Assignment of the tool length offset independent of tool type.
= 2	The assignment of the tool length components is, independent of the actual tool type, always the same as for turning and grinding tools. This setting is recommended for measuring milling tools and drill bits.

MD51786 \$MNS_J_MEA_T_PROBE_MEASURE_DIST	Measurement path to calibrate the probe or for measurements with stationary spindle.
= 10	Default value

SD55628 \$SCS_MEA_TP_FEED_MEASURE	Feedrate for the calibration of a tool probe with stationary spindle.
= 300	Default value

Calibrating tool probes

Use the following general cycle machine data to set the access level as of which the "Calibrate probe" softkey is displayed. This requires that the automatic tool measurement is enabled using the setting data SD54782 (bit 2 = 1).

MD51070 \$MNS_ACCESS_CAL_TOOL_PROBE	Calibrate tool probe access level.
= 7	Access rights: Access level 7 (keyswitch 0)

= 4	Read access rights: Access level 4 (keyswitch 3), default value
= 3	Write access: Access level 3 (user)

9.10.3.4 Measuring without electronic probe in JOG

Measure workpiece

For manual measuring functions in the JOG operating mode, for which a rotating spindle is required, the following applies:

- The channel state must be reset at the instant in time that the measured value is transferred.
- The spindle can be moved or positioned either using the T,S,M menu or using the bits in DB3800.DBB5006.
- The following preconditions apply when using the DB3800.DBB5006 technology functions with fixed speed:

Setting the machine data			
MD10709 \$MN_PROG_SD_POWERON_INIT_TAB [0] = 0			setting data to be initialized
MD35035 \$MA_SPIND_FUNCTION_MASK	Bit 4 = 0 Bit 5 = 1		Spindle functions
SD43200 \$SA_SPIND_S			Speed setting value

Note

The entry in SD43200 \$SA_SPIND_S is kept at power on.

With these settings, the spindle velocity for manual operation, which is set in SD41200 \$SN_JOG_SPIND_SET_VELO, is no longer effective.

9.10.4 Measuring in the AUTOMATIC mode

Requirement



Software option

In order to use the "Measuring in AUTOMATIC" function, you require the software option: "Measuring cycles"

You have already made the settings from Chapter "Measuring cycles and measurement functions, general (Page 235)".

Workpiece measurement

Sequence in milling technology:

1. The probe has been selected in the tool list as type 7xx (probe).
2. The probe has been inserted in the tool spindle.
3. The probe is activated in the actual NC channel.

Sequence in turning technology:

1. Tool type 580 (3D probe, turning) is selected.
2. Tool is activated in the actual NC channel.

Tool measurement

To measure tools, an appropriate probe must be located in the machine space so that this can be reliably and safely reached with a tool in the spindle.

The following tool types are supported with tool measurement:

- Milling technology: Tool types 1xx and 2xx
- Turning technology: Tool types 5xx, 1xx and 2xx

For the specified tool types, the tool lengths and the tool radii can be measured.

Channel-specific cycle setting data

Using the following channel-specific cycle setting data, you can adapt the **workpiece and tool measuring** in the "Program" operating area to specific requirements. As a general rule, no changes are required.

SD55613 \$SCS_MEA_RESULT_DISPLAY		Selects the screen display of the measuring result
= 0	No screen display of the measuring result (default value).	
= 1	Screen display of the measuring result is displayed for 8 seconds.	
= 3	The measuring cycle stops an internal machine data, the measuring result is statically displayed on the screen! Continue with NC start, the measuring result screen is deselected.	
= 4	The measuring result is only displayed on the screen for cycle alarms 61303, 61304, 61305, 61306. Continue with NC start, the measuring result display on the screen is deselected.	

SD55614 \$SCS_MEA_RESULT_MRD		Setting of the program control of the measurement result display MRD
= 0	Switch off the measurement result display	
= 1	Switch on the measurement result display	

SD55623 \$SCS_MEA_EMPIRIC_VALUE[n]	Empirical values
= 0	Default value

SD55618 \$SCS_MEA_SIM_ENABLE	Measuring cycle simulation
= 0	When calling measuring cycles in conjunction with SINUMERIK Operate, ShopMill or ShopTurn simulation, the measuring cycles are skipped, default value.
= 1	Default setting: When calling measuring cycles in conjunction with SINUMERIK Operate, ShopMill or ShopTurn simulation, the measuring cycles are run through. However, no corrections are performed and there is no logging. There is no screen display of the measuring result.

Additional information

Additional information on SD55618 is provided in the
Measuring Cycles Programming Manual.

Configuring input screens for measuring cycles in the program editor

Using the following setting data, you configure the display screen and the correction options when making measurements in the AUTOMATIC mode:

- SD54760 \$SNS_MEA_FUNCTION_MASK_PIECE measure workpiece: Measuring cycles in the milling area, Chapter "Measuring workpieces, general (Page 254)"
- SD54762 \$SNS_MEA_FUNCTION_MASK_TOOL measure tool: Measuring cycles in the milling area, Chapter "Measuring tools at the milling machines (Page 258)"

Note

If you have created the prerequisites described in this chapter and you have set and checked the machine/setting data, you can perform measurements at the machine with a workpiece or tool probe in the AUTOMATIC mode!

If you wish to set up workpiece measurement on a turning machine, then additional settings are required. Refer to Chapter "Measuring workpieces at the turning machines (Page 264)" for the settings.

If you wish to set-up tool measurement using a tool probe, then you must also make additional settings. Please refer to Chapters "Measuring tools at the turning machines (Page 265)" or "Measuring tools at the milling machines (Page 258)" for the settings.

A description is provided in the following chapters as to which settings you can make in order to adapt measuring to the specific requirements of your particular machine.

9.10.4.1 Measuring workpieces, general

Using the following general and channel-specific cycle setting data, you can appropriately adapt workpiece measurement to the specific requirements in the "Program" operating area.

Requirement

You have already made the settings from the Chapter "Measuring cycles and measurement functions, general (Page 235)".

Settings

General cycle setting data

SD54760 \$SNS_MEA_FUNCTION_MASK_PIECE		Settings for the input screen, measuring cycles in AUTOMATIC, workpiece measurement.
Bit 1	Display select softkey 3D measurement	
Bit 3	Enable probe calibration field to be selected	
Bit 4	Select calibration input measuring feed ¹⁾	
Bit 6	Enable NP correction in the basis reference (SETFRAME) to be selected	
Bit 7	Enable NP correction in channel-specific basic frame to be selected	
Bit 8	Enable NP correction in global basic frame to be selected	
Bit 9	Enable NP correction in adjustable frame to be selected	
Bit 10	Enable NP correction coarse and fine to be selected	
Bit 11	Select tool offset, geometry and wear	
Bit 12	Select tool offset, not inverted and inverted	
Bit 13	Select tool offset L1, R or L1, L2, L3 R	
Bit 14	Select tool offset, zero offset (_TZL)	
Bit 15	Select tool offset, dimensional difference monitoring (_TDIF)	
Bit 16	Select workpiece measurement with spindle reversal	
Bit 17	Selects align workpiece probe in the switching direction	
Bit 18	Select number of measurements (_NMSP)	
Bit 19	Select offset with mean value generation (_TMV) ¹⁾	
Bit 20	Select experience values (_EVNUM)	
Bit 21	Select total setting-up offset	
Bit 22	Select calibration to unknown or to known center point	
Bit 24	Select calibration with/without position deviation	
Bit 25	Select zero offset when measuring the angulation of the spindle	
Bit 26	Selects tool offset, do not enable	
Bit 27	Selects measure tolerance of the linear vector for kinematics, do not enable	
Bit 28	Enable tool offset adjust length to be selected	

Channel-specific cycle setting data

SD55630 \$SCS_MEA_FEED_MEASURE		Measuring feedrate [mm/rev]
= 300	Measuring feedrate when calibrating the workpiece probe	

Note**Measuring feedrate for workpiece measuring**

All measuring cycles use the value saved in SD54611 \$SNS_MEA_WP_FEED[n] as measuring feedrate after the tool probe has been calibrated.

A different measuring feedrate can be assigned for each calibration field [n].

When calibrating the probe, either the measuring feedrate from SD55630 \$SCS_MEA_FEED__MEASURE is used, or the measuring feedrate can be overwritten in the input screen when calibrating. To do this, SD54760 MEA_FUNCTION_MASK_PIECE bit 4 must be set to 1.

SD55632 \$SCS_MEA_FEED_RAPID_IN_PERCENT	Percentage reduction
= 50	Default value of the percentage reduction of the rapid traverse velocity for internal cycle intermediate positioning without collision monitoring. The setting of the channel-specific cycle setting data SD55600 SCS_MEA_COLLISION_MONITORING must be = 0.

SD55634 \$SCS_MEA_FEED_PLANE_VALUE	Feedrate of the intermediate positioning in the working plane with active collision monitoring.
= 1000	Default value

SD55636 \$SCS_MEA_FEED_FEEDAX_VALUE	Feedrate of the intermediate positioning in the infeed axis with active collision monitoring.
= 1000	Default value

SD55638 \$SCS_MEA_FEED_FAST_MEASURE	Fast measuring feedrate
= 900	Default value

SD55640 \$SCS_MEA_FEED_CIRCLE	Feedrate with circle programming
= 1000	Default value

SD55642 \$SCS_MEA_EDGE_SAVE_ANG	Safety angle when measuring a corner (CYCLE961)
= 0	Degrees

This angle is added to the angle that the user entered between the workpiece reference edge and the 1st axis of the active coordinate system in a positive direction, and refers to the corner measurement in the automatic mode (CYCLE961).

The angle of the user specifications is designated "α0" in the parameterization screen corner measurement. For compatibility programs, the designation is _STA1.

Notice:

For compatibility programs, \$SCS_MEA_EDGE_SAVE_ANG should be set = 10.

9.10.4.2 Measuring workpieces at the milling machines

Measuring in the "Program" operating area can be adapted corresponding to the specific requirements using the channel-specific cycle setting data.

Settings

SD55625 \$SCS_MEA_AVERAGE_VALUE[n]	Number of mean values.
= 0	Default value

The following cycle setting data are written to by the measuring cycles with the "Calibrate probe" measuring function. User parameterization is not necessary here. However, after the probe has been calibrated, you can check these values and if required, evaluate the probe quality, e.g. for position deviations, no values > 0.1 mm should be reached. Otherwise, the probe must be mechanically readjusted.

The measuring feedrate at the time of calibration is used for all subsequent applications of the measuring cycles.

Note

Observe the manufacturer's instructions for the probe.

SD54600 \$SNS_MEA_WP_BALL_DIAM[n]	Effective diameter of the probe ball of the workpiece probe.
SD54601 \$SNS_MEA_WP_TRIG_MINUS_DIR_AX1[n]	Trigger point, minus direction, 1st measuring axis in the plane.
SD54602 \$SNS_MEA_WP_TRIG_PLUS_DIR_AX1[n]	Trigger point, plus direction, 1st measuring axis in the plane.
SD54603 \$SNS_MEA_WP_TRIG_MINUS_DIR_AX2[n]	Trigger point, minus direction, 2nd measuring axis in the plane.
SD54604 \$SNS_MEA_WP_TRIG_PLUS_DIR_AX2[n]	Trigger point, plus direction, 2nd measuring axis in the plane.
SD54605 \$SNS_MEA_WP_TRIG_MINUS_DIR_AX3[n]	Trigger point, minus direction, 3rd measuring axis in the tool direction.

9.10 Measuring cycles and measurement functions

SD54606 \$SNS_MEA_WP_TRIG_PLUS_DIR_AX3[n]	Trigger point, plus direction, 3rd measuring axis opposite to the tool direction. In the default case = 0.
SD54607 \$SNS_MEA_WP_POS_DEV_AX1[n]	Position deviation, 1st measuring axis in the plane.
SD54608 \$SNS_MEA_WP_POS_DEV_AX2[n]	Position deviation, 2nd measuring axis in the plane.
SD54609 \$SNS_MEA_WP_STATUS_RT	Calibration status of the axis positions
SD54610 \$SNS_MEA_WP_STATUS_GEN[n]	Calibration status
SD54611 \$SNS_MEA_WP_FEED[n]	Measuring feedrate when calibrating

9.10.4.3 Measuring tools at the milling machines

Settings

SD54762 \$SNS_MEA_FUNCTION_MASK_TOOL	Setting for the input screen, measuring cycles in the automatic mode, tool measurement
Bit 3	Setting for the input screen, measuring cycles in the automatic mode, tool measurement, enable tool probe calibration data field to be selected
Bit 4	Select calibration input measuring feed (VMS) ¹⁾
Bit 5	Select input feedrate and spindle speeds when probing
Bit 7	Select measurement in MCS and WCS
Bit 8	Select measurement, absolute and incremental
Bit 9	Select tool offset, geometry and wear
Bit 10	Select individual cutting edge measurement
Bit 11	Selects spindle reversal when calibrating in the plane
Bit 12	Select number of measurements (_NMSP)
Bit 13	Select experience values (_EVNUM)

¹⁾ Input, measurement feedrate is valid for AUTOMATIC and JOG

SD55634 \$SCS_MEA_FEED_PLANE_VALUE	Feedrate of the intermediate positioning in the working plane with active collision monitoring.
= 1000	Default value

SD55636 \$SCS_MEA_FEED_FEEDAX_VALUE	Feedrate of the intermediate positioning in the infeed axis with active collision monitoring.
= 1000	Default value

Calibration data of the tool probe, referred to the machine coordinate system

Before calibration is started, the position of the tool probe in the machine coordinate system (MCS) must be entered into the following general cycle setting data. In this case, the reference point is the outer diameter or the tool length of the active tool in the spindle. If there is no tool in the spindle, the reference points are the spindle center point and the tool reference point at the spindle.

Note

Calibrate probe

If you have calibrated the tool probe in JOG mode, then the calibration data has already been correctly entered in: SD54632 \$SNS_MEA_TP_AX_DIR_AUTO_CAL[k]

You do not need to recalibrate the tool probe in the AUTOMATIC mode.

Index [k] stands for the number of the actual data field (probe number -1).

SD54625 \$SNS_MEA_TP_TRIG_MINUS_DIR_AX1[k]	Trigger point of the 1st measuring axis in the negative direction.
SD54626 \$SNS_MEA_TP_TRIG_PLUS_DIR_AX1[k]	Trigger point of the 1st measuring axis in the positive direction.
SD54627 \$SNS_MEA_TP_TRIG_MINUS_DIR_AX2[k]	Trigger point of the 2nd measuring axis in the negative direction.
SD54628 \$SNS_MEA_TP_TRIG_PLUS_DIR_AX2[k]	Trigger point of the 2nd measuring axis in the positive direction.
SD54629 \$SNS_MEA_TP_TRIG_MINUS_DIR_AX3[k]	Trigger point of the 3rd measuring axis in the negative direction.
SD54630 \$SNS_MEA_TP_TRIG_PLUS_DIR_AX3[k]	Trigger point of the 3rd measuring axis in the positive direction.
SD54631 \$SNS_MEA_TP_EDGE_DISK_SIZE[k]	Tool probe, edge length / disk diameter.
SD54632 \$SNS_MEA_TP_AX_DIR_AUTO_CAL[k]	Axes and directions for calibrating in AUTOMATIC mode.
SD54634 \$SNS_MEA_TP_CAL_MEASURE_DEPTH[k]	Distance between the upper edge of the tool probe and lower edge of the tool (calibration depth, measuring depth for milling radius).
SD54635 \$SNS_MEA_TPW_STATUS_GEN[k]	Calibration status
SD54636 \$SNS_MEA_TPW_FEED[k]	Measuring feedrate when calibrating

The general cycle setting data SD54632 \$SNS_MEA_TP_AX_DIR_AUTO_CAL, is used to define in which axes and directions it is possible to calibrate the tool probe.

Decimal place		
ONES		1st axis
	= 0	axis not possible
	= 1	only minus direction
	= 2	only plus direction
	= 3	both directions
TENS		2nd axis

Decimal place		
	= 0	axis not possible
	= 1	only minus direction
	= 2	only plus direction
	= 3	both directions
HUNDREDS	3rd axis	
	= 0	axis not possible
	= 1	only minus direction
	= 2	only plus direction
	= 3	both directions

Example

If the general cycle machine data SD54632 \$SNS_MEA_TP_AX_DIR_AUTO_CAL has the value 123, the tool probe is calibrated as follows in the G17 plane:

- X in both directions
- Y only in plus direction
- Z only in minus direction

SD54633 \$SNS_MEA_TP_TYPE[k]		Probe version
= 0	Compatibility (measuring cycles: cube, turning surface shows probe, milling surface shows disk)	
= 101	Disk in XY, working plane G17.	
= 201	Disk in ZX, working plane G18.	
= 301	Disk in YZ, working plane G19.	
= 2	Probe	
= 3	Cube	

Calibration data of the tool probe referred to the workpiece coordinate system

Before calibration is started, the position of the tool probe in the workpiece coordinate system (WCS) must be roughly entered into the following general cycle setting data. In this case, the reference point is the outer diameter or the tool length of the active tool in the spindle. If there is no tool in the spindle, the reference points are the spindle center point and the tool reference point at the spindle.

Note

When measuring tools, ensure that the data of the adjustable work offset or the basic reference always correspond to the data when calibrating (measuring in WCS!).

Always make measurements and calibrate with the same adjustable work offset.

