

SIEMENS

Overview of controllers for vendors
of machine tools

SINUMERIK Operate – Turning

SINUMERIK 840D sl / SINUMERIK 828D

Edition

08/2021

www.siemens.com/sinumerik

SINUMERIK

SINUMERIK 828D/ SINUMERIK 840D sl SINUMERIK Operate - Turning

Control system overview for machine tools' sales
people

Valid for:

Controls
SINUMERIK 828D / SINUMERIK 840D sl
Software
CNC software version 4.9

08/2021

A5E41992653B AC




Preface

Preface	
Compact overview	1
System overview	2
CNC operation with SINUMERIK Operate	3
CNC operation in manual mode (JOG)	4
Tool management	5
Data management	6
CNC operation in automatic mode (AUTO)	7
CNC functionalities	8
CNC programming methods	9
Workpiece visualization	10
CNC technology cycles	11
Complete machining	12
Multi-channel machining	13
Automation	14
Digitalization	15
Tools and information	16
Safety functions	17

Legal information

Warning notice system

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

 DANGER
indicates that death or severe personal injury will result if proper precautions are not taken.
 WARNING
indicates that death or severe personal injury may result if proper precautions are not taken.
 CAUTION
indicates that minor personal injury can result if proper precautions are not taken.
NOTICE
indicates that property damage can result if proper precautions are not taken.


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

Qualified Personnel

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

Proper use of Siemens products

Note the following:

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

Trademarks

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

Disclaimer of Liability

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

Preface

Preface

Scope of validity

This document provides you with an overview of the range of functions included in the **SINUMERIK 828D** and **SINUMERIK 840D sl** with **SINUMERIK Operate V4.9** for turning machines.

The document is focusing on vendors and dealers of machine tools.

Organization of the information

- Of the varied functional features of the SINUMERIK products, only those are listed which are of direct value to the machine user.
- All functions contained in the machine's basic configuration are identified as follows:
 - ☑ Basic configuration
- All functions not contained in the machine's basic configuration are identified as follows:
 - ☑ Option: ...
- You can find a summary of the most important benefits in the chapter "Summary of unique features".
- For information on marketing the options through the machine manufacturer, please see the technical description of each machine.

Subject to change without prior notice

Contact person at machine manufacturer

Marketing & Sales

Phone:	+49 xxx xxx
Fax:	+49 xxx xxx
Email:	xxx@machinemanufacturer.com

Service

Phone:	+49 xxx xxx
Fax:	+49 xxx xxx
Email:	xxx@machinemanufacturer.com

Homepage:

<http://www.machinemanufacturer.com>

SINUMERIK user support worldwide contact

Name	First name	Region	Email
Dorko	Jan	Austria	jan.dorko@siemens.com
Marino	Gustavo	Brazil	marino.gustavo@siemens.com
Katsarov	Petar	Bulgaria	petar.katsarov@siemens.com
Xu	Zheng Shun	China	zhengshun.xu@siemens.com
Meriaho	Juha	Finland	juha.meriaho@siemens.com
Chevalier	Francois	France	francois.chevalier@siemens.com
Bui	Quang-Minh	Germany	quang-minh.bui@siemens.com
Palla	Csaba	Hungary	csaba.palla@siemens.com
Narayanan	Shankar	India	narayanan.shankar@siemens.com
Widjayanto	Arif	Indonesia	arif_widjayanto@cncdesign.com.au
Marrelli	Fabio	Italy	fabio.marrelli@siemens.com
Cavazos Valle	Alicia del Carmen	Mexico	alicia.cavazos@siemens.com
Ouwehand	Hans	The Netherlands	hans.ouwehand@siemens.com
Libner	Zbigniew	Poland	zbigniew.libner@siemens.com
Kudinov	Alexander	Russian Federation	alexander.kudinov@siemens.com
Lee	ByungChan	South Korea	byungchan.lee@siemens.com
Lopez Gomez	Javier	Spain	javier.lopez_gomez@siemens.com
Schindler	Ramona	Switzerland	ramona.schindler@siemens.com
Tan	Ben	Taiwan	ben.tan@siemens.com
Dinler	Namik Ozkan	Turkey	ozkan.dinler@siemens.com
Coombes	Mark	United Kingdom	mark.coombes@siemens.com
Pollack	Christopher	USA	chris.pollack@siemens.com
Phuc	Ta Hong	Vietnam	ta-hong.phuc@siemens.com

Homepage:

For further information please visit ...