SD54640 \$SNS_MEA_TPW_TRIG_MINUS_DIR_AX1[k]	Trigger point of the 1st measuring axis in the negative direction.
SD54641 \$SNS_MEA_TPW_TRIG_PLUS_DIR_AX1[k]	Trigger point of the 1st measuring axis in the positive direction.
SD54642 \$SNS_MEA_TPW_TRIG_MINUS_DIR_AX2[k]	Trigger point of the 2nd measuring axis in the negative direction.
SD54643 \$SNS_MEA_TPW_TRIG_PLUS_DIR_AX2[k]	Trigger point of the 2nd measuring axis in the positive direction.
SD54644 \$SNS_MEA_TPW_TRIG_MINUS_DIR_AX3[k]	Trigger point of the 3rd measuring axis in the negative direction.
SD54645 \$SNS_MEA_TPW_TRIG_PLUS_DIR_AX3[k]	Trigger point of the 3rd measuring axis in the positive direction.
SD54646 \$SNS_MEA_TPW_EDGE_DISK_SIZE[k]	Tool probe, edge length / disk diameter.
SD54647 \$SNS_MEA_TPW_AX_DIR_AUTO_CAL[k]	Automatic calibration of tool probe, enable axes/directions
SD54648 \$SNS_MEA_TPW_TYPE[k]	Probe version
= 0	Compatibility (measuring cycles: cube, turning surface shows probe, milling surface shows disk)
= 101	Disk in XY, working plane G17.
= 201	Disk in ZX, working plane G18.
= 301	Disk in YZ, working plane G19.
= 2	Probe
= 3	Cube
SD54649 \$SNS_MEA_TPW_CAL_MEASURE_DEPTH[k]	Distance between the upper edge of the tool probe and lower edge of the tool (calibration depth, measuring depth for milling radius).
SD54650 \$SNS_MEA_TPW_STATUS_GEN[k]	Calibration status
SD54651 \$SNS_MEA_TPW_FEED[k]	Measuring feedrate when calibrating

The following general cycle setting data SD54647 \$SNS_MEA_TPW_AX_DIR_AUTO_CAL is used to define in which axes and directions it is possible to calibrate a tool probe.

Decimal place		
ONES	1st axis	
	= 0	axis not possible
	= 1	only minus direction
	= 2	only plus direction
	= 3	both directions
TENS	2nd axis	
	= 0	axis not possible
	= 1	only minus direction
	= 2	only plus direction
	= 3	both directions
HUNDREDS	3rd axis	
	= 0	axis not possible
	= 1	only minus direction

Decimal place		
	= 2	only plus direction
	= 3	both directions

Example

If the general cycle machine data SD54647 \$SNS_MEA_TPW_AX_DIR_AUTO_CAL has the value 123, the tool probe is calibrated as follows in the G17 plane:

- X in both directions
- Y only in plus direction
- Z only in minus direction

Monitoring when measuring with a rotating spindle

SD54670 \$SNS_MEA_CM_MAX_PERI_SPEED[0]		Maximum permissible peripheral speed of the tool to be measured.
= 100	Default value	

SD54671 \$SNS_MEA_CM_MAX_REVOLUTIONS[0]		Maximum permissible tool speed of the tool to be measured. The speed is automatically reduced when exceeded.
= 1000	Default value	

SD54672 \$SNS_MEA_CM_MAX_FEEDRATE[0]		Maximum permissible feedrate to probe the tool to be measured at the probe.
= 20	Default value	

SD54673 \$SNS_MEA_CM_MIN_FEEDRATE[0]		Minimum feedrate for the first probing of the tool to be measured at the probe. This avoids excessively small feedrates for large tool radii.
= 1	Default value	

SD54674 \$SNS_MEA_CM_SPIND_ROT_DIR[0]		Spindle direction of rotation to measure tools.
4 = M4	Default value	

Note

If the spindle is already rotating when the measuring cycle is called, this direction of rotation remains independent of the setting of this data.

SD54675 \$SNS_MEA_CM_FEEDFACTOR_1[0]	Feedrate factor 1
= 10	Default value
= 0	Only single probing with the feedrate calculated by the cycle. However, as a minimum, the value from SD54673[0] \$SNS_MEA_CM_MIN_FEEDRATE.
= ≥1	First probing with feedrate. However, as a minimum with the value from SD54673[0] \$SNS_MEA_CM_MIN_FEEDRATE) SD54675[0] \$SNS_MEA_CM_FEEDFACTOR_1

SD54676 \$SNS_MEA_CM_FEEDFACTOR_2[0]	Feedrate factor 2
= 0	Second probing with the feedrate calculated by the cycle. This is only effective for SD54673 \$SNS_MEA_CM_FEEDFACTOR_1[0] > 0, default value.
= ≥1	Second probing with the calculated feedrate from SD54673 \$SNS_MEA_CM_MIN_FEEDRATE[0] feedrate factor 2. Third probing with the calculated feedrate.

Note

Feedrate factor 2 should be less than feedrate factor 1.

SD54677 \$SNS_MEA_CM_MEASURING_ACCURACY[0]	Specified measuring accuracy. The value of this parameter always refers to the last probing of the tool at the probe.
= 0.005	Default value

Measurement with rotating spindle: Measured value correction using correction tables

SD54691 \$SNS_MEA_T_PROBE_OFFSET	Activates the measuring result correction
= 0	No data, default value
= 1	Correction in the cycle. This is only effective if SD54690 \$SNS_MEA_T_PROBE_MANUFACTURER>0.
= 2	Correction using user-defined correction table

SD54689 \$SNS_MEA_T_PROBE_MANUFACTURER	Activate pre-configured compensation tables for several tool probe models (customer-specific).
= 0	No data, default value
= 1	TT130 (Heidenhain)
= 2	TS27R (Renishaw)

Correction values for users

If the general cycle setting data SD54691 \$SNS_MEA_T_PROBE_OFFSET= 2, the following settings apply:

SD54695 to SD54700	Correction values for radius measurement.	See the subsequent general cycle setting data.
SD54705 to SD54710	Correction values for length measurement.	See the subsequent general cycle setting data.

SD54695 \$SNS_MEA_RESULT_OFFSET_TAB_RAD1[n]	Radius measurement
SD54705 \$SNS_MEA_RESULT_OFFSET_TAB_LEN1[n]	Length measurement
= 0	0
= 1	1st radius
= 2	2nd radius
= 3	3rd radius
= 4	4th radius

9.10.4.4 Measuring workpieces at the turning machines**General settings for turning**

MD52740 \$MNS_MEA_FUNCTION_MASK	Measuring cycles function screen
Bit 1	Workpiece measurement, measuring in the third geometry axis (Y, when turning
= 0	The measuring cycles do not support a third geometry axis (Y axis)!
= 1	Setpoint input and parameterization (SETVAL, _TUL, _TLL, SZO) refer to the third geometry axis (Y axis). The correction of the tool length or zero offset is however realized in the second geometry axis (X axis, ordinate) active components (i.e. measure in Y and correct in X). The correction target can be influenced using parameter _KNUM!
Bit 16	Measuring input, tool probe

Calibration data of the tool probe, referred to the machine coordinate system

Before calibration is started, the position of the tool probe in the machine coordinate system (MCS) must be entered into the following general cycle setting data.

SD54615 \$SNS_MEA_CAL_EDGE_BASE_AX1[n]	Calibration slot base referred to the 1st measuring axis.
SD54617 \$SNS_MEA_CAL_EDGE_PLUS_DIR_AX1[n]	Calibration slot edge in the positive direction of the 1st measuring axis.
SD54618 \$SNS_MEA_CAL_EDGE_MINUS_DIR_AX1[n]	Calibration slot edge in the negative direction of the 1st measuring axis.
SD54619 \$SNS_MEA_CAL_EDGE_BASE_AX2[n]	Calibration slot base referred to the 2nd measuring axis.
SD54620 \$SNS_MEA_CAL_EDGE_UPPERE_AX2[n]	Upper calibration slot edge referred to the 2nd measuring axis.

SD54621 \$SNS_MEA_CAL_EDGE_PLUS_DIR_AX2[n]	Calibration slot edge in the positive direction of the 2nd measuring axis.
SD54622 \$SNS_MEA_CAL_EDGE_MINUS_DIR_AX2[n]	Calibration slot edge in the negative direction of the 2nd measuring axis.

Note

For a standard lathe with axes X and Z (G18), axis Z is the 1st measuring axis and axis X is the 2nd measuring axis.

9.10.4.5 Measuring tools at the turning machines

Calibration data of the tool probe referred to the machine coordinate system

If you wish to calibrate the tool probe in the machine coordinate system, then the position of the tool probe in the machine coordinate system must be entered into the following general cycle setting data.

SD54625 \$SNS_MEA_TP_TRIG_MINUS_DIR_AX1[k]	Trigger point in minus direction of the 1st measuring axis (for G18 Z)
SD54626 \$SNS_MEA_TP_TRIG_PLUS_DIR_AX1[k]	Trigger point in plus direction of the 1st measuring axis (for G18 Z)
SD54627 \$SNS_MEA_TP_TRIG_MINUS_DIR_AX2[k]	Trigger point in minus direction of the 2nd measuring axis (for G18 X).
SD54628 \$SNS_MEA_TP_TRIG_PLUS_DIR_AX2[k]	Trigger point in plus direction of the 2nd measuring axis (for G18 X).

Calibration data of the tool probe referred to the workpiece coordinate system

If you wish to calibrate the tool probe in the workpiece coordinate system, then the position of the tool probe in the workpiece coordinate system must be entered into the following general cycle setting data. In this case, the reference point is the outer diameter or the tool length of the active tool in the spindle.

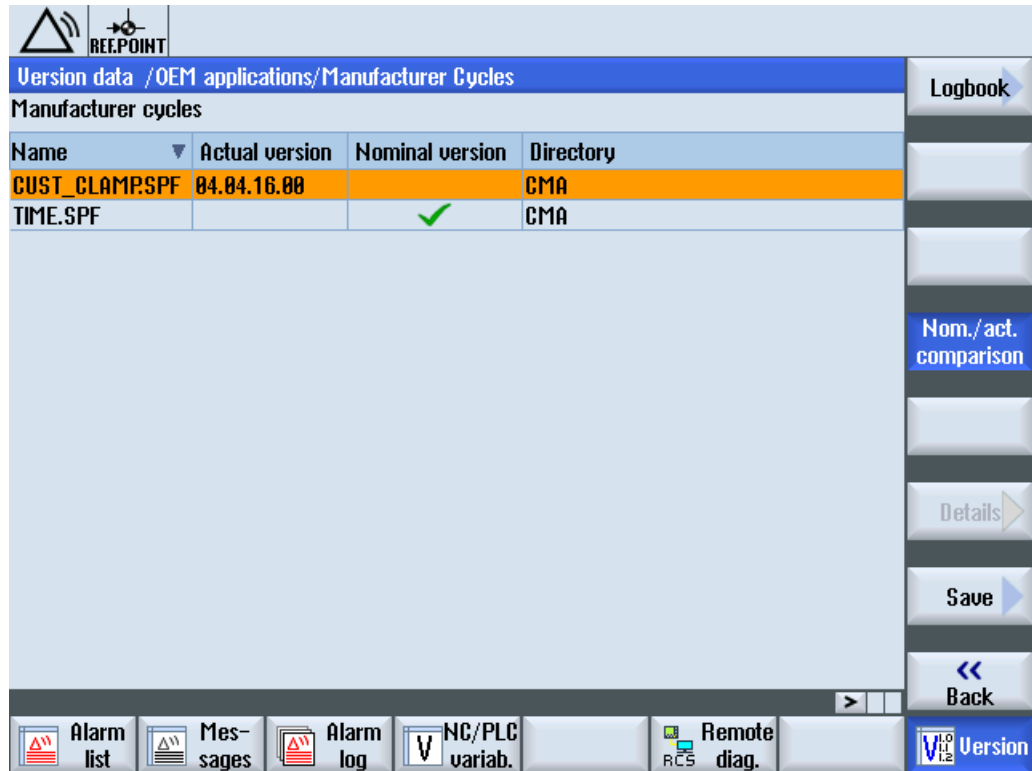
Index [k] stands for the number of the actual data field (probe number -1).

SD54640 \$SNS_MEA_TPW_TRIG_MINUS_DIR_AX1[k]	Trigger point minus direction of the 1st measuring axis (for G18 Z).
SD54641 \$SNS_MEA_TPW_TRIG_PLUS_DIR_AX1[k]	Trigger point plus direction of the 1st measuring axis (for G18 Z).
SD54642 \$SNS_MEA_TPW_TRIG_MINUS_DIR_AX2[k]	Trigger point minus direction of the 2nd measuring axis (for G18 X).
SD54643 \$SNS_MEA_TPW_TRIG_PLUS_DIR_AX2[k]	Trigger point plus direction of the 2nd measuring axis (for G18 X).

9.11 Compare cycles version

9.11.1 Display cycles version

If cycles have version details, you can display them in the version screen.



Precondition

The version details are included in the cycles files in the following form:

```
;VERSION: <Version> ;DATE: <YYYY-MM-DD>
```

Example:

```
;VERSION: 05.05.05.00 ;DATE: 2012-11-30
```

Procedure



1. Select the "Diagnostics" operating area.



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.



3. Select the "OEM applications /manufacturer cycles" areas and press the "Details" softkey.

The "Manufacturer Cycles" window opens.

The "Planned-actual comparison" softkey is selected by default.

9.11.2 Specify cycles version

General

You have the possibility to compare a required cycles version with the existing cycles version in the version view.

Name	Actual version	Nominal version	Directory
CUST_CLAMP:SPF	04.04.16.00	05.05.05.00	CMA
TIME.SPF		05.05.05.00	CMA

File overview

The following files are required:

Name	Path	Meaning
versions.xml	/siemens/sinumerik/hmi/data/version/oem/cma	Version file
versions.xml	/oem/	Version path file

Adapting the version file

The cycles version specification requires a manual adaptation of the version file.

1. Copy the "versions.xml" sample file from the following directory:
/siemens/sinumerik/hmi/data/version/oem/cma
2. Save the sample file in its own directory under /oem.
The standard directory for this purpose is /oem/sinumerik/hmi/data/version/oem/cma.
3. Open the file.
4. Enter the planned version for the cycles files in the <info> tag with "defaultFileVersion" and the details for the complete version in the <Version> tag.
5. Close the file to save the changes.

Example:

```
<info defaultFileType=" *.spf *.cpf" defaultFileVersion="05.05.00.00"
linkname="CMA" linkpath="//NC/CMA.DIR" EffDirOrder="CUS CMA">
<Name>Manufacturer cycles</Name>
<Version>5.5.0.0</Version>
<Link>
<Name>CUS</Name>
<Path>//NC/CUS.DIR</Path>
</Link>
</info>
```

Adapting the version path file

If the version file is saved in the standard directory, and a current oem\versions.xml is available, no changes are required. In all other cases, the path data can be manually changed:

1. Open the versions.xml file under /oem
2. Replace the standard path in the <Path> tag for the "Manufacturers Cycles" with your own path for the version file.
3. Close the file to save the changes.

Standard:

```
<Component>
<Name>Manufacturer Cycles</Name>
<Path>/siemens/sinumerik/hmi/data/version/oem/cma</Path>
```

```
</Component>
```

Example:

```
<Component>  
<Name>Manufacturer Cycles</Name>  
<Path>/oem/sinumerik/hmi/data/version/oem/cma</Path>  
</Component>
```


Spindle function

10.1 Spindle control

You can select from the following spindle control settings:

End of program

The following M functions are used for the end of program:

M function from MD10714 \$MN_M_NO_FCT_EOP	Main program end	Spindle continues to run
M2, M30	End of main program and return jump to beginning of program	Spindle stops
M17	End of subprogram and return jump into the main program	Spindle continues to run

If machine data MD10714 \$MN_M_NO_FCT_EOP is not equal to zero, then a distinction is made between M2 / M30 (program end of an NC program) and the M function set in MD10714 \$MN_M_NO_FCT_EOP.

Example

MD10714 \$MN_M_NO_FCT_EOP = 32 means "M32" for the program end of a program generated in the "JOG" or "MDA" operating mode.

Among other things, this functionality is required in order to continuously start the spindle in manual operation (e.g. for scratching).

Configuring keys

If you implement a manual control using the keys on the machine control panel, then this is realized via the following interface signals in the spindle data block:

DB3800.DBX5006.0	Spindle stop
DB3800.DBX5006.1	Spindle start clockwise rotation
DB3800.DBX5006.2	Spindle start, counter-clockwise rotation

The spindle can be started and stopped in the following state:

DB3300.DBX3.7 = 1	Channel in reset state
DB3300.DBX3.6 = 1	Channel state interrupted
DB3300.DBX3.3 = 1	Program state interrupted

Note

If the spindle must be stopped when the program is running, then set the interface signal DB380x.DBX4.3, "Feed stop/spindle stop" in the user PLC.

Additional settings:

MD11450 \$MN_SEARCH_RUN_MODE	Search run parameterization
Bit 1 = 1	Automatic ASUP start (PROG_EVENT) after output of action sets. Alarm 10208 is not output until the ASUB is completed. Notice: The PROG_EVENT cycle cannot be changed for the SINUMERIK 828D. Manufacturer-specific entries are possible only in the CYCPE_MA and CYCPE_US cycles.

Additional information

A detailed description on how to configure the spindle, is provided in the Axes and Spindles Function Manual

Direction of rotation (only for ShopTurn)

In the ShopTurn user interface, ensure that the direction of rotation of the spindle and C axis is correctly displayed and when programming ShopTurn functions, that the correct direction of rotation is executed. You must base these settings on the actual direction of rotation of the spindle/C axis on the machine.