CNC4you-Portal (<http://siemens.com/cnc4you>)

Technical online documentation

(<https://support.industry.siemens.com/cs/document/109476679/technical-online-documentation-for-sinumerik-sinamics-simotion-and-simotics?dti=0&lc=en-WW>)

Table of contents

	Preface	3
1	Compact overview	9
2	System overview	13
2.1	SINUMERIK 828D	14
2.1.1	Data storage - SINUMERIK 828D	15
2.2	SINUMERIK 840D sl	16
2.2.1	Data storage - SINUMERIK 840D sl	17
2.2.2	Panels.....	18
3	CNC operation with SINUMERIK Operate	23
3.1	Animated Elements.....	23
3.2	Onboard documentation.....	24
3.3	Multitouch operation	25
3.3.1	Multi-touch operation, basic configuration	25
3.3.2	Multitouch operation with sidescreen	26
3.3.3	SINUMERIK Operate Display Manager	27
3.4	Shortcuts	28
4	CNC operation in manual mode (JOG).....	29
4.1	TSM universal cycle.....	29
4.2	Work offsets	30
4.3	Measure workpiece.....	31
4.4	Measure tool	32
4.5	Stock removal cycle	33
4.6	Positioning cycle	34
4.7	Manual machine	35
5	Tool management	37
5.1	Tool table	37
5.2	Monitoring of tool life and workpiece count	38
5.3	Replacement tools	39
5.4	Setup data	40
6	Data management	41
6.1	Program manager	41
6.2	Ethernet networking	42

7	CNC operation in automatic mode (AUTO).....	43
7.1	Block search	43
7.2	Program control.....	44
7.3	Execution from external storage devices	45
7.4	Basic block display	46
7.5	Simultaneous recording	47
7.6	Logging measurement results in automatic operation	48
8	CNC functionalities.....	49
8.1	80-bit NANO floating-point accuracy	49
8.2	Block change times	50
8.2.1	SINUMERIK 828D	50
8.2.2	SINUMERIK 840D sl	50
8.3	Jerk limitation.....	51
8.4	Dynamic feedforward control.....	52
8.5	Adaptive Control & Monitoring (ACM)	53
8.6	Contour handwheel	55
9	CNC programming methods	57
9.1	programGUIDE DIN/ISO and SINUMERIK high-level language	58
9.1.1	Introduction	58
9.1.2	Program editor	59
9.1.3	Languages	60
9.1.4	programGUIDE input support	61
9.2	ShopTurn machining step programming	62
9.2.1	Introduction	62
9.2.2	Sequence editor	63
9.2.3	Interlinking of sequences	64
9.2.4	Graphical view	65
10	Workpiece visualization.....	67
10.1	2D simulation	67
10.2	3D simulation	68
11	CNC technology cycles	69
11.1	CNC technology cycles for programGuide and ShopTurn	69
11.2	Highlights of machining cycles.....	70
11.2.1	Stock removal along contour with blank contour	70
11.2.2	Engraving cycle	71
11.2.3	Counterspindle cycle.....	72
11.3	Residual material detection for contour cycles	73
11.4	In-process measuring for workpieces and tools.....	74