- If you use the functions cylinder surface transformation and face machining, then a precondition is that the function was correctly commissioned.
See Cylinder surface transformation (TRACYL_K/TRACYL) (Page 180)
- The direction of rotation of the spindle / C axis for the M function M3 that is displayed in the ShopTurn interface is defined in part using machine data:
See: Defining the direction of rotation in Chapter: Setting-up ShopTurn cycles for turning (Page 167)
- The direction of spindle rotation (M3 / M4) is assigned to the positive rotational direction of the C axis via interface signal DB38xx.DBX2001.6. The bit defines whether M3 and C+ rotate in the same direction (= 0) or in opposite directions (= 1). The corresponding setting options can be found in the following sections:
 - Direction of rotation of counter-spindle (Setting up the counterspindle under ShopTurn (Page 172))
 - Direction of rotation of main spindle (Setting-up ShopTurn cycles for turning (Page 167))

10.2 Spindle diagnostics

10.2.1 Spindle diagnostics



Software option

For the "Spindle diagnostics" you require the "S-Monitor" option.

Main spindle drives are monitored via various status signals. The DRIVE-CLiQ connection at the spindle can be used to evaluate these signals in the drive. The status signals are evaluated for more effective diagnostics of the spindle operating states. The following information is displayed on the user interface with the aid of the "S-Monitor" software option:

- Operating hours
- Temperatures
- Speed/torque
- Clamping system
- Logistics data

Requirements

- A spindle with DQI encoder: r0459, bit 14 = 1.
- Drive telegram 139 is configured for the spindle.
- The spindle functionality for the machine axis is present when
MD35000 \$MA_SPIND_ASSIGN_TO_MACHAX > 0
The value corresponds to the spindle number.

More information

More information on the spindle with SMI 24 (Weiss spindle) is provided in the
Axes and Spindles Function Manual

Data overview

The "S1 Spindle Diagnostics" window shows the following information:

Parameter	Value
Operating hours • Spindle under control • Spindle under speed	h
Number of clamping cycles (tool change) Definition of a clamping cycle: Released clamping state [3] → Clamped clamping state [7, 8 or 10] → Released clamping state [3].	
Motor temperature	°C

Parameter	Value
Clamping state (sensor 1)	V
Piston free (sensor 4)	
e.g. defined rotary angular position of the shaft (sensor 5)	Yes/No
Maximum speed	rpm

Procedure

1. Select the "Diagnostics" operating area.
2. Press the menu forward key.
3. Press the "Spindle diagnostics" softkey to obtain a data overview.
The "S1 Spindle Diagnostics" window opens.

Note

If several spindles are in operation, select the desired spindle using the "Spindle +" or "Spindle -" softkey.

See also

Tool change with hybrid spindle from the company Weiss (<https://support.industry.siemens.com/cs/ww/en/view/109475707>)

10.2.2 Temperatures

Which temperatures are evaluated and displayed depends on the number and the mounting location (e.g. for sensor S6, bearing temperature front) of the sensors installed in the spindle. The motor temperature is not displayed when the temperature sensor sensor type parameter p0601 = 0 or 1.

Note**Temperature limits**

The limit values displayed here refer to the spindle monitoring and have no effect on the drive parameter values set in the drive that apply for the shutdown response of the spindle. The temperature limit values are preset in the SMI24 and cannot be changed in this window. If necessary, you must adapt the shutdown limit values for the drive.

Up to three temperatures can be evaluated:

Signal	Sensor	Column/meaning	Description
Motor temperature (Page 276)	KTY	Actual	Current motor temperature. If an overtemperature protection is present and responds, the value 250° C is displayed.
Motor temperature Alarm		Limit	Stored threshold for motor overtemperature as of which an alarm is issued due to KTY motor overtemperature.
		Duration of the upper limit violation	Displays the accumulated hours during which an upper limit violation was present.
		Last upper limit violation	Displays the time stamp [date] and [time] at which the last upper limit violation was present.
		Number of upper limit violations	Accumulated number of upper limit violations.
Motor temperature Fault		Limit	Stored threshold for motor overtemperature as of which a fault is issued due to KTY motor overtemperature.
		Duration of the upper limit violation	Displays the accumulated hours during which an upper limit violation was present.
		Last upper limit violation	Displays the time stamp [date] and [time] at which the last upper limit violation was present.
		Number of upper limit violations	Accumulated number of upper limit violations.
Overtemperature protection	PTC	Actual	When the overtemperature protection responds, "Active" is displayed (in red). The value 250° C is displayed in the "Actual" column as motor temperature.
		Limit	Stored threshold for motor overtemperature as of which a fault is issued due to PTC winding motor overtemperature.
		Last upper limit violation	Displays the time stamp [date] and [time] at which the last upper limit violation was present.
		Number of upper limit violations	Accumulated number of upper limit violations.
Additional temperature (Page 277) (e.g. bearing temperature front)	S6 (KTY)	Actual	Current temperature at the measuring point.

Signal	Sensor	Column/meaning	Description
Additional temperature Alarm		Limit	Stored temperature threshold as of which an alarm is issued, e.g. due to bearing overtemperature.
		Duration of the upper limit violation	Displays the accumulated hours during which an upper limit violation was present.
		Last upper limit violation	Displays the time stamp [date] and [time] at which the last upper limit violation was present.
		Number of upper limit violations	Accumulated number of upper limit violations.
Additional temperature Fault		Limit	Stored temperature threshold as of which a fault is issued, e.g. due to bearing overtemperature.
		Duration of the upper limit violation	Displays the accumulated hours during which an upper limit violation was present.
		Last upper limit violation	Displays the time stamp [date] and [time] at which the last upper limit violation was present.
		Number of upper limit violations	Accumulated number of upper limit violations.

10.2.3 Motor temperature sensor

Shutdown response

The alarm and fault thresholds of the motor temperature sensor that are displayed on the user interface as limits, are stored in the SMI24. They are used for the statistical evaluation of the SMI24 and for the shutdown response of the drive. A delay time setting for the violation of the alarm threshold stored in the SMI24 until the fault is triggered is not relevant for the display.

Please note: The motor parameter values of p0604, p0605 (alarm/fault threshold) and the delay time p0606 are predefined and cannot be changed.

The delay time in p0606 has no effect on the values displayed on the user interface. This means: When the alarm threshold is violated and the delay time set in p0606 has expired, the "Motor temperature fault" signal does not change the "Last upper limit violation" display value and the "Number of upper limit violations" is not incremented.

The parameter values of p0604, p0605 (alarm/fault threshold) and p0606 (delay time) have an effect on the shutdown response of the spindle. The spindle is shut down when the fault threshold is violated or the alarm threshold is violated and the delay time set in p0603 has expired.

Currently, the delay time is not predefined by the SMI24. The parameter value of p0603 is set to the factory setting (240 s).

Alarms:

- 207015 = motor temperature sensor alarm
- 207016 = motor temperature sensor fault

10.2.4 Additional temperature sensor

Shutdown response

The alarm and fault thresholds of the additional temperature sensor that are displayed on the user interface as limits, are stored in the SMI24. They are used only for the statistical evaluation of the SMI24. A delay time setting for the violation of the alarm threshold stored in the SMI24 until the fault is triggered is not relevant for the display.

Please note: The parameter values of p4102[0..1] (alarm/fault threshold) and p4103 (delay time) are predefined and cannot be changed.

Changing the parameter values of p4102[0..1] (alarm/fault threshold) and p4103 (delay time) has no effect on the values displayed on the user interface.

The parameter values of p4102[0..1] (alarm/fault threshold) and p4103 (delay time) have an effect on the shutdown response of the spindle. The spindle is shut down when the fault threshold is violated or the alarm threshold is violated and the delay time set in p4103 has expired. These parameters can be adapted accordingly to prevent an unwanted shutdown of the spindle.

Currently, the delay time is not predefined by the SMI24. The parameter value of p4103 is set to the factory setting (0 s). This means: The timer is deactivated.

Alarms:

- 207017 = additional temperature alarm threshold violated
- 207018 = additional temperature fault threshold violated

10.2.5 Temperature histograms

The motor temperature and optionally additional temperatures, such as bearing temperature or housing temperature, are displayed in a histogram. The color coding indicates the following temperature ranges:

Color	Meaning
	Temperature range up to the first limit.
	Temperature range between the first and second limit.
	Temperature range above the second limit.

The heights of the individual bar fields correspond to the parameterized temperature ranges of the relevant temperature measuring point.

The specified times for the individual temperature ranges of a measuring point (sensor) are the operating hours under speed.

The currently measured temperatures are displayed below the bars.

10.2.6 Speed/torque

To check the loads on the spindle, the speed and the torque are recorded in a histogram as a function of the operating hours. The color coding indicates the following states:

- Light green: low load
- Green: rated load
- Yellow: high load
- Orange: critical load

The heights of the individual bars correspond to the percentage time shares (dwell time) in relation to the operating hours under speed. The accumulated number of operating hours under speed for the spindle are shown above or below the respective bar for the corresponding speed or torque range. The width of the bar corresponds to the evaluated speed or torque range.

Possible conclusions

- The speed histogram is an indication of the load on the bearings and rotary gland caused by the speed.
- The speed histogram is an indication of the load on the bearings caused by the stock removal forces. The relationship between torque and radial force changes depending on the tool diameter (lever arm), the milling arc (superimposition of the cutting forces of the individual cutting edges) and the cutting force coefficients (the stock removal force consists of cutting force, feedrate force and passive force - only the cutting force is acquired via the torque).

10.2.7 Clamping system

The diagnosis of the clamping system offers the following capabilities:

- Fast and robust detection of the clamping states
- Monitoring of the clamping task and the clamping system
- Sensors S1 and S4 in the spindle provide information about the clamping device.
- Sensor S5 detects a position, e.g. the defined rotatory angular position of the shaft.

Clamping states

The preset limits for the three clamping states are displayed in the upper section of the window.

The clamping state is defined for the specified voltage range of the analog output signal of sensor S1.

The values in the "Voltage min." and "Voltage max." columns indicate in which voltage range the analog signal of sensor S1 must lie for the corresponding clamping state.

The preset speed limitation "Speed max." specifies the maximum permissible speed for the clamping state and is limited to this by the drive.

Signal overview

The current state values of the sensors, the tool clamping, the speed setpoint and the active speed limitation are displayed in the signal overview.

Signal	Sensor	Description	Value	Meaning
State	S1	Clamping state	3	Released
			7 or 8	Clamped with tool
			10	Clamped without tool
Piston free	S4	Plunger sensing	Yes	Release plunger not in contact with the shaft.
			No	Release plunger in contact with the shaft.
Shaft in the change position	S5	Position sensing	Yes	For example, the shaft is located at a defined rotary angular position.
			No	For example, the shaft is not located at a defined rotary angular position.
Speed set-point		Specification of the speed		Speed specification by the user.
State		Clamping states	0	Sensor S1 not available or state values inactive.
			1	The state initialization is in progress.
			2	Released with signal (error state)
			3	Released
			4	Clamping with tool
			5	Releasing with tool
			6	Releasing without tool
			7	Clamped with tool and S4 = 0 (No)
			8	Clamped with tool and S4 = 1 (Yes)
			9	Clamping without tool
			10	Clamped without tool
			11	Clamped with signal (error state)

State bar diagram

The clamping state is displayed by a state bar. The ranges shown in green indicate the state ranges for "Clamped with or without tool" and "Tool released". The arrow marking indicates the current analog sensor voltage of sensor S1.

More information

For information on the associated interface signals (NC → PLC), refer to the following documentation:

PLC Function Manual.

Procedure

1. The "S1 Spindle Diagnostics" window is open.
The "S1 Clamping System" window is open without the "S-Monitor" software option.
2. Press the "Clamping system" softkey.
The "S1 Clamping System" window opens and displays the acquired data.
3. Press the "Statistics" softkey to read out the clamping times and perform a diagnosis of the clamping system.
The "S1 Clamping System Statistics" window opens and displays the acquired data.
- OR -
Press the "Back" softkey to return to the data overview.

10.2.8 Clamping system: Speed limits

For safety reasons and to protect the spindle, only maximum speeds are permitted for certain clamping states. The drive limits the spindle speed in the respective clamping state to the entered speed.

Clamping state	Meaning
Released	The tool clamping system is in contact with the release plunger. Please note: The release plunger presses with the release force on the clamping system.
Clamping	The tool clamping system is in the transition state from "Released" to "Clamped with tool".
Releasing from state "Clamped with tool"	The tool clamping system is in the transition state from "Clamped with tool" to "Released".
Releasing from state "Clamped without tool"	The tool clamping system is in the transition state from "Clamped without tool" to "Released".
Clamped with tool	The tool clamping system is in the "Clamped with tool" state.
Clamping without tool	The tool clamping system is in the transition state from "Clamped with tool" to "Clamped without tool".
Clamped without tool	The tool clamping system is in the "Clamped without tool" state.

You can adapt the preset speed limits to your requirements by pressing the "Change" softkey.

NOTICE**Damage to the spindle through too high a speed**

Changes to the preset speed limits can result in changes of the operating states and to damage on the spindle.

Procedure

1. The "S1 Spindle Diagnostics" window is open.
The "S1 Clamping System" window is open without the "S-Monitor" software option.
2. Press the "Clamping system" softkey.
The "S1 Clamping System" window opens and displays the acquired data.
3. Press the "Speed limitation" softkey to change the limit values for the speeds of the clamping states.
The "S1 Speed Limitations" window opens.
4. Press the "Change" softkey to enter the desired speed limits depending on the clamping state.
Confirm with "OK".

10.2.9 Clamping system: Diagnostic statistics

This window displays the statistics for clamping operations and speed violations.

Statistical data	Meaning
Number of correct clamping cycles	Accumulated number of correct clamping cycles with or without tool. Correct clamping cycle means: "Released" clamping state [3] → "Clamped" clamping state [7, 8 or 10] → "Released" clamping state [3]. A clamping cycle is considered to be correct when a clamping and subsequent release operation are performed via the appropriate clamping states (without "State return"). Starting from one clamping state, the next clamping state that can be reached must not be the previous state. There is no dependency on time.
Clamping time (without tool)	The time within a clamping operation between the clamping states "Tool released" and "Clamped without tool".
• Within tolerance	Accumulated number of clamping operations without tool that were within the specified time.
• Outside tolerance	Accumulated number of clamping operations without tool that were outside the specified time.
• Specification	Factory specification of the time for the clamping operation without tool.
• Current	Measured time for the last performed clamping operation without tool.
Clamped (without tool)	Diagnostics during the "Clamped without tool" clamping state.
• Speed violations	Accumulated number of events during which the spindle speed was greater than the set limit as delivered.

Statistical data	Meaning
Speed threshold	Limit for the spindle speed in the "Clamped without tool" clamping state (as-delivered value). Changing this speed limit via the diagnostics overview "S1 clamping system" → "Speed limits" → "Change" has no effect on the value specified here.
Faulty clamping/releasing operations (total)	Accumulated number of clamping operations that have not run through the clamping states "Released" clamping state [3] → "Clamped" clamping state [7, 8 or 10] → "Released" clamping state [3] without interruption. A clamping cycle is considered to be faulty when, starting from one state, the next state that is reached is the previous state again ("State return").

10.2.10 Clamping system: Clamping time statistics

This window displays the statistics for the determined clamping times.

Statistical data	Meaning
Last determined clamping time	Measured time for the last performed clamping operation. Definition of a clamping operation: "Released" clamping state [3] → "Clamped with tool" clamping state [7 or 8]. Clamping state 3 = Released; clamping state 7 = Clamped with tool and S4 = 0 (No); clamping state 8 = Clamped with tool and S4 = 1 (Yes)
Reference clamping time (with tool)	During the spindle start-up phase, the first 100 clamping operations are measured by the SMI24 and the minimum value, maximum value and the mean value determined.
Minimum	The shortest clamping time determined from the first maximum 100 clamping operations.
Maximum	The longest clamping time determined from the first maximum 100 clamping operations.
Mean	The mean clamping time determined from the first maximum 100 clamping operations.
Mean clamping time (without tool)	After the first 100 clamping times determined for the reference time, the mean values for the other measured clamping times are calculated.
24 h	Mean clamping time of the clamping cycles determined in the last 24 hours.
10 days	Mean clamping time of the clamping cycles determined in the last 10 days.
100 days	Mean clamping time of the clamping cycles determined in the last 100 days.

If a Power On of the SMI24 is performed during the start-up phase, the previously determined values are discarded. The counter for the first 100 clamping operations is set to zero and the reference times up to the 100th clamping operation can be determined again. After the start-up phase (first 100 clamping operations), the values of the reference clamping time are "frozen". Even after a Power On of the SMI24, the values remain stored and can no longer be changed.

10.2.11 Fetching the logistics data

Requirement

The spindle characteristics can only be read out with the "Service" access level.

Spindle characteristics

The following spindle manufacturer characteristics can be read out via DRIVE-CLiQ:

- Manufacturer's identifier
- Drawing number
- Serial number
- Date of manufacture
- Service date and service information (1...4):
The service information corresponds to the reference number for the service report from the WEISS Spindeltechnologie GmbH company.

Service and diagnostics

11.1 NC/PLC variables

11.1.1 Displaying and editing PLC and NC variables

Changes can only be made to the NC/PLC variables with the appropriate password.

WARNING

Incorrect parameterization

Changes in the states of NC/PLC variables have a considerable influence on the machine. Incorrect configuration of the parameters can endanger life and cause damage to the machine.

In the "NC/PLC Variables" window, enter the NC system variables and PLC variables that you want to monitor or change in the list:

- 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 variables.
The columns can be displayed and hidden.
- Format
Specify the format in which the variable will 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) • Input bit (Ix), input byte (IBx), input word (IWx), input double word (IDx)
Outputs	• Output bit (Ax), output byte (ABx), output word (AWx), output double word (ADx) • Output bit (Qx), output byte (QBx), output word (QWx), output double word (QDx)
Bit memory	Memory bit (Mx), memory byte (MBx), memory word (MWx), memory double word (MDx)
Times	Time (Tx)

PLC variables	
Counters	- Counter (Cx) - Counter (Cx)
Data	- Data block (DBx): Data bit (DBXx), data byte (DBBx), data word (DBWx), data double word (DBDx) - Data block (VBx): Data bit (VBXx), data byte (VBBx), data word (VBWx), data double word (VBDx)

Formats	
B	Binary
H	Hexadecimal
D	Decimal without sign
+/-D	Decimal with sign
F	Floating point (for double words)
A	ASCII character

Notation examples

Permissible notation for variables:

- PLC variables: EB2, A1.2, DB2.DBW2, VB32000002
- NC variables:
 - NC system variables: Notation \$AA_IM[1]
 - User variables / GUD: Notation GUD/MyVariable[1,3]
 - OPI notation: /CHANNEL/PARAMETER/R[u1,2]

Note

If the PLC user program writes a string in an NC/PLC variable, the string will only be displayed correctly if the variable is parameterized as a field variable of the type "A" (ASCII) on the NC side.

Example of a field variable

Variable	Format
DBx.DBBy[<number>]	A

Inserting variables

The start value for "Filter/Search" of variables differs. For example, to insert the variable \$R[0], enter the following start value:

- 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.

The GUD from the machine data is only displayed in the Search window for the variable selection when the associated definition file has been activated. Alternatively, enter the variable you are searching for manually, e.g. GUD/SYG_RM[1]

The following machine data is representative for all variable types (INT, BOOL, AXIS, CHAR, STRING): MD18660 \$MN_MM_NUM_SYNACT_GUD_REAL[1].

Note

- System variables can be dependent on the channel. When the channel is switched over, the values from the selected channel are displayed.
You have the option of having the variable displayed for a specific channel, e.g. \$R1:CHAN1 and \$R1:CHAN2. The values of channel 1 and channel 2 are displayed, irrespective of the channel you are in.
- For user variables (GUD), it is not necessary to specify whether they are global or channel-specific GUD. The first element of a GUD array starts with index 0 as for NC variables.
- Use the tooltip to show the OPI notation for NC system variables (except for GUD).

Servo variables

Servo variables can only be selected and displayed at "Diagnostics" → "Trace".

Changing and deleting values

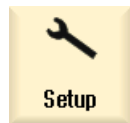


1. Select the "Diagnostics" operating area.



2. Press the "NC/PLC variables" softkey.

- OR -



1. Select the "Setup" operating area.



2. Press the "PLC" and "NC/PLC variab." softkeys.



The "NC/PLC Variables" window opens.

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



4. Press the <INPUT> key.
The operand is displayed with the value.



5. Press the "Details" softkey.
The "NC/PLC Variables: Details" window opens. The information for "Variable", "Comment" and "Value" is displayed in full length.



6. Position the cursor in the "Format" field and select the required format with <SELECT>.



7. Press the "Display comments" softkey.
The "Comments" column is displayed. You have the option of creating comments or editing existing comments.
Press the "Display comments" softkey once again to hide the column.



8. Press the "Change" softkey if you would like to edit the value.
The "Value" column can be edited.



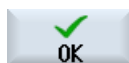
9. Press the "Insert variable" softkey if you wish to select a variable from a list of all existing variables and insert this.
The "Select Variable" window opens.



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.



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



12. Press the "OK" softkey to confirm the changes or the deletion.

- OR -



Press the "Cancel" softkey to cancel the changes.

Editing a variable list

You can edit the variable list using the "Insert line" and "Delete line" softkeys.



When you press the softkey, a new line is inserted before the line marked by the cursor.

You can only use the "Insert line" softkey if there is at least one empty line at the end of the variable list.

The softkey is deactivated if there is no empty line.



When you press the "Delete line" softkey, the line marked by the cursor is deleted.

An empty line will be added at the bottom of the variable list.

Changing operands

Depending on the type of operand, you increment or decrement the address or address index by one place at a time using the "Operand +" and "Operand -" softkeys.

Note

Axis name as index

If the axis name is the index, the "Operand +" and "Operand -" softkeys have no effect, e.g. \$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]

11.1.2 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. 
3. Press the "Save screen" softkey.  The "Save screen: Select archiving" window opens.
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.



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, then you will receive a prompt.



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.

11.2 Displaying the Service overview

The operating state of machine axes and drives is displayed in the "Service Overview" window. You configure the display via the "Extended selection >" softkey or directly via the drop-down list according to the following criteria:

- All NC axes and drives (= default)
- NC axes
- Drives without NC axis assignment

Enable displays

- On/Off1
- Off2
- Off3
- From the drive: Operation enabled
- From the infeed: Operation enabled
- Enable pulses
- Speed controller enable NC
- Pulses enabled
- Drive ready
- Heat sink temperature
- Power section in i²t limiting
- Motor temperature
- **Measuring system 1 active / measuring system 2 active**
The states for measuring system 1/2 have the following meaning:

Symbol	Meaning
	The position measuring system is active.
	The position measuring system is parked. - OR: - The position measuring system is not configured.
	The position measuring system is passive.

Procedure

1. Select the "Diagnostics" operating area.



2. Press the menu forward key and the "Axis diag" softkey.



3. Select an overview in the "Selection" field.

- OR -



Press the "Go to selection" softkey to select an overview.

11.2.1 Selecting axes and drives

To display certain enable signals and statuses of machine axes, you can arrange a selection of all the axes and drives that are available in any sequence you wish.

Note

The following selection configurations have been previously defined, and cannot be changed or deleted:

- All NC axes and drives
- NC axes
- Drives without NC axis assignment

Selection overview

The following information is provided in the selection overview.

Column	Description
Bus No.	Bus number for drives without NC assignment. No entry for NC axes.
Slave address	Slave address for drives without NC assignment. No entry for NC axes.
Device No.	Device number for drives without NC assignment. No entry for NC axes.
Drive object No.	Drive object number for drives without NC assignment. No entry for NC axes.

Column	Description
Name/identifier	For NC axes contains the standard machine axis identifier and the user-defined machine axis names. For drives without NC assignment, contains the drive object names.
Assignment	NC is displayed for an internal object assignment. PLC is displayed for an external object assignment.

Note

The selection overview differs in the information provided regarding the slave address and device number between PROFIBUS and PROFINET configuration.

Procedure

1. Select the "Diagnostics" operating area.



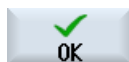
2. Press the menu forward key and the "Axis diag" softkey.
The "Service Overview" window opens.



3. Press the "Extended selection" softkey.
The window "Extended selection: Axes and drives" opens.



4. Press the "New selection" softkey.
The "New selection" dialog is opened.



5. Enter a display and a file name in the entry fields.
6. Confirm the entry with the "OK" softkey.
The "Edit selection" window is opened for the newly created selection.
7. Set checkmarks in the check boxes for the axes and drives that you wish to display.

- OR -

Press one of the following softkeys to obtain a certain selection:
All axes/drives are selected.



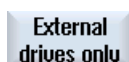
No axis/drive is selected.



Only internal drives are selected.



Only external drives are selected.



Those axes are selected that are assigned to a real drive.



- | | |
|---|--|
|  | All axes are selected, which are defined in a minimum of one channel. |
|  | 8. Press the "OK" softkey to confirm your selection. |
|  | 9. Press the "Delete selection" softkey to delete the actual selection. |
|  | 10. Press the "Edit selection" softkey to edit the actual selection.
The "Edit selection" window opens. |
|  | 11. If you click the "Back" softkey, you will return to the "Service Overview" main screen. |

11.2.2 Axis diagnostics

Application

The following information is displayed in the "Service Overview" window:

- Check of the setpoint branch (e.g. programmed position setpoint, speed setpoint, spindle speed setpoint)
- Check of the actual value branch (e.g. actual position value, measuring system 1 or 2, actual speed value)
- Optimization of the position control loop of the axis (e.g. following error, control deviation, servo gain factor)
- Check of the entire control loop of the axis (e.g. through position setpoint/actual-value comparison and speed setpoint/actual-value comparison)
- Check of hardware errors (e.g. by checking the encoder: If the axis is moved mechanically, the actual position value must change)
- Setting and check of the axis monitoring functions.

Procedure

- | | |
|---|---|
|  | 1. Select the "Diagnostics" operating area. |
|  | 2. Press the menu forward key and the "Axis diagnostics" softkey.
The "Service Overview" window opens. |
|  | |
|  | 3. Press the "Service axis" softkey.
The "Service Axis/Spindle" window opens. |

4. Press the "Axis +" or "Axis -" softkey to scroll the axes forward or backward.
- OR -
- Press the "Axis selection" softkey.
The "Axis Direct Selection" window opens.
Select the required axis directly from those available in the drop-down list box.
6. Confirm the selection with "OK".
The values of the axis are displayed.



Display data

Display data	Meaning
Following error	The difference between the position setpoint and the actual position value of the active measuring system 1 or 2. Unit: mm, inch or degrees
System deviation	The difference between the position setpoint at the position controller input and the actual position value of the active measuring system 1 or 2. Unit: mm, inch or degrees
Contour deviation (axial)	The actual contour deviation is displayed with this value (variations of the following error caused by equalization operations on the speed controller due to load changes). The contour deviation results from the difference between an actual position pre-calculated from the position setpoint and the actual position value of active measuring system 1 or 2. Unit: mm, inch or degrees
Servo gain factor (calculated)	The servo gain factor in the display is calculated by the NC according to the following equation: $K_v - \text{factor} = \frac{\text{Speed setpoint}}{\text{Following error}} ;$ Unit (for default setting): $\frac{[m/min]}{[mm]} ;$ Velocity setpoint = setpoint currently being output to the axis/spindle Further information: Function Manual Axes and Spindles
Active measuring system	0: No measuring system active. 1: Measuring system 1 active. 2: Measuring system 2 active.
State of measuring system 1 State of measuring system 2	The state of measuring system 1 or 2 is output here: Active - Parked - Passive

Display data	Meaning
Actual position value measuring system 1	The actual position of the axis measured via measuring system 1 or 2. The position is displayed in the machine coordinate system (no work offsets or tool offsets taken into account). Unit: mm, inch or degrees
Actual position value measuring system 2	
Position setpoint	Specified position transferred from the interpolator to the position control Unit: mm, inch or degrees
Absolute compensation value measuring system 1	Display of the absolute compensation value for measuring system 1 or 2. The compensation value consists of the sum of backlash and leadscrew error compensation for the actual axis position. Unit: mm, inch or degrees
Absolute compensation value measuring system 2	
Compensation, sag + temperature	Display of the compensation value calculated for the current axis position based on the total of the sag and temperature compensations. Unit: mm, inch or degrees
Actual speed value, active encoder	The pulses supplied by the encoder are evaluated by the NC and displayed. Unit: % 100% means maximum speed.
Speed setpoint, drive	Speed setpoint transferred to the drive (= speed setpoint from position controller and feedforward control). Unit: % 100% means maximum speed setpoint.
Programmed spindle speed setpoint	Speed setpoint programmed by the user. Unit: rpm e. g.: Input: S1000; display: 1000 rpm Display applies to spindles only.
Spindle speed setpoint current	Current active speed setpoint with correct sign, including calculated compensation value and any active speed limitation (specified by setting or machine data) Unit: rpm Display applies to spindles only.
Position offset to the leading axis / spindle actual value	The currently valid position offset value is displayed here (relative to the actual value) if a position offset (angular offset between the following and leading spindle) has been programmed within the synchronous spindle functionality. Unit: mm, inches, degrees Further information about the synchronous spindle: Function Manual Axes and Spindles
Position offset to the leading axis / spindle setpoint	The currently valid position offset value is displayed here (relative to the setpoint) if a position offset (angular offset between the following and leading spindle) has been programmed within the synchronous spindle functionality. Unit: mm, inches, degrees
Override	The effective correction factor of the feed or spindle correction switch is displayed. Unit: %

Display data	Meaning
Current gear stage	Display of the current actual gear stage With axes, this is only displayed if a spindle is assigned to the axis. The display corresponds to the NC/PLC interface signal: DB31, ... DBX16.0-2 (actual gear stage) Further information about the spindle: Function Manual Axes and Spindles
Parameter set (axis)	Displays which of the six parameter sets of the position controller is active. Further information: Function Manual Axes and Spindles
Controller mode	Display of the current controller state: • Position control • Speed control • Hold • Parking • Follow-up • Braking Further information: Function Manual Basic Functions
Feedforward control mode	Indicates whether and if so, which mode of the dynamic feedforward control for the axis is active: • Inactive • Velocity The velocity-dependent speed feedforward control is active. • Torque The acceleration-dependent torque feedforward control is active (in combination with the speed feedforward control) Further information on compensation: Function Manual Monitoring and Compensating
"Referenced" state	Status display for reference point approach (axis): • Curr. MS need not be referenced (active measuring system does not require referencing) • Curr. measuring system referenced • Curr. measuring system must be referenced (active measuring system requires referencing) The display depends on the settings in the machine data: • MD34110 \$MA_REFP_CYCLE_NR • MD20700 \$MC_REFP_NC_START_LOCK The display corresponds to the NC/PLC interface signal: DB31, ... DBX60.4 and 60.5 (referenced/synchronized 1 or 2) Further information on reference point approach: Function Manual Axes and Spindles

Display data	Meaning
QEC state	Indicates whether and if so, which method of the quadrant error compensation (QEC) is active for the axis: • Inactive • Neural QEC learning active • Conventional QEC active • Conventional QEC w. adapt. Corr. v. active (conventional QEC with adaptation of the compensation value active) • Neural QEC active • Neural QEC w. adapt. Measurement duration active (neural QEC active with adaptation of the measuring time active) • Neural QEC w. adapt. Decay time corr. v. active • (neural QEC with adaptation of decay time of compensation value active) • Neural QEC w. adapt. Meas. dur. + decay time Corr. v. active (neural QEC active with adaptation of measuring time and decay time of compensation value active) Further information on compensation: Function Manual Monitoring and Compensating
"Travel to fixed stop" state	Indicates whether or not the axis has fulfilled the conditions for "Fixed stop reached" when the "Travel to fixed stop" function is active (IS DB31, ... DBX62.5): • Normal control ("Travel to fixed stop" function not activated) • Fixed stop reached • Failed Further information on travel to fixed end stop: Function Manual Monitoring and Compensating
Torque limitation value	Indicates the value programmed via FXST[x] or SD43510 \$SA_FIXED_STOP_TORQUE or the value defined via MD37010 \$MA_FIXED_STOP_TORQUE_DEF for the clamping torque for "Travel to fixed stop". Unit: % of maximum torque

11.2.3 Service drive

Display drive and motor data

The "Service Drive" window displays important information about the status of the motors and drive modules, such as motor temperature and DC-link voltage.

Procedure



1. Select the "Diagnostics" operating area.



2. Press the menu forward key and the "Axis diag" softkey.
The "Service Overview" window opens.



3. Press the "Service drive" softkey.
The "Service Drive" window opens.



4. Press the "Drive +" or "Drive -" softkey to scroll forward or backward.



- OR -



Press the "Drive selection" softkey.
The "Drive direct selection:" window opens.
Select the desired drive directly from the selection list.



5. Confirm the selection with "OK."
The drive data is displayed.

Overview

The individual status displays, warnings, messages, etc. that are displayed in the "Service Drive" window are explained in the following.

Additional information

Additional information about interface signals is provided in:

- Basic Functions Function Manual
- NC Variables and Interface Signals List Manual

You can find information about switching over encoder data sets in the
Monitoring and Compensating Function Manual

You can find additional information about drive parameters (SERVO drive object) in the
List Manual SINAMICS S120/S150.

PLC pulse enable

The display, whether the pulse enable from the PLC is available for the drive corresponds to the interface signal:

DB380x, ... DBX4001.7 "Pulse enable".

Status	Meaning	Display
1	Pulse enable for this drive is activated by the PLC.	Yes
0	The pulses for the drive module are disabled from the PLC.	No

Speed controller enable NC

The display, whether the speed controller enable from the NC is available for the drive corresponds to the interface signal:

DB390x ... DBX1.6 "Speed controller active".

Status	Meaning	Display
1	Speed controller enable from the NC present.	Yes
0	Speed controller enable from the NC not present.	No

Rampup function generator quick stop

The status display for the ramp-function generator quick stop corresponds to the interface signal:

DB3900 ... DBX4000.1 "Ramp-function generator disable active".

Status	Meaning	Display
1	Ramp-up function generator quick stop is active. The drive is stopped without a ramp function with speed setpoint = 0 and without pulse suppression.	Yes
0	Ramp-up function generator quick stop is not active for the drive.	No

Pulses enabled

The message whether the pulses have been enabled for the drive corresponds to the interface signal:

DB390x, ... DBX4001.7 "Pulses enabled".

Status	Meaning	Display
1	The drive module pulses are enabled. The axis/spindle can now be traversed.	Yes
0	The drive module pulses are suppressed. The axis/spindle can therefore not be traversed.	No

Drive ready

The display of the current status of the selected drive corresponds to the interface signal: DB390x, ... DBX4001.5 "Drive ready".

Status	Meaning	Display
1	The drive is ready.	Yes
0	The drive is not ready.	No

Ramp-up phase

The display of the current ramp-up phase of the selected drive corresponds to the drive parameter:

r0002 "Drive operating display".

Number of faulty signs-of-life

Display of communications errors detected in hardware between NC and drive.