12	Complete machining	75
12.1	End face machining (TRANSMIT)	75
12.2	Peripheral surface machining (TRACYL)	76
13	Multi-channel machining	77
13.1	Overview	77
13.2	programSYNC job list	78
13.3	Double editor.....	79
13.4	Balance cutting (stock removal)	80
13.5	Simulation	81
14	Automation.....	83
14.1	SINUMERIK Run MyRobot /EasyConnect.....	83
14.2	SINUMERIK Run MyRobot /Handling.....	84
14.3	SINUMERIK Run MyRobot /Direct Handling.....	85
15	Digitalization	87
15.1	Digitalization - Overview	87
15.2	Manage MyResources /Tools	89
15.3	Manage MyResources /Programs	90
15.4	Analyze MyPerformance (in line).....	91
15.5	Analyze MyPerformance /OEE Monitor (MindSphere).....	92
15.6	Analyze MyPerformance /OEE Tuning (MindSphere)	93
15.7	Manage MyMachines	93
15.8	Manage MyMachines /Remote	94
15.9	Optimize MyMachining /AC AUTO	95
15.10	SINUMERIK Edge & Applications	96
15.10.1	SINUMERIK Edge	96
15.10.2	Optimize MyMachining /Trochoidal	97
15.10.3	Analyze MyWorkpiece /Capture	98
15.10.4	Analyze MyWorkpiece /Monitor	99
15.10.5	Analyze MyMachine /Condition	100
15.10.6	Analyze MyWorkpiece /Toolpath.....	101
16	Tools and information	103
16.1	DXF reader	103
16.2	SinuTrain for SINUMERIK Operate.....	104
16.3	CNC4you	106

17	Safety functions.....	107
17.1	SINUMERIK Safety Integrated	107
17.2	Collision avoidance	108
17.2.1	Protect MyMachine /Axis Lock	108
17.2.2	Protect MyMachine /3D Primitives	109
17.2.3	Protect MyMachine /3D STL.....	110
17.2.4	Protect MyMachine /Open.....	111
	Index.....	113

Compact overview

Siemens Machine Tool Systems - a strong partner for the machine tool world ...

Siemens Machine Tool Systems portfolio

- ✓ Turning is easy for all SINUMERIK CNCs. Dependent on the scope of functions and performance required, and the number of axes, a suitable SINUMERIK is available for every application. Machining with main and counterspindle, Y-axes and driven tools are optimally supported by the extensive turning and milling cycle package for SINUMERIK 828D and SINUMERIK 840D sl controls. (Page 13)



User-friendliness - effective operation like on a PC

- ✓ SINUMERIK Operate offers a high degree of user-friendliness that is otherwise only expected from personal computers. SINUMERIK Operate thus sets the standard for the efficient operation of machine tools. (Page 23)



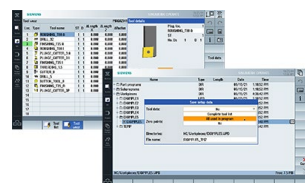
Setup functions "Intelligent-JOG"

- ✓ SINUMERIK Operate sets standards for these "functions of everyday life". Thanks to an intelligent JOG mode and intuitive tool management, all the typical setup functions feature interactive, graphical support. The "Manual Machine" function offers some highlights for cycle-controlled turning machines. (Page 29)



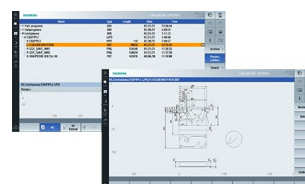
Tool management - powerful but nevertheless easy to use

- ✓ SINUMERIK, as the preferred CNC for series production, offers powerful tool management. Thanks to SINUMERIK Operate, tool management is also "easy to use" for operation sequences in the production of individual parts and small series. (Page 37)



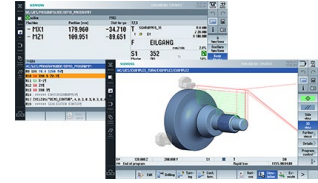
Data management like on a PC

- ✓ SINUMERIK Operate offers a modern program management system that makes the functions and user-friendliness of PC operating systems available in CNCs for the first time (Page 41)



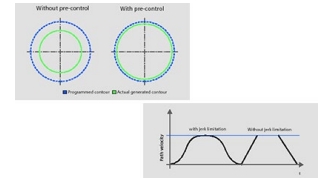
CNC operation in automatic mode (AUTO)

- ✓ SINUMERIK Operate offers numerous functions for the AUTO mode - from execution from external memories, block search and program control all the way to logging of measurement results. (Page 43)



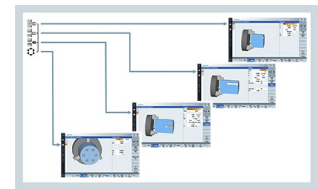
SINUMERIK CNC performance - the benchmark in all aspects

- ✓ SINUMERIK CNCs set standards in all aspects of machining performance – maximum accuracy while at the same time protecting the mechanical system of the machine. (Page 49)