Note

If the display shows a value other than "0", please contact your Siemens regional office!

SC1 drive alarm message

Displays (yes/no) whether messages of status class 1 are active. Status class 1 messages are alarms with the following properties:

- They lead to internal responses (e.g. regenerative braking, immediate pulse suppression)
- They are modal.

This is a group message. For details on the actually pending drive alarms, please refer to the "Drive System Diagnostics" window, see Service drive (Page 298).

DC-link voltage, smoothed

The display of the smoothed actual value of the DC-link voltage of the selected drive corresponds to the drive parameter:

r0026 "DC-link voltage smoothed".

Unit: Volts

Speed setpoint, smoothed

The display of the smoothed speed setpoint - measured according to the setpoint limitation for the P component of the speed controller - corresponds to the drive parameter: r1438 "Speed controller speed setpoint".

Unit: rpm

Actual speed value

The display of the smoothed actual value of the motor speed corresponds to the drive parameter: r0021 "Actual speed value smoothed".

Unit: rpm

Actual current value, smoothed

The display of the smoothed actual current value corresponds to the drive parameter: r0078[1] "Actual current value, torque-generating".

Unit: A

Motor temperature

The display of the current temperature in the motor corresponds to the drive parameter: r0035 "Motor temperature"

Unit: °C

Integrator disabling

The display as to whether the integrator of the speed controller is active, corresponds to the interface signal:

DB390x, ... DBX4001.6 "n-controller integrator disabled".

Status	Meaning	Display
1	The requested shutdown of the speed controller integrator is active in the drive. The speed controller has been switched over from the PI to P control response.	Yes
0	The integrator of the speed controller is enabled. The speed controller functions as a PI controller.	No

Parking axis

Displays (yes/no) whether it is a parking axis/spindle.

Note

With parking axes/spindles, all encoder-specific monitoring and evaluation functions are switched off. This allows the encoder to be withdrawn without initiating an alarm.

Specified drive data set

The display (standard: DDS0) of which of the eight drive parameter sets is to be activated by the PLC corresponds to the interface signal:

DB380x, ... DBX4001.0 to 4001.2 "Drive parameter set selection A, B, C".

Actual drive data set

The display (standard: DDS0) of which of the eight drive parameter sets is currently active corresponds to the interface signal:
DB390x, ... DBX4001.0 to 4001.2 "Active drive parameter set A, B, C".

Specified motor data set

The display (MDS0...3) of which motor data set is to be activated by the PLC corresponds to the interface signal:
DB380x, ... DBX4001.3 to 4001.4 "Motor selection A, B".

The following assignment applies:

Motor data set	Coding	
MDS0	0	0
MDS1	0	1
MDS2	1	0
MDS3	1	1

Actual motor data set

The display (MDS0...3) of which motor data set is currently active corresponds to the interface signal:
DB390x, ... DBX4001.3 to 4001.4 "Active motor A, B".

Operating mode

The display of the control type of a drive corresponds to the drive parameter:
p1300[0...n] "Open-loop/closed-loop control mode".

Depending on "n", the following "values" are displayed:

n	Display
20	Speed control (without encoder)
21	Speed control (with encoder)
23	Torque control (with encoder)

Position actual value measuring system 1/2

The actual position of the axis measured via measuring system 1/2. The position is displayed in the machine coordinate system (no work offsets or tool offsets taken into account).

Unit: mm, inch or degrees

Heat sink temperature

The display of whether the heat sink temperature is OK corresponds to the interface signal: DB3390x, ... DBX4002.1 "Heat sink temperature prewarning".

Status	Meaning	Display
1	The drive signals a "Heat sink temperature prewarning" to the PLC.	Overtemperature
0	The drive module heat sink temperature pre-warning has not responded.	OK

Motor temperature

The display of whether the motor temperature is OK corresponds to the interface signal: DB390x, ... DBX4002.0 "Motor temperature prewarning".

Status	Meaning	Display
1	The motor temperature has exceeded the warning threshold configured in the drive.	Overtemperature
0	The motor temperature is below the warning threshold.	OK

Ramp-up function completed

The status display of the drive as to whether the ramp-up has been completed corresponds to the interface signal: DB390x, ... DBX4002.2 "Ramp-up completed".

Status	Meaning	Display
1	The actual speed value has reached the speed tolerance band specified via p2164 after a new speed setpoint specification and has not left the band for the duration of p2166.	Yes
0	The ramp-up procedure is still active after the speed setpoint has been changed.	No

Torque lower than threshold setting

The status display of the drive as to whether the threshold torque has been undershot corresponds to the interface signal: DB390x, ... DBX4002.3 " $|M_d| < M_{dx}$ ".

Status	Meaning	Display
1	The current torque utilization lies below the torque utilization threshold set in p2194. The drive signals the PLC that the torque setpoint $ M_d $ does not exceed the threshold torque M_{dx} .	Yes
0	The torque setpoint $ M_d $ is larger than the threshold torque M_{dx} . This signal can be used to determine whether the motor is overloaded.	No

Speed lower than minimum setting

The status display of the drive as to whether the minimum speed has been undershot corresponds to the interface signal:
DB390x, ... DBX4002.4 " $|n_{act}| < n_{min}$ ".

Status	Meaning	Display
1	The actual speed value $ n_{act} $ is less than the threshold minimum speed n_{min} set in p2161.	Yes
0	The actual speed value is greater than the threshold minimum speed.	No

Speed lower than threshold setting

The status display of the drive as to whether the threshold speed has been undershot corresponds to the interface signal:
DB390x, ... DBX4002.5 " $|n_{act}| < n_x$ ".

Status	Meaning	Display
1	The actual speed value $ n_{act} $ is less than the threshold speed n_x set in p2155.	Yes
0	The actual speed value is greater than the set threshold speed.	No

Actual speed = set speed

The status display of the drive as to whether the actual speed value matches the speed setpoint corresponds to the interface signal:
DB390x, ... DBX4002.6 " $|n_{act}| < n_{set}$ ".

Status	Meaning	Display
1	The speed deviation between the setpoint and actual value is within the tolerance specified in p2163.	Yes
0	The speed deviation between setpoint and actual value is outside the specified tolerance.	No

Setpoint/actual Encoder Data Set active encoder

Shows the required and the active encoder data set of the encoder.

Diagnostics for alarms

This information is also provided as a diagnostic tool for diagnosing the causes of alarms such as:

- Drive fault:
 - ⇒ **SC1 drive alarm message** is set.
 - ⇒ Check **specified drive data set, specified motor data set, DC-link voltage**.
- Alarm 25040 "Standstill monitoring",
Alarm 25050 "Contour monitoring",
Alarm 25060 "Speed setpoint limitation",
Alarm 25080 "Positioning monitoring"
 - ⇒ the drive enable may be missing (**PLC pulse enable** not present); this results in the display **Pulses enabled** = No.
- **Motor temperature** = exceeded
 - ⇒ check the current motor temperature.

Information about alarms and their rectification is provided in the online help.

Additional information

You can find information on how the NC control responds to individual alarms and the associated remedial action in the

Diagnostics Manual

11.3 PROFINET diagnostics

11.3.1 Displaying the PROFINET configuration

Display the PROFINET status for diagnostic purposes during the configuration or when errors occur. This diagnostics window is only intended for information purposes. You cannot make changes there.

PROFINET connections

- PN1
- PN2

The PPU variant PPU 24x.3 BASIC only has one PROFINET connection (PN).

Note

PROFIBUS DP connection

The SINUMERIK 828D only has one integrated PROFIBUS DP. The NC communicates with the drives via this integrated PROFIBUS DP. It cannot be accessed from outside on the PPU and is therefore not described further here.

Display	Meaning/information
Status	
Bus status	POWER_ON (0): Status after the control is switched on. OFFLINE (1): A basic initialization has been performed. STOP (2): Start in accordance with the hardware configuration (SDB). CLEAR (3): The PROFINET devices have been parameterized and configured in accordance with the hardware configuration (SDB) and taken into the cyclic data exchange with zero output data. OPERATE (4): The cyclic data exchange with the PROFINET devices is running. ERROR (9): A fatal error had been detected (e.g. invalid or faulty SDB). (<n>) Corresponds to the OPI variable value for the bus state.
Bus configuration	
S7 subnet ID	S7 subnet ID of the PROFINET subnet
Baud rate	Transmission rate in Mbd
Cycle time	Configured bus cycle time in ms; defines the position control cycle clock at the same time.
Sync. Component (TDX)	Configured interval in ms for the cyclic data exchange within a PROFINET cycle.
PROFIBUS diagnostics/devices	
Device name	Name of the PROFINET device.
Device no.	Number of the PROFINET device.

Display	Meaning/information
Assignment	Information as to whether the PROFINET device is assigned to the NC or to the PLC. NC: e.g. one or more drives controlled by the NC. PLC: e.g. I/Os or an axis controlled by the PLC, NC/PLC.
Communication state	Information as to whether the PROFINET device is identified on the bus via the communication state. ✔ Green: The PROFINET device has been detected on the PROFINET and the data exchange is working error-free with the assigned component (NC and/or PLC). ✘ Red: Incorrect response / no communication
Synchr. with NC	Information as to whether the PROFINET device is running synchronously to NC on the bus. ✔ Green: The PROFINET is running synchronously to the NC on the PROFINET, i.e. there is an isochronous data exchange. ✘ Red: Incorrect response / no communication ○ Gray: The PROFINET device is not assigned to the NC, but rather to the PLC.
Number of subslots	Number of configured subslots within the PROFINET device.

Additional information

Additional information about the network interface for PROFINET is provided in the:

- PPU and Components Equipment Manual
- Commissioning Manual Commissioning CNC

Procedure



1. Select the "Diagnostics" operating area.

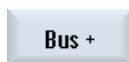


2. Press the menu forward key and the "Bus TCP/IP" softkey. The "PROFINET/PROFIBUS Diagnostics" window opens.



3. Press the "Bus -" or "Bus +" softkey to select the desired bus.

...



- OR -



4. Press the "Select bus" softkey and select the desired bus in the displayed list.



5. Press the "OK" softkey.

11.3.2 Displaying details for PROFINET devices

Further information about the subslots of a selected PROFINET device is displayed here.

Note

PROFIBUS DP connection

The SINUMERIK 828D only has one integrated PROFIBUS DP. The NC communicates with the drives via this integrated PROFIBUS DP. It cannot be accessed from outside on the PPU and is therefore not described further here.

Display	Meaning/information
Device	
Name	Name of the PROFINET device.
Number	Number of the PROFINET device.
IP address	IP address of the PROFINET device.
Assignment	Information as to whether the PROFINET device is assigned to the NC or to the PLC.
Communication state	Current communication state of the PROFINET connection to the device. Green: The communication is active. Red: The communication is not active.
Slots	
No.	Number of the slot.
Subslot no.	Number of the subslot within the device.
I/O addr.	I/O address in the I/O address space of the PLC assigned to this subslot. For NC axes, the setpoint and the actual value must always be configured on the same I/O address.
Type	Specification, whether the subslot is an input, output or diagnostics slot. If the subslot is assigned to an NC axis, then the output is always denoted as setpoint and the input always as actual value.
Length (bytes)	Length of the I/O area reserved for the subslot in the STEP7 I/O address space.
Communication state	Actual communication state of the subslot If a value is not available for the NC axes, then the display remains hidden. Green: The communication is active. Red: The communication is not active. Gray: No NC axis.

Display	Meaning/information
Machine axis	Display of the name defined in the machine data for this slot If the slot is not assigned to any NC axis, then the display remains hidden.
Telegr. type (NC)	Displays the assigned telegram type If a telegram type is not assigned in the NC machine data, then the display remains hidden.

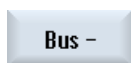
Procedure



1. Select the "Diagnostics" operating area.

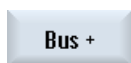


2. Press the menu forward key and the "Bus TCP/IP" softkey.
The "PROFINET/PROFIBUS Diagnostics" window opens.



3. Press the "Bus -" or "Bus +" softkey to select the desired bus.

...



4. Select the PROFINET device for which you want to display details.
5. Press the "Details" softkey.
The "PROFINET Diagnostics ... - Details" window opens.

See also

Accessible nodes (Page 91)

11.4 Backing up log data for service case

If system problems occur, it is necessary to back up all of the existing log files to provide them to the hotline for diagnostics. There are two options available to you:

- Backing up log data to USB
- Backing up log data as ZIP file

11.4.1 Backing up log data to USB

Backing up all log data using a special key combination:

<CTRL> + <ALT> + <D>

This function generates a directory on the USB flash drive/CF card. If a USB flash drive or CF card is available, the directory is created on the storage medium. The directory name has the following structure:

LOG_date_time

Example

LOG_171102_083615 stands for a directory created on 02/11/2017 at 8:36:15. This therefore ensures that a directory is not overwritten by pressing <CTRL> + <ALT> + <D> several times. The directory contains all of the logbook and debug information available in the system.

11.4.2 Backing up log data as ZIP file

Backing up all log data using a special softkey in the Diagnostics operating area:

"Diagnostics" → "Alarm list" → "Save diag. data"

This function collects all available LOG files of the operating software and stores them as a ZIP file in the following directory:

\user\sinumerik\didac\out_<Date-Time>.7z

In there is a problem with the system, you can send the ZIP file to the SINUMERIK hotline to help with the analysis of the problem.

11.5 Trace

The trace provides you with an oscilloscope function that allows the following data (signals) to be graphically displayed:

- General NC data
- PLC data

Trace session

The function serves for troubleshooting and resolving faults as well as to analyze the machine and process performance. In a session, data (signals) are recorded briefly before and after an event.

Graphic display

The recorded data is optically displayed in the form of curves, the so-called characteristics graphs.

You have the option of individually setting the recording type.

Saving and downloading data

You can store the recorded data in an XML file. The saved file can then be re-loaded so that it can be evaluated at a later time.

Recording fundamentals: Signals (data)

Signals (data) specify what is to be recorded:

- NC variables (axis data, drive data, channel data, etc.)
- PLC data

Cyclic events



Interpolation clock cycle (IPO clock cycle)

PLC cycle OB1



Position control clock cycle (servo clock cycle)

Non-cyclic events



Starting a session

- Recording using a trigger
- Manually starting and stopping a trace

Analyzing the result

The following functions are available to analyze the result:

- Scaling characteristic graphs
- Zooming characteristic graphs
- Defining a characteristic range using markers

Sequence

- Create a trace session file as follows:
 - Select the variable and define its representation type.
 - Set the graphics window, and the save mode
 - Set the trigger property to define the start and end of recording
- Save the recording and its associated properties that have been set
- Analyze the result by making the settings for the display in order to view additional details (e.g. zoom in, scroll, select section, etc.)

See also

Trace options (Page 318)

Saving the trace file (Page 329)

Variables filter/search (Page 313)

Trace settings (Page 315)

Evaluate a trace (Page 319)

11.5.1 Variables filter/search**Searching for PLC/NC/servo variables**

To limit the selection of variables for PLC/NC/servo, set a filter and/or enter a start value for the search:

Filter:	Display of the following variables:
System variable	System variables
NC	NC variables
Axis	Axis variables
Channel	Channel variables
Mode groups	Variables of the mode groups in OPI notation
Servo	Servo trace variables
Global user data	GUD such as: SGUD, MGUD, UGUD, GUD4, ...
PLC	PLC variables
PLC alarms	Manufacturer-specific PLC alarm signals
All (no filter)	All variables in alphabetical order

Inserting variables

The start value for "Filter/Search" of variables differs. For example, to insert the variable \$R[0], enter the following start value:

- 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.

The GUD from the machine data is only displayed in the Search window for the variable selection when the associated definition file has been activated. Otherwise, the sought variables must be entered manually, e.g. GUD/SYG_RM[1]

The following machine data is representative for all variable types (INT, BOOL, AXIS, CHAR, STRING): MD18660 \$MN_MM_NUM_SYNACT_GUD_REAL

Searching for drive variables

The restriction of the search is performed in this case via the selected drive object. In the Search window, you can restrict the selection by entering alphanumeric values, for example. Further restrictions of the selection are possible via the "All" or "Recommended" interconnections.

Procedure

For a trace session, in the "Selected Variables for Trace: Session ..." window, set the variables whose signals are to be traced.



1. In the "Select variables for trace: Session..." window, a trace session is displayed.

2. Position the cursor in the "Variable" column in an empty line.

- OR -

Position the cursor in the "Variable" column on a variable that you wish to replace by another.



3. Press the "Insert variable" softkey.
The "Select Variable" window opens.

Note:

When selecting a "Drive parameter", firstly select the drive object (DO) and then insert the desired parameter from the list of corresponding parameters.



4. Press the "Filter/search" softkey to start a specific search.

In the "Filter" selection box, select a variable area.

- OR -

Directly enter the variable into the "Search" entry field.

The cursor jumps to the term that is found.

Press the "Delete search" softkey if you wish to terminate the search.



5. Press the "Add" softkey to enter the required variable into a trace session.
The "Variable Attributes" window opens and you can select the appropriate value (e.g. channel data, drive bus no.) from a combobox.



- OR -

Press the "Replace" softkey to replace an already selected version by another one.

6. Press the "OK" softkey to add or replace the variable.

Each variable is automatically allocated a reference number, which is displayed in the upper section of the window. This number is opened automatically allocated. If a version is removed, then the following variables are re-numbered.

7. Now, select the desired settings for display, such as color, pen, qty., event. etc.

Editing variables of a trace session

Key combination	Effect
Ctrl + X or right-click, shortcut menu "Cut"	Press the key combination or open the shortcut menu in order to cut out a variable.
Ctrl + C or right-click, shortcut menu "Copy"	Press the key combination or open the shortcut menu in order to copy a variable.
Ctrl + V or right-click, shortcut menu "Paste"	Press the key combination or open the shortcut menu in order to insert a variable.