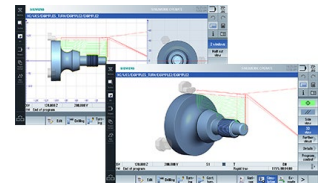
CNC programming methods - optimally prepared for all production tasks

- ✓ A major advantage of SINUMERIK are two CNC programming methods that are well established on the market: AV-based, highly productive DIN/ISO programming as well as a workshop-based workstep programming. This gives you unparalleled flexibility. (Page 57)



Workpiece visualization - more safety through simple and fast control

- ✓ Realistic 2D and 3D simulations offer reliable programming and quotation pricing. (Page 67)



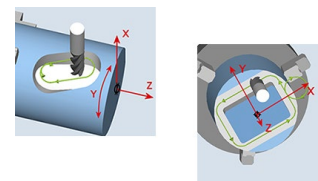
CNC technology cycles - the little helpers for daily CNC programming

- ✓ Irrespective of whether you use programGUIDE or ShopTurn – in either case the full range of technological cycles, position patterns and geometries is available to you. (Page 69)



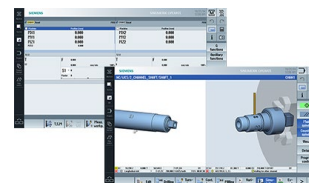
Complete machining

- ✓ No matter whether the face or cylinder surface of turned parts is to be machined: The machining plane is transformed at exactly the right position with TRANSMIT and TRACYL. And that is all completely automatic, thanks to the NC functionality in SINUMERIK Operate. (Page 75)



Multi-channel machining - efficient programming

- ✓ The CNC has to be flexible and powerful to handle large stock removal volumes and short cycle times for mass production. SINUMERIK Operate facilitates efficient programming of multi-channel machining. The result is greater reliability by means of 3D simulation with the 3-plane view and extensive control and optimization possibilities through the automatic calculation of machining time. (Page 77)



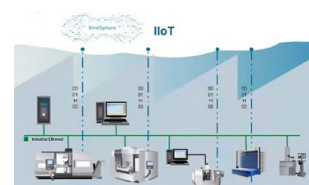
Automated cell

- ✓ Robots must be easy to integrate into machine tools and production processes. SINUMERIK Run MyRobot offers solutions ranging from simple connecting and user-friendly integration for handling tasks up to high-precision motion control of machines with robot kinematics. (Page 83)



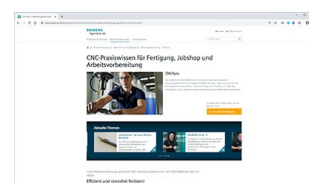
Digitalization on the shop floor

- ✓ Digitalization is clearly a domain of Siemens, not only with powerful IT solutions for SINUMERIK. The strength of Siemens Digital Industries is the digitalization of the entire shop floor. (Page 87)



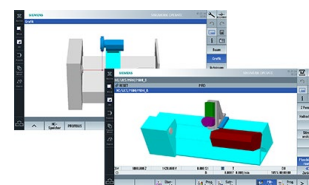
Tools and information

- ✓ The useful helpers - DXF Reader and SinuTrain for SINUMERIK! On the information platform CNC4you you will find helpful tips & tricks and a download area. (Page 103)



Safety functions

- ✓ SINUMERIK Safety Integrated permits the unrestricted movement of the machine in set-up mode with open protective doors, thus offering the machine user a significant plus in terms of user friendliness. Collision avoidance functions provide protection against collisions in the workspace. (Page 107)



System overview

SINUMERIK 828D and SINUMERIK 840D sl controls, and the easily understandable and intuitive SINUMERIK Operate programming interface provide a tailored solution for all CNC turning machines used worldwide.

SINUMERIK Operate

Characteristic features of SINUMERIK Operate include...