11.5.2 Trace settings

11.5.2.1 Trace settings (PLC, NC, servo)

In the "Trace Settings (PLC, NC, Servo)" window, you define the conditions for starting and stopping the trace session.

Condition for Start trace

With "Start trace" softkey	The recording is started immediately after pressing the "Start trace" softkey.
If variable	The session is activated by a variable. The desired variable is selected via the "Insert variable" softkey or "Quick list" softkey.
Condition and value	After the session has been activated, this defines at which value the trace recording is started. The following values are offered: =, ≠, >, >=, <, <=, ascending, descending, changed.
Data acquisition [s]	
Specifies where the recorded data is saved:	
- With "On NCK", the recorded data is saved in the NC work memory until the recording is stopped. Only then is the session file saved on the CompactFlash card. - With "CF card" or "On hard disk", the data is saved during the recording on the CompactFlash card or hard disk (e.g. of the PCU). As a consequence, more signals can be processed.	

Condition for Stop trace

With "Stop trace" softkey	The recording is stopped after pressing the "Stop trace" softkey.
If the time has expired	Defines the period which after expiry, the trace recording is exited.
If the memory is full	The trace session is terminated when the specified data volume (KB) is reached.
If variable	The recording is stopped by a variable. The desired variable is selected via the "Insert variable" or "Quick list" softkey.
Condition and value	After the session has been activated, this defines at which value the trace recording is exited. The following values are offered: =, ≠, >, >=, <, <=, ascending, descending, changed.
Delay time [s]	This defines how long the trace still runs after the session has been exited.
Overwrite at memory limit (KB)	Defines from which memory limit the data is overwritten (ring buffer).

Note**Loss of recorded data when maximum duration is exceeded**

Traces with the parameter assignment "Stop trace - If variable - Overwrite at memory limit" may not exceed a maximum period between the start and the finish of the recording.

If the maximum period is exceeded, the stop event is ineffective and the recording cannot be completed as usual.

The maximum period can be calculated as follows:

Use the temporary setting "Stop trace - When time expired" to find out how much memory space is required per minute for the set recording, e.g. 602 kB/min.

The maximum permissible time in minutes which may pass between the start and finish of a recording is computed using the following formula:

$$2^{31} (2\,147\,483\,648) \text{ bytes} / \text{memory space required for the set recording in bytes per minute}$$

$$602 \text{ kB} = 602 * 1\,024 \text{ bytes} = 616\,448 \text{ bytes, per minute } 2\,147\,483\,648 \text{ bytes} / 616\,448 \text{ bytes per minute} = 3\,483 \text{ minutes} = 58 \text{ hours}$$
Note**Stop condition with non-effective overtravel time and memory limit**

The settings for the overtravel time and memory limit are not effective if you select PLC signals with the result "PLC cycle OB1" for the trace.

Set at least one PLC signal before the trace to the result "Interpolation cycle (IPO cycle)" so that the data for the overtravel time and memory limit are recorded.

11.5.2.2 Trace settings (drive)

In the "Trace Settings (Drive)" window, you define the conditions for starting and stopping the trace session:

Condition for Start trace

With "Start trace" softkey	The recording is started immediately after pressing the "Start trace" softkey.		
If variable	The recording is activated after pressing the "Start trace" softkey when the selected variable clearly satisfies the condition and the specified value. The following conditions are offered: • Positive edge • Negative edge • With bit mask The "Bit mask" softkey is displayed when the "Bit mask" condition is selected in order to select the desired bits. Irrespective of the maximum value (hexadecimal) which is possible through the bit mask, a "Value" (decimal) can be entered at which the trace recording starts precisely. Example: The trace recording is started when the following bits are set: Bit 0 = 0; bit 1 = 1; bit 2 = 0; bit 3 = 1		
	Bit mask = fH	Condition	Value (decimal)
	Variable: CU/r722/3.3.1	With bit mask ☒ Bit 0 ☒ Bit 1 ☒ Bit 2 ☒ Bit 3	10
Preprocessing time [s]:	The input of a preprocessing time corresponds to the functional parameter p4722 "Trace trigger delay": • A positive preprocessing time starts the recording before the trigger event. • A negative preprocessing time starts the recording after the trigger event. When entering a preprocessing time, the entire duration of the recording must be taken into account: • If the preprocessing time is greater than the entire duration of the recording, the entry is rejected. • If too large a preprocessing time is entered, the trigger event may not be recorded.		
Data storage on CU			
Drive trace rate [μs]:	Enter a factor with the "Sampling rates" softkey to set the effective sampling rate. With a higher sampling rate, the recording duration is also longer.		

Condition for Stop trace

If the time has expired	Defines the period which after expiry, the recording is exited.
If the memory is full	The recording is terminated when the specified data volume (KB) is reached.

11.5.2.3 Trace options

You define the following properties of a trace session in the "Trace - Options" window.

- Setting the graphic window
 - Displaying main grid lines: You define whether a horizontal grid is displayed and in which color.
 - Displaying grid sub-lines: You define whether a vertical grid should be displayed and in which color.
 - Background: You select the background color of the graphic window.
- Defining storage options
 - Activate the "Confirm file overwrite" checkbox in order to receive a confirmation prompt each time a trace session is to be overwritten. If the checkbox is deactivated, the trace session is overwritten without a confirmation prompt.
 - Activate the "Save file in both formats" checkbox in order to also save the data as a csv file (Page 329) in addition to the xml session file. The file in csv format cannot be imported again.
 - Activate the check box "Save file in HMI Advanced Servo Trace Format" in order to store additional export data in the HMI Advanced servo-trace format. The following file formats are supported:
 - SVT files (trace session settings)
 - st1 files (variables of the first and second trace sessions)
 - st2 files (variables of the third and fourth trace sessions)

Procedure



1. You are in the "Select Variables for Trace: ..." window and a trace session has been selected.



2. Press the "Options" softkey.
The "Trace - Options" window opens.



3. Select the desired settings and press the "OK" softkey in order to accept the settings.

11.5.3 Starting the trace

In the "Trace" window, you obtain the curve with the data of the variables selected for a trace session.

The diagram comprises two Cartesian coordinates:

- Time axis
The horizontal x axis supplies the time values.
- Value axis
The vertical y axis displays the resolution fineness (amplitude).

Note**Display of real values of PLC variables**

If the value format is displayed incorrectly in the trace, extend the variable specification in the "Select variable for trace" window with the parameter assignment :REAL, e.g. to DB21.DBD76:REAL. The real values of the variable will then be displayed in the trace.

Recording the trace session

- Manually starting and stopping a trace
- Starting and stopping recording using a trigger

Procedure

1. You have listed the desired variables for a trace section.



2. Press the "Display trace" softkey.
The "Trace" window is opened. The recording is loaded and the procedure is displayed in progress window.



3. Press the "Start trace" softkey if you wish to immediately start recording.

- OR -

The recording is triggered by the trigger condition.



4. If you wish to manually end the recording, press the "Stop trace" softkey.

- OR -

The recording is stopped by the trigger condition.

11.5.4 Evaluate a trace**11.5.4.1 Selecting attributes of a variable****Attributes**

For a trace session, select the variables and assign the corresponding attributes:

Column	Meaning
Variable	Address of the variables

Attributes	Meaning
Comment	- Name of the selected variables - A general description for the variables.
Color	Color selection, the representation of the characteristic graphs. When creating a variable, the next color from the available palette is automatically used.
Pen	Line type selection to display the characteristic graphs. - No line - Solid line - Dashed line - Dash-dot line - Dash-dot-dot line - Dotted line - Stepped line XY - Stepped line YX "* line" (line made up of * symbols) + line - X line
Qty.	Defines whether the characteristic of the variables is displayed in the "Trace" window. If the checkbox is not selected, then no characteristic graphs are displayed.
Events	Event selection which triggers that the signal is traced: - Interpolation clock cycle (IPO clock cycle) - Interpolation cycle (IPO2) - Position control clock cycle (servo clock cycle) - PLC cycle OB1 - Drive trace rate - Start geometry axis/direction change - Stop geometry axis - Machine axis start / direction change - Machine axis stop - NC start (program runs) - NC start (program ran) - Start of the data recording

Attributes	Meaning
	- Block start, type 1 or block end (all program levels, without intermediate blocks) - Block start, type 2 or block end (all program levels, with intermediate blocks) - Block start, type 3 or block end (all main programs, without intermediate blocks) - Block start or block end, block search (all program levels, without intermediate blocks) - Geo axis start / direction change (2nd event) - Geo axis stop (2nd event) - Block start, type 2 (2nd event: all program levels, without intermediate blocks) - Block end, type 2 (2nd event: all program levels, with intermediate blocks) - Block start, block advance, type 1 (2nd event) - OEM test event 1 (non-cyclic, block advance) - OEM test event 2 (non-cyclic, main run)
	- Activate/deactivate synchronized action - Synchronized action initiated (condition fulfilled) - Signaled alarm - Cancel softkey pressed - Program level change - Block end (Interpreter) - Trace end (last event!) - Start trace
	- WRTPR part program command - WRTPR part program command (block search) - Tool change - Cutting edge change - Tool change (block search) - Cutting edge (block search) - Start trigger initiated - Stop trigger initiated - Block end (block advance) - Block end (2nd event: Block search) - WRTPR part program command (block advance) - Tool change (block advance) - Cutting edge (block advance)
Event channel	This means that the channel can be defined in SINUMERIK in which a specific event occurs. For events which are not channel-specific, the field remains empty.

Attributes	Meaning
Bit mask	In the case of an integer variable, a bit mask can be specified in this field. For a bit mask, the particular signal value is AND'ed before being displayed. After the mask has been applied, all of the selected bits are moved to the right so that it looks as if the bit or the bits all start with bit zero. This means, if all bits with the exception of bit 7 have been masked, then the integer number that would have been obtained would either have a value of 0 or 1, however, not 0 or 128. If all of the bits with the exception of bit 7 and 0 have been masked, the resulting integer number would either have the value 0, 1, 2 or 3, however, not 0, 1, 128 or 129.
Decimal places	This setting is used to define how many places to the right of the decimal point are displayed at the axis identifiers.
Coord. axis	The coordinate axes are displayed to the left or right in the graphic window or there is no display.
Display Y	Value input or 0
Scale factor	Defines the scale.
Units	Displays the measurement unit, e.g. mm/min. The system specifies this and it cannot be changed.

The most important information and settings are displayed at a glance in the detail window:

- Variable address
- Comment with a description of the variables
- Events
- Channel, axis, access level, mode group, etc.
- Smaller graphic display with the event; when this event occurs the characteristic graphs are recorded as well as their attributes, such as color or line type.

Procedure



1. You are in the "Select variables for trace: ..." window.
2. Position the cursor to the required variable, and press the "Details" softkey. A subscreen with the most important information as well as the graphic display is shown in the lower half of the window.
3. Press the "Details" softkey again to display the subscreen.

11.5.4.2 Setting trace views

In order to permit a precise evaluation of the recorded signals, you have the option to edit the view as well as the display of the characteristic graphs:

- Selection/legend
 - Show legend: In addition to the graphic display, the data of the variable is displayed as comment, numbered and in color.
 - Make a selection: You define which variables are displayed in the graphic window.
- Scale
 - Specifying scaling factors for minimum and maximum X and Y value
- Zoom
 - Increasing or decreasing the section
- Cursors: The cursor display provides functions that allow one, two or both curves to be precisely measured.
 - Cursor A
 - Cursor B
 - Both cursors
 - Snap to curve: The cursor precisely moves along the curve
 - Point mode: Cursor jumps directly to various values.
 - High point mode: The cursor jumps from one peak value to the next.
 - Low point mode: Cursor jumps to the lowest values.

Requirement

The "Trace" graphic window is open and the trace has been recorded.

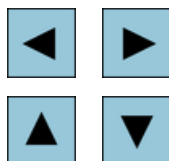
Editing curves



You jump from one variable to the next using the <TAB> key.



You jump back using the <Ctrl> and <TAB> keys.



With the cursor keys, you set marks, move to individual values and can define a zoom range.

11.5.4.3 Selecting a variable

You have the option of displaying the selected variables as legend and to make a selection using a checkbox.

Legend
Numbering the variables
Axis identifier
Comment of the variables

Displaying a legend and selecting a variable



1. A recording is opened in the "Trace" graphics window.
2. Press the "Legend / selection" softkey.
The graphic window is halved. The variables are displayed as comment and with checkbox in the left-hand section of the window.
3. Activate or deactivate the required variable using the checkbox to display it as curve.
4. Press the softkey again to hide the legend area again.

11.5.4.4 Scaling the display

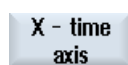
The scaling allows you to make a precise measurement of selected variables from the X time axis as well as from the Y value axis. You have the option of saving the larger or smaller display in a user-defined way

Setting	Meaning	
X minimum or X maximum	Highest or lowest value of the X time axis. The values are used for recording when the trace is started again.	
	Fixed scale, X axis	Values of the X time axis are kept.
	Strip chart recorder mode	The values of the X time axis are used while recording.
Y minimum or Y maximum	Defining the highest or lowest value of the Y value axis (amplitude).	
Offset or resolution	Defining the rms values (mean values) of the amplitudes. The scale is changed by a factor of 10.	

Scaling the time and value axis



1. A recording is opened in the "Trace" graphics window.
2. Press the "Scale" softkey in order to scale the display.



The "Select X-Y Scaling" input window opens and the values of the X time axis are displayed.



3. Press the "X time axis" softkey again if you wish to hide the values.

- OR -



Press the "Y selected curves" softkey to display the values of the Y value axis.



Press the softkey again if you wish to hide the values.

- OR -

You have selected both axes.



4. Press the "Scale +" or "Scale -" softkey until the desired scaling is reached.



- OR -

Enter the scaling values directly into the input window and select the appropriate property.



5. Press the "Scale" softkey to save the settings.

- OR -



Press the "Adapt selected" softkey to display the selected characteristic graphs in the full graphic window.



Press the "Fit one" softkey to display the selected characteristic graphs in the full graphic window.

11.5.4.5 Zooming the display

Using the zoom function, you have the option of viewing selected sections of curves in detail. After you have defined an area or a section, you can increase or decrease the size of it (zoom in or zoom out).

Procedure



1. A recording is opened in the "Trace" graphics window.



2. Press the "Zoom" softkey if you wish to zoom-in or zoom-out on the characteristic graphs.



3. The "X time axis" or "Y value axis" softkey is selected.





4. Press the "Adapt individually" or "Adapt all" softkey if you wish to display all selected characteristic graphs in a separate area positioned one above the other.

or



- OR -



Press the "Adapt together" softkey if you wish to adapt all of the characteristic graphs to the graphic window.

As all of the curves fill the complete window area, they are displayed superimposed on one another.

- OR -



Press the "Individually adapt X" softkey if you wish to normalize the X time axis only in the graphic window.

- OR -



Press the "Individually adapt Y" or "Adapt all" softkey if you only wish to scale the Y value axis in the graphic window.

or



- OR -



Press the "Adapt Y together" softkey if you wish to adapt the value axis of all characteristic graphs to the graphic window.

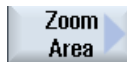
As all curves fill the complete window area, they are displayed superimposed.



5. Press the "Zoom +" or "Zoom -" softkey until the desired display size is reached.



Zoom in/zoom out area

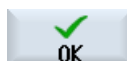


Press the "Zoom area" softkey to define a specific section.

A frame is displayed.



Press the "Zoom +" or "Zoom -" softkey to change the frame step-by-step.



Press the "OK" softkey to display the zoom area.

11.5.4.6 Position the cursor

Using the cursor, you have the option of determining the precise values from a defined section. To do this, the cursor is positioned directly on the curve.

Procedure



1. A recording is opened in the "Trace" graphics window.



2. Press the "Cursor A" softkey.
The reference number, color of the variables and the color of the coordinate crossing point are displayed on the softkey.



3. Position "(A)" at the position on the curve where you wish to start the measurement.

The coordinates are specified in a tooltip and in the status line.



4. Press the "Cursor B" softkey.
The reference number, color of the variables as well as the color of the coordinate crossing point are also displayed on this softkey.
Position "(B)" at the position on the curve where you wish to end the measurement.



5. Press the "Both cursors" softkey to display the area between A and B.
The coordinates of A and B as well as the difference " Δ " are displayed in the status line.



Press the active "Cursor A" and "Cursor B" softkeys again to deselect the position.



When re-selected, the positions are reset.



Press the "Crop to screen" softkey to display the area, defined by "Cursor A" and "Cursor B", in the complete graphic window.



Press the "Snap to waveform" softkey to move the cursor along the curve step-by-step.

11.5.4.7 Acquiring measured values

You have the option of directly determining the following values of a curve:

- Peak-to-peak value
- Peak-to-valley value

Procedure



A recording is opened in the "Trace" graphics window.



Press the "Point Mode" softkey to continuously move the cursor along the curve.



Press the "Peak Mode" softkey to position the cursor at the highest value.



Press the "Valley Mode" softkey to position the cursor at the lowest value.



If you press the "Peaks" softkey, the cursor only goes to the peak-peak values.



If you press the "Minimum" softkey, the cursor only goes to the peak-valley values.

11.5.5 Trace session files

11.5.5.1 Creating a session file

Procedure



1. Select the "Diagnostics" operating area.



2. Press the menu forward key and the "Trace" softkey.



3. Press the "Load trace" and "New" softkeys.
The "New Session" window opens in which the "Session" name is recommended. The file format is automatically specified as ".xml".



4. Supplement the file name or enter a new file name.



5. Select a variable type and press the "OK" softkey.

You return to the "Select variables for trace:...." window.

The name of the session that has been newly created is displayed in the header.

See also

Variables filter/search (Page 313)

11.5.5.2 Saving the trace file

There are two options of saving a trace session:

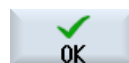
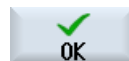
- Variables and settings: Only the variables and the settings are saved.
- Variables, settings and recorded values: The variables, settings and the recorded values are saved.

Note

Save option

If you have activated the "Save file in both formats" checkbox at "Trace" → "Options (Page 318)", the session file is saved as a csv file in addition to xml format.

Procedure



1. You have edited an active trace session.
2. Press the "Store trace" softkey.
The "Save Trace: Session ..." window opens and offers the following storage options.
3. Activate the corresponding checkbox and press the "OK" softkey.
The "Store Trace" window opens and specifies the file name in the "Name of trace file:" entry field. The file format is automatically specified as ".xml" and cannot be changed.
4. Press the "OK" softkey to save the session file.
If the session file already exists, then you obtain a corresponding confirmation prompt.
5. Press the "OK" softkey if you wish to overwrite the existing session file.
- OR -
Press the "Save as" softkey if you do not wish to overwrite the existing session file.
A new name is suggested but you can specify a name yourself.
- OR -
Press the "Change directory" softkey if you wish to store the session file in a directory other than in the standard "Trace" directory.
The data tree opens and you can select between the configured drives.
Press the "New directory" softkey if you wish to create a new directory.
The "New Directory" window opens in which you can assign a directory name.



Press the "OK" softkey.

The "Store Trace" window opens in which the names of the target directory and the session file are displayed.

You can change the name of the session file.



6. To save the session file, press the "OK" softkey one more time.

- OR -

If the session file with this name already exists, then you obtain a corresponding confirmation prompt.

11.5.5.3 Load trace session file

You have the option of loading already saved session data, to restart them or to display the trace that has been recorded.

Settings	Meaning
Variables, settings	You can restart a trace session that has already been created.
Variables, settings and recorded values	You can view the result of a trace session, change the variable and if required, restart.

Precondition

A session file of recorded traces is available.

Procedure



1. You have selected the "Trace" function.



2. Press the "Load trace" softkey.

The "Load Session File: Please Select File" window opens.



3. Select the desired session file from the appropriate directory, e.g. "Trace", and press the "Display trace" softkey.

The "Load trace: session..." is opened.

4. Select the checkbox "Variables, settings", if you wish to immediately start the trace session.



5. Press the "OK" softkey and the "Start trace" softkey to restart the trace.



- OR -

Select the checkbox "Variables, settings and recorded values", if you wish to display the trace session.

Press the "OK" softkey.



The data of the selected session are loaded displayed in the "Trace" window.

11.6 Drive system

11.6.1 Displaying drive states

Status display

The "Drive System Diagnostics" window displays the status information about the drive units and the associated drive objects. The status symbols have the following meaning:

Display icon		Meaning
	Green	The drive object is in (cyclic) operation, without any detected problems.
	Yellow	The drive object has detected a less serious problem, i.e. a warning is pending, for example, or enables are missing.
	Red	This drive object has detected a serious problem, for example, an alarm is pending.
	Gray	The drive status could not be determined for this drive object.
#	Special characters	Error when reading the data.

Procedure



1. Select the "Diagnostics" operating area.



2. Press the menu forward key and the "Drive system" softkey.



3. The "Drive System Diagnostics" window opens.
The name of the selected drive object is displayed in the window title.



Press the "Drive object +" or "Drive object -" softkey.
The next (+) or previous (-) drive object is selected.



- OR -



Press the "Select drive unit" softkey.
The "Select Drive Unit" window opens.

4. Select the desired drive unit via the drop-down list box and press "OK" to confirm the selection.

See also

Displaying details of the drive objects (Page 332)

11.6.2 Displaying details of the drive objects

Drive system diagnostics - details

The following information is displayed for the drive objects:

- Status display (r0002)
 - The status display (parameter r0002) of the drive object is displayed.
- Missing enable signals (r0046)
 - The enable signals are diagnosed except for the drive objects that do not have enable signals (e.g. Control Unit). Missing enables are shown in a pane below the displays.
- Commissioning parameter filter
 - (p0009) – control unit
The value of the "Device commissioning parameter filter" control unit parameter (p0009) is displayed.
 - (p0010) – further drive object
The value of the "Drive object commissioning parameter filter" parameter (p0010) is displayed.
- Current fault
The alarm number of the actual drive object fault is displayed or "No fault present", when no drive object fault is pending.
You obtain an overview of the pending faults with the following data in an additional window:
 - Alarm number
 - Time: Date and time
 - Alarm text
- Current warning
The number of the current drive object warning is displayed or "No warning present", when no drive object warning is pending.
You obtain an overview of the pending alarms with the following data in an additional window:
 - Alarm number
 - Time: Date and time
 - Alarm text

Meaning of the symbols

Display icon		Meaning
	Green	The relevant value of the drive object signals no problem detected.
	Yellow	The relevant value of the drive object signals a less serious problem, i.e. a warning is pending, for example, or enables are missing.
	Red	The relevant value of the drive object signals a serious problem, for example, an alarm is pending.
	Gray	The drive status could not be determined for this drive object.
#	Special characters	Error when reading the data.

Procedure



1. Select the "Diagnostics" operating area.



2. Press the menu forward key and the "Drive system" softkey.



The "Drive System Diagnostics" window opens.



3. Select the drive object for which you want to display further details.
4. Press the "Details" softkey.

The "Drive System Diagnostics - Details" window opens.

The name of the drive object is displayed in the window header.



Press the "Faults" softkey.

The "Drive Unit Faults" window opens that shows an overview of the faults that have been output.



Press the "Alarms" softkey.

The "Drive Unit Alarms" window opens that shows an overview of the alarms that have been output.



You can display the pending SI messages if Safety Integrated is integrated in your controller.

11.7 Action log

11.7.1 Setting the action log

In the "Action Log Settings" window you specify whether the operating sequences are to be logged.

Here, you select which control actions should be logged, so that operator actions and sequences can be tracked and understood at a later date.

When the action log is active, you can display recorded traces in the system data using the following files:

- "actual_actionlog.com", actual action log report
- "actual_crashlog.com", crashlog backup

Data that can be logged

Logging on	Logging is activated or deactivated.
Alarm status change	Incoming and outgoing alarms are logged.
Keyboard actions	All actions on the operator panel front and on an external keyboard are logged.
Channel status change	NC/PLC states are logged via the information channel status. If this information can be obtained in a timely manner, in many cases these states can be used to verify the operation of the machine control panel.
Window switchover	The form and dialog names (assigned by the programmer) are logged each time a window is opened or closed.
Writing NC/PLC data	The writing of NC and PLC variables is logged.
File access	Copying to the NC is logged.
Function calls in the NC (PI service)	Some program calls, for example, ASUB, are logged.
Curr. program status	Additional information for particular events is activated. For important alarms that require NC stop, NC start, or NC reset, actual values and the current block are recorded for example. The program ascertains which channel and which spindle are used for the additional information.

Write interval file**Settings for updating files:**

- "automatic": The action log saves the information to an internal buffer. If the buffer is full the entries are written to the CF card. The entries may be lost when the control is switched off.
- "after every event": All entries are backed up immediately, this prevents data loss due to a power failure or similar.
Notice: As CF cards only permit a limited number of write accesses, this setting is not recommended for normal operation.
- "time-controlled": New entries are backed-up for a particular length of time. An additional input field appears in which you can specify a time in seconds.

Save log as file for alarm(s) The alarm numbers for which a "crash log" is generated are specified. The alarms are entered separated by a comma.

Generating a crashlog file

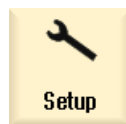
"actual_crashlog.com" is a backup file of the actual log when a critical event occurs (for example, Emergency Stop alarm).

The file is then no longer stored in the ring buffer of the action log and can no longer be overwritten with new entries.

The entries in the crash log file are only overwritten when yet another critical events occurs.

At the beginning, the file has no entries and is only written to after the following event:

- The alarm entered in the "Save log as file for alarm(s)" entry field is output.

Procedure

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



2. Press the "HMI", "Diagnostics", and "Action log" softkeys



3. The "Action Log Settings" window opens.
The "Data selection" area is active by default.
4. Activate the "Logging on" checkbox to activate the action log.
5. Activate the relevant checkbox if you want to acquire particular events in the action log.

11.7.2 Inserting variables

In the "Action Log Settings" window, you can insert up to 50 variables that you want to log or monitor.

You select available variables from the drop-down list or add them manually to the list.

Inserting variables from a variable list

1. You are in the "Action Log Settings" window.
2. Press the "Variable selection" softkey.
Window "Action log variables for continuous monitoring" opens.
3. Press the "Insert variable" softkey to insert a variable from a list of all existing variables.
The "Select variable" window opens.
4. Select the desired variable from the list.
5. Click the "Add" softkey.
The "Variable attributes" window opens.
6. Accept or, if necessary, change the displayed properties, for example, the axis assignment.
7. To limit the variable display, select the relevant category in the header, e.g. "System variables".
- OR -
Press the "Filter/search" softkey to limit the display of variables via the "Filter" selection box and/or select the desired variable via the "Search" entry field.
8. To delete the search query, press the "Clear search" softkey.
9. Press the "OK" softkey to confirm changes.
- OR -
Press the "Cancel" softkey to cancel the changes.

Inserting variables manually

1. You are in the "Action Log Settings" window.
2. Press the "Variable selection" softkey.
The "Variables for ongoing monitoring" window opens.
3. Position the cursor in the "Variable" column and enter the required variable.
4. In the "Format" column, select the supported variable format from the drop-down list.
5. Press the "OK" softkey to confirm the changes or the deletion.
- OR -
Press the "Cancel" softkey to cancel the changes.

11.7.3 Displaying the log file

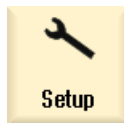
Loggable data that was logged when the action log function was active is displayed in the "Action Log File" window.

Log files

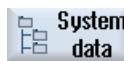
The actual data can be displayed using the following files:

- "actual_actionlog.com"
- "actual_crashlog.com"

Procedure



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



2. Press the "System data" softkey.

The data tree is displayed.

You will find the saved files in the HMI data folder under /logs/action log.

3. Select the desired com file and press the "Open" softkey or double-click the file.



4. Press the "Display new" softkey to update the display of events in the log (actual_actionlog.com).

The entries made since the log file was called up are displayed.

11.7.4 Searching in the log files

You can search for specific events in the log.

Precondition

The relevant log file is open.

Procedure



1. Press the "Search" softkey.

A new vertical softkey bar appears. The "Search" window opens at the same time.

2. Enter the desired search term in the "Text" field.



3. Position the cursor in the "Direction" field and choose the search direction (forward, backward) with the <SELECT> key.



4. Press the "OK" softkey to start the search.

If the text you are searching for is found, the corresponding line is highlighted.



Press the "Continue search" softkey to continue the search, as necessary.



- OR -

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

Additional search option



1. Press the "Go to End" softkey to scroll to the end of a large log. You get to the oldest entry of the recording.



2. Press the "Go to Beginning" software to return quickly to the most recent log entry.

11.7.5 Storing a log

You can store the displayed log, e.g. in the "HMI data/Logs/Alarm logs" directory. It is stored as a binary file as well as ASCII file. You can read an ASCII file with any editor.

In addition, you can save logs in the file format "*csv" (Comma Separated Value). You can evaluate the CSV file in Microsoft Excel.

The following logs can be generated:

- action.com (binary file)
- action.log (ASCII file)
- action.csv (Comma Separated Value file)
- crash.com (binary file)
- crash.log (ASCII file)
- crash.csv (Comma Separated Value file)

Requirement

The relevant log file is open.

Procedure

1. Press the "Save" softkey.
2. Select the storage path under which the log is to be saved.
3. You can choose between two file types for the action log ("action").
Select the file type from the "Type" drop-down list:
 - LOG
 - CSV

The log is saved in the specified directory under the name suggested by the system by default.

Note

If required, you can also assign your own name for the log file when saving it.

Note

The "Save" softkey is only available for files that have not yet been saved.

11.7.6 Structure of a log file

Content of the log file

The following data is recorded for every logged event:

HMI version		NC version	
System	Entry keyword		Date/time
	Entry text		
	DETAILS		

Entry		Description
HMI and NC version output		
System:		
	HMI	Keystrokes, window change
	NC	Write variables, PI services
	MSG	Alarm entries
	USR	Entries generated by the machine manufacturer for the user
	ERR	Action log error, e.g. log file not found
Entry keyword:		
	HMI_START	Entry of an HMI boot
	HMI_EXIT	Entry of an HMI shutdown
	PLC_CRASH	Entry of a PLC crash
	KEY_PRESSED	Entry of a keystroke

Entry		Description	
	KEY_HOLD	Entry of a key hold	
	KEY_RELEASED	Entry of a key released	
	ALARM_ON	Entry of an incoming alarm event	See below "Alarm display"
	ALARM_OFF	Entry of an alarm going event	
	ALARM_ACK	Entry of an alarm acknowledgement event	
	OPEN_WINDOW	Entry of a window opening	
	CHN_STATE_CHANGED	Entry of a channel status change	
	OPMODE_CHANGED	Entry of an operating mode change	
	TOOL_CHANGED	Entry of a tool change	
	OVERRIDE_CHANGED	Entry of override changes	
	DOM_CMD	Entry of a download to the NC	
	PI_CMD	Entry of a PI service	
	WRITE_VAR	Entry of a write to NCK/PLC variable	
	AREA_CHANGED	Entry of an area changeover	
	NC_CONNECTION	Entry of an coming/going NC connection	
	USER	User entry via the OEM interface	
	ACTIVATED	Action log was activated	
	DEACTIVATED	Action log was deactivated	
	INTERNAL	Internal action log entry	
Date/time		Date and time of the event For HMI_Start, PLC_Crash or date change since the last entry, the intermediate period between the entry keyword and date is designated with an "-".	
Entry text		Event data is written in plain text.	
DETAILS		If additional information regarding an entry is known, e.g. a recorded IPO trace.	

Example

Note

DB19.DBW24 interface signal

The DB19.DBW24 interface signal used in the example does not apply to the SINUMERIK 840D sl. It is DB1900.DBW4 for the SINUMERIK 828D.

```

NCK    WRITE_VAR                                18.02.2009   09:40:12
        ncui.local wrote variable: DB19.DBW24 = 0

HMI    KEY_PRESSED                              18.02.2009   09:40:12
        Key up: "Return" (1000004/ 0)

HMI    HMI_START -----                        18.02.2009   09:34:15
        HMI started.
```

Alarm display

```
MSG      ALARM                                     Date/time
      No.      Deletion criterion  Text
      DETAILS
```

Entry		Description
MSG		
ALARM		Type of alarm: ALARM_ON, ALARM_OFF, ALARM_ACK
Date/time		Data and time of the event
No.		Alarm number
Deletion criterion:		Information as to how the alarm was acknowledged
	AUTOMATIC	Automatic acknowledgement
	POWER ON	Acknowledgement using PowerOn
	RESET	Acknowledgement using NCK reset
	NC START	Acknowledgement using NC start
	NC RESET	Acknowledgement using NC reset
	ALARM CANCEL	Acknowledgement using Alarm Cancel key
	RECALL	Acknowledgement using the Recall key
	HMI	Acknowledgement using HMI
	PLC	Acknowledgement using PLC
Text		Display of the English alarm text
DETAILS		If additional information regarding an entry is known.

Example

```
MSG      ALARM_ON                                     20.02.2009   14:25:37
      8020    POWER ON   : Option 'activation of more than 1 channels' not
      set
      DETAILS:
      Mode: JOG Program: canceled Channel: interrupted
      Program-Level information:
      Level Program running:                               Invoc      Offset
      1          /_N_MPF0                                0          0
```

11.7.7 Advanced settings

You define the size of the log file of the action log in the "actlog.ini" configuration file.

Procedure

1. You can copy a sample configuration file "actlog.ini" from the following directory: /siemens/sinumerik/hmi/template/cfg.
2. Save or create the file in the /oem/sinumerik/hmi/cfg or /user/sinumerik/hmi/cfg directory.
3. Open the file and after "; ActionLogSize=", enter the file size (in bytes).

"actlog.ini" configuration file

```
Template for the configuration of Action Log
;
; To activate the settings remove the
; commentary ';' at the beginning of the line
[ActionLogSettings]
;To change the path of the internal action log file use the following
setting
;ActionLogPath=/user/sinumerik/hmi/ac_log/action.com
;To change the size of the internal action log file use the following
setting
;ActionLogSize=5000000
;To change the path of the internal crash log file use the following
setting
;CrashLogPath=/user/sinumerik/hmi/ac_log/crash.com
```

11.8 HMI trace

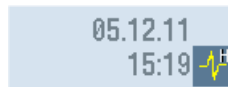
Creating an HMI trace

The "HMI trace" function is available for extended diagnostics of the operating software. The prerequisite is at least access level 3 (user).

Procedure to create an HMI trace file:

1. An expert of the Service & Support Center provides a sltrc_keybitset.ini file. Copy the sltrc_keybitset.ini file to a USB memory medium and load the file on the controller or operator panel.
2. In the operating area, select "Start-up" → "HMI" → "Diagnostics >" → "HMI trace".
3. Select the INI file on the USB memory medium with the "Load configuration" softkey.
4. Start the trace by activating the checkbox or from any place on the user interface with the following key combination:
<Alt> + <T> on the operator panel
<Alt> + <Shift> + <T> on an external keyboard

The following icon is displayed at the top right in the header to show that a trace is active:



5. On the user interface, repeat the operating sequence whose cause is to be clarified.
6. Stop the trace by deactivating the checkbox or from any place on the user interface with the following key combination:
<Ctrl> + <T> on the operator panel
<Ctrl> + <Shift> + <T> on an external keyboard
7. A sltrc.out output file is generated which you can copy to the storage medium with the "Export data" softkey.
8. Send the sltrc.out file for analysis back to the Service & Support Center.

Note

On completion of the analysis, it is recommended to delete all files under the following path: ../user/sinumerik/hmi/log/sltrc

Note

"Extended" softkey

The other options in this dialog support suitably qualified service engineers in the analysis of the system and evaluation of the HMI trace.

Appendix

A.1 Information about third-party software used

Copyright 1995 Sun Microsystems, Inc.

Printed in the United States of America.

All Rights Reserved.

This software product (LICENSED PRODUCT), implementing the Object Management Group's "Internet Inter-ORB Protocol", is protected by copyright and is distributed under the following license restricting its use. Portions of LICENSED PRODUCT may be protected by one or more U.S. or foreign patents, or pending applications.

LICENSED PRODUCT is made available for your use provided that you include this license and copyright notice on all media and documentation and the software program in which this product is incorporated in whole or part. You may copy, modify, distribute, or sublicense the LICENSED PRODUCT without charge as part of a product or software program developed by you, so long as you preserve the functionality of interoperating with the Object Management Group's "Internet Inter-ORB Protocol" version one. However, any uses other than the foregoing uses shall require the express written consent of Sun Microsystems, Inc.

The names of Sun Microsystems, Inc. and any of its subsidiaries or affiliates may not be used in advertising or publicity pertaining to distribution of the LICENSED PRODUCT as permitted herein. This license is effective until terminated by Sun for failure to comply with this license. Upon termination, you shall destroy or return all code and documentation for the LICENSED PRODUCT.

LICENSED PRODUCT IS PROVIDED AS IS WITH NO WARRANTIES OF ANY KIND INCLUDING THE WARRANTIES OF DESIGN, MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE, NONINFRINGEMENT, OR ARISING FROM A COURSE OF DEALING, USAGE OR TRADE PRACTICE.

LICENSED PRODUCT IS PROVIDED WITH NO SUPPORT AND WITHOUT ANY OBLIGATION ON THE PART OF SUN OR ANY OF ITS SUBSIDIARIES OR AFFILIATES TO ASSIST IN ITS USE, CORRECTION, MODIFICATION OR ENHANCEMENT. SUN OR ANY OF ITS SUBSIDIARIES OR AFFILIATES SHALL HAVE NO LIABILITY WITH RESPECT TO THE INFRINGEMENT OF COPYRIGHTS, TRADE SECRETS OR ANY PATENTS BY LICENSED PRODUCT OR ANY PART THEREOF.

IN NO EVENT WILL SUN OR ANY OF ITS SUBSIDIARIES OR AFFILIATES BE LIABLE FOR ANY LOST REVENUE OR PROFITS OR OTHER SPECIAL, INDIRECT AND CONSEQUENTIAL DAMAGES, EVEN IF SUN HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

Use, duplication, or disclosure by the government is subject to restrictions as set forth in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 and FAR 52.227-19.

SunOS, SunSoft, Sun, Solaris, Sun Microsystems and the Sun logo are trademarks or registered trademarks of Sun Microsystems, Inc.

SunSoft, Inc.
2550 Garcia Avenue
Mountain View, California 94043

Copyright (c) 1991 by AT&T.

A.1 Information about third-party software used

Permission to use, copy, modify, and distribute this software for any purpose without fee is hereby granted, provided that this entire notice is included in all copies of any software which is or includes a copy or modification of this software and in all copies of the supporting documentation for such software.

THIS SOFTWARE IS BEING PROVIDED "AS IS", WITHOUT ANY EXPRESS OR IMPLIED WARRANTY. IN PARTICULAR, NEITHER THE AUTHOR NOR AT&T MAKES ANY REPRESENTATION OR WARRANTY OF ANY KIND CONCERNING THE MERCHANTABILITY OF THIS SOFTWARE OR ITS FITNESS FOR ANY PARTICULAR PURPOSE.

This product includes software developed by the University of California, Berkeley and its contributors.

QLocale's data is based on Common Locale Data Repository v1.6.1.

Note

You can find additional information on the product DVD in the Readme_OSS file about the third-party software used.

A.2 Abbreviations

Abbreviation	Description
ASCII	American Standard Code for Information Interchange: American coding standard for the exchange of information
Mode group	Mode group
OPI	Operator Panel Interface
CNC	Computerized Numerical Control: Computerized numerical control
CEC	Cross Error Compensation: Sag compensation
CP	Communication Processor
DCP	Discovery and basic Configuration Protocol: ProfiNet protocol
DHCP	Dynamic Host Configuration Protocol
DIN	Deutsche Industrie Norm
DNS	Domain Name System
DIR	Directory: Directory
DRAM	Dynamic Random Access Memory
DRF	Differential Resolver Function: Differential resolver function (handwheel)
EES	Execution from External Storage
ERTEC	Enhanced Real Time Ethernet Controller
ESR	Extended stop and retract
ETC	ETC key ">"; Softkey bar extension in the same menu
FRAME	Data block (FRAME)
FIFO	First In - First Out: Method of storing/retrieving data in/from a memory.
GDIR	Global Directory: Global part program memory
GUD	Global User Data: Global user data
GUI	Graphical User Interface
MSD	Main Spindle Drive
HSC	High Speed Cutting: High-speed machining
ILC	Intelligent Load Control: Intelligent load adjustment
INC	Increment: Incremental dimension
INI	Initializing Data: Initializing data
IP	Internet Protocol
IPO	Interpolator
ISO	International Standard Organization
JOG	Jogging: Setup mode
LED	Light Emitting Diode: Light emitting diode
LLDP	Link Layer Discovery Protocol
LUD	Local User Data: Local user data
MAC	Media Access Control
MB	Megabyte
MD	Machine data
MDA	Manual Data Automatic: Manual input
MCS	Machine Coordinate System
MLFB	Maschinenlesbare Fabrikatbezeichnung

Abbreviation	Description
MPF	Main Program File: NC part program (main program)
MCP	Machine Control Panel: Machine control panel
NC	Numerical Control: Numerical control
NCK	Numerical Control Kernel: NC kernel with block preparation, traversing range, etc.
NCU	Numerical Control Unit: NCK hardware unit
NFS	Network File System
NTP	Network Time Protocol
ZO	Zero offset
OEM	Original Equipment Manufacturer
OID	Object Identifier: Information identifier
OP	Operator Panel
PCU	Programmable Control Unit
PI service	Program Invocation Services
PG	Programming device
PLC	Programmable Logic Control
RCS	Remote Control System
RFC	Request for Comments
REF	REference point approach function
REPOS	REPOSition function
ROV	Rapid Override: Input correction
RPA	R-Parameter Active: Memory area in NCK for R- NCK for R parameter numbers
SBL	Single Block: Single block
SD	Setting Data
SDB	System Data Block
SEA	Setting Data Active: Identifier (file type) for setting data
SK	Softkey
SMB	Server Message Block
SNMP	Simple Network Management Protocol
SPF	Sub Program File: Subroutines
SRAM	Static Random Access Memory: Static RAM (non-volatile)
SSH	Secure Shell: Network protocol
SW	Software
SYF	System Files: System files
TCU	Thin Client Unit
TEA	Testing Data Active: Identifier for machine data
TMA	Tool Magazine Active: Magazine data
TO	Tool Offset: Tool offset
TOA	Tool Offset Active: Identifier (file type) for tool offsets
UDP	User Datagram Protocol: Network protocol
UFR	User frame
VNC	Virtual Network Computing
FD	Feed Drive

Abbreviation	Description
WCS	Workpiece Coordinate System
TM	Tool management

Index

"

"Siemens Industry Online Support" app, 15

3

3D simulation, 119, 124

A

Access levels, 27

Access right

Effect, 31

Global, 31

individual, 30

Remote access, 31

Action log

actlog.ini, 341

Log file, 339

Open log file, 337

Saving log file, 339

Search for events, 337

Setting the log size, 341

Settings, 334

action.com, 338

action.csv, 338

action.log, 338

actlog.ini, 341

Actual value window

Setting font size, 96

actual_actionlog.com, 334

actual_crashlog.com, 334

Adapt to load, 225

Adapt tool measurement

Examples when milling, 260

for turning, 265

when milling, 259

Adapt workpiece measurement, 254

Archive

Login data (credentials) handling, 49

Automatic save data archive, 65

Axis diagnostics, 294

B

Backing up log data

USB, 311

ZIP file, 311

C

Changing the design, 48

Channel operating message

Standard configuration, 102

Channel operational message, 101

Circular position pattern, 151

Circumferential groove, 151

Clamping

Setting up function, 126

Contour grooving, 173

Contour milling, 150

Contour turning, 173

Coordinate system

MD52000, 130

crash.com, 338

crash.csv, 338

crash.log, 338

CUST_782.SPF, 136, 228

CUST_800.SPF, 136

CUST_832.SPF, 136, 231

CUST_CLAMP, 136

CUST_M6, 136, 138

CUST_MEACYC

Structogram, 239

CUST_MEACYC.SPF, 136

CUST_MULTICHAN, 136, 142

CUST_T, 136, 138

CUST_TECHCYC.SPF, 136

Aligning, 140

Cycle alarms, 197

CYCLE495, 189

CYCLE60, 151

CYCLE63, 150

CYCLE64, 150

CYCLE782, 225

CYCLE79, 151

CYCLE800

Structogram, 218

CYCLE832, 230

Top Surface, 232

CYCLE84, 135
 CYCLE840, 135
 CYCLE930, 173
 CYCLE950, 173
 CYCLE951, 173
 CYCLE952, 173
 CYCLE953, 160
 CYCLE99, 173
 CYCLE9960, 223
 Cycles version
 Define, 267
 Displaying, 267
 Cylinder surface transformation
 in turning, 181
 Milling, example, 156
 under milling, 154
 Cylindrical grinding, 188
 Displaying softkeys, 129

D

Data matrix code, 16
 Date
 Selecting the format, 37
 Setting, 38
 Device registration, 41
 DHCP server detection, 90
 Diagnostics
 Ethernet/PROFINET network, 90
 HMI trace, 343
 Direction of rotation
 Counterspindle, 170
 Main spindle, 164
 Drilling, 134
 Drive
 Global part program memory, 78
 Logical drive, 75
 Set EXTCALL, 77
 Setting up, 76
 Drive system
 Diagnostics, 331
 Diagnostics - details, 332

E

End face machining
 in turning, 182
 Engraving, 151
 Ethernet/PROFINET
 Diagnostics, 90

EXTCALL
 switch off and on again, 77

F

fast block search, 108
 Form-truing, 189

G

G code groups
 Configuring the display, 99
 G group 59, 230
 General Data Protection Regulation, 17
 Generating a crashlog file, 335

H

High-Speed Settings, 230
 Hirth teeth, 193
 HMI trace, 343
 HOLES2, 151

I

Inclined Y axis
 in turning, 183
 Inserting a run up screen, 46
 Intelligent load adjustment, 225

K

Keyboard
 Setting, 45
 Kinematics
 Check list, 197
 measured, 223
 with Hirth gearing, 193

L

Language
 Defining the selection mode, 43
 Log file
 Action log, 339

M

- Machine data
 - Configuring plain texts, 72
- Machine operating area, 95
- Machining time recording
 - activating, 117
- Manual machine
 - Activating the function, 109
 - Parameter assignment, 109
- Milling
 - Cylinder surface transformation, 154
 - Direction of rotation for position pattern, 152
 - Displaying softkeys, 129
 - Example, setting up a milling machine, 156
 - Settings under ShopMill, 152
 - Surface turning, 160
 - Tool measurement, 245
 - Workpiece measurement, 243, 257
- Multiple clamping of different workpieces (option), 158
- Multiple edge, 151
- mySupport documentation, 14

N

- NC/PLC variables
 - Changing, 287
 - displaying, 285
- Network
 - Configuring, 82
- Network diagnostics, 86
- Network interfaces
 - Configuring, 87

O

- Offset
 - Configuring, 116
- OpenSSL, 17
- Operating state, 291

P

- Password
 - Changing, 29
 - delete, 30
 - Set, 29

Probe

- Example of a test program, 236
- Testing the function, 235
- to measure a tool, 235
- to measure the workpiece, 235

Product support, 15**PROFINET**

- Connections, 307
- Devices, 309

PROG_EVENT.SPF, 137, 307**Program list**

- Configuring a softkey, 79
- Creating, 79

Program test

- activating, 103
- deactivate, 103

Programming G code programs, 113**Q**

- QR code, 41

R

- Replacing the logo, 97

S**Screenshots**

- Copy, 47
- Creating, 47

Search mode

- Configuring, 107

Send feedback, 13**Service drive**

- Display, 298
- Parameter description, 299

Service overview, 291

- Axes and drives, 293

Setting up a counterspindle

- ShopTurn, 169

Setting-up a turning machine with milling tools, 177**Setting-up measuring in JOG, 241****ShopMill**

- Settings for milling, 152

ShopTurn

- Setting up a counterspindle, 169
- Settings for turning, 164

Sidescreen

- activating, 57
- Display ABC keyboard, 59

- Display machine control panel, 60
- Display translations, 63
- In-house configuration, 61
- Siemens Industry Online Support
 - App, 15
- Simulation
 - deactivate, 122
 - from kinematic chain, 123
 - General conditions, 119
 - Grinding technology, 121
 - Milling technology, 121
 - Tool change time, 123
 - Turning technology, 121
- Simultaneous recording, 124
 - Activating the machining time, 125
 - Option, 124
- SINUMERIK, 9
- slmagcodeconfig.ini, 99
- SLOT2, 151
- Softkey
 - Access level, 35
 - Original setting, 35
- Spindle control, 271
- Spindle diagnostics, 273
 - Additional temperature sensor, 277
 - Clamping state, 279
 - Clamping time statistics, 282
 - Diagnostic statistics, 281
 - Logistics data, 283
 - Motor temperature sensor, 276
 - Speed limitation, 280
 - Spindle load, 278
 - Temperature, 274
 - Temperature histograms, 277
- Spindle utilization
 - Display of time remaining, 42
- Standard scope, 10
- STAT, 104
- Station-related network diagnostics, 86
- Status display
 - Inserting a logo, 97
- Status display with icons, 114
- Stock removal, corner, 173
- Surface grinding, 188
 - Displaying softkeys, 129
- Swiveling
 - activating, 190
 - Commissioning machine kinematics, 198
 - Configuring the input dialog, 191
 - Display variants, 200
 - Example HEAD_2 (chain), 207
 - Example MIXED_45 (chain), 211

- Example of turning machine with B axis (chain), 212
- Example TABLE_45 (chain), 209
- Retraction, 203
- Tool carrier via chain, 204

T

- Tailstock
 - Option, 124
- Tapping, 135
- TEACH IN, 105
- Technical support, 15
- Technology
 - Cylindrical grinding, 188
 - Drilling, 134
 - Milling, 150
 - Surface grinding, 188
 - Swiveling, 190
 - Turning, 173
- Thread-cutting, 173
- Time
 - Selecting the format, 37
 - Setting, 38
- Time server, 39
- Tool measurement
 - in turning, 251
 - under milling, 245
- Topologies, 91
- TRAANG
 - in turning, 183
- Trace
 - Changing the scale, 324
 - Creating a session, 328
 - Cursor A/B positioning, 327
 - Data acquisition, 315
 - Data storage on CU, 317
 - Define zoom area, 326
 - Display peak-to-peak value, 327
 - Display peak-to-valley value, 327
 - Displaying details of the variables, 322
 - Drive variables, 314
 - Editing a session, 315
 - Editing the graphic display, 323
 - Filtering variables, 313
 - Insert variable, 314
 - Load session, 330
 - Overview, 312
 - PLC/NC/servo variables, 313
 - Replace variable, 315
 - Save options, 318
 - Save session, 329

- Searching for variables, 313
- Selecting a variable, 314, 324
- Sequence, 313
- Session, 314
- Setting grid lines, 318
- Settings (drive), 316
- Settings (PLC, NC, servo), 315
- Start recording, 319
- Starting a session, 330
- Stop recording, 319
- Variable attributes, 319
- Zooming characteristic graphs, 325

TRACYL

- in turning, 181

Training, 15

Transformation

- Settings for milling, 157

TRANSMIT

- in turning, 182

TU, 104

Turning

- Cylinder surface transformation, 181
- Displaying softkeys, 129
- End face machining, 182
- Example, two-channel turning machine, 179
- Examples, setting up a turning machine, 177
- Inclined Y axis, 183
- Settings under ShopTurn, 164
- Speed limitation, 186
- Technology cycles, 173
- Tool measurement, 251
- Workpiece measurement, 264
- Zero point of the X-axis displaced, 184

U

User view

- Creating, 68
- Delete, 71
- Editing, 69
- Importing, 67

V

- Variable screen forms, 289

VNC operating area

- Settings, 51
- setup, 50

W

WEB Browser operating area

- Settings, 54
- setup, 54

- Websites of third-party companies, 10

- Work station safety, 30

Workpiece measurement

- under milling, 243, 257
- under turning, 264

Z

Zero point displaced

- under turning, 184