- HMI-Advanced, ShopMill and ShopTurn combined under one interface
- Intuitive and clear operation and programming, including Animated Elements
- Display in the modern Windows style
- New powerful functions around setting up, programming, tool and program management
- New functions for complete machining (single-chuck machining)

Two options are available for the **programming**:

- DIN-ISO programming with programGuide (CNC text editor with programGuide cycle support, and DIN-ISO and readable CNC high-level language commands) for mid-sized and large series
- ShopTurn machining step programming with graphical interactive CNC machining step editor and CNC programming without DIN-ISO knowledge for small series

programSYNC – Efficient programming for multi-channel machines

Turning-milling centers with several saddles are considered to be the high-end machines in this segment. The individual tool carriers are distributed and managed by SINUMERIK in different channels. Programs must be created for each channel that then run simultaneously later during the machining. With the uniform SINUMERIK Operate user interface, Siemens provides a standard user interface which enables programs for two channels to be created simultaneously by means of a double editor and to align them with the **programSYNC** function. In this way, efficient programming is possible directly on the control.

2.1 SINUMERIK 828D

The CNC performance variants PPU 270.4/PPU 271.4 and PPU 290.4 of the SINUMERIK 828D can be flexibly combined with the software variants described below.

This allows you to adapt the SINUMERIK 828 perfectly to the power requirements of the respective machine concept.



Software variant 28x

- Up to 10 axes/spindles (turning and G tech)
- Up to 2 machining channels (T, M, G)
- 768 tools, 1536 cutting edges
- 10 MB user memory
- Additionally up to 2 auxiliary axes

Software variant 26x

- Up to 6 axes/spindles
- 1 machining channel
- 256 tools, 512 cutting edges
- 5 MB user memory
- Additionally up to 2 auxiliary axes

Software variant 24x

- Up to 5 axes/spindles
- 1 machining channel
- 128 tools, 256 cutting edges
- 3 MB user memory



You can find further information in catalog NC 82

Benefits



- Improved efficiency thanks to state-of-the-art operating technologies and functions
- Scalable solutions thanks to tailored hardware and software for the compact class

2.1.1 Data storage - SINUMERIK 828D

Internal memory				External storage
	828D SW 24x	828D SW 26x	828D SW28	Execution from external storage (EES) (option P75*) Network, USB storage media, compact flashcard
	USB / CF card can be used for data transport or for execution with EXTCALL			
Option P77	-	100 MB		
SW	3 MB	5 MB	10 MB	Execution from the CNC expanded user memory (option P77)
Internal memory can be expanded via option P77 → 100 MB				External storage via option P75* → can be expanded almost without limit

* Option P75 not available for SW 24x

2.2 SINUMERIK 840D sl

SINUMERIK 840D sl is an open CNC for modular premium machine concepts. With powerful, innovative system functions, the SINUMERIK 840D sl opens up a boundless range of technologies. SINUMERIK 840D sl is leading the way in exploiting global machining trends; this makes it the preferred CNC in the industries of the future.



- Drive-based modular CNC
- Multi-technology CNC
- Up to 93 axes/spindles
- Up to 30 machining channels
- Modular panel concept up to 19" color display
- SIMATIC S7-300 PLC



You can find further information in catalog NC 62

Benefits



- Increased productivity of the machines thanks to faster controls and innovative machine concepts
- Improved efficiency for operation thanks to state-of-the-art operating technologies and functions
- Improved quality by perfectly adapting the control to the machine behavior
- Simplified engineering thanks to additional system support for configuring, testing and optimizing
- Future-oriented expansions for digitalization and integration in automation concepts

2.2.1 Data storage - SINUMERIK 840D sl

Internal memory					External storage
	NCU	NCU	NCU	NCU + PCU	
Option P77 + PCU				up to 40 GB	
Option P77 + option P12 ¹⁾			up to 6 GB		
Option P77		100 MB			
					Execution from external storage devices (EES, option P75) Network, USB storage media, compact flashcard
CNC user memory (option D00)	NCU 710.3B: 10 to 16 MB NCU 720.3B and NCU 730.3B: 10 to 22 MB				
Internal memory can be expanded via option P77 + PCU → 40 GB Internal memory can be expanded via option P77 + P12 → 6 GB Internal memory can be expanded via option P77 → 100 MB					Execution from the CNC expanded user memory (option P77)
					External storage via option P75 → can be expanded almost without limit

1) HMI user memory, alternative to PCU

2.2.2 Panels

SINUMERIK OP 08T



- Operator panel 191 mm wide, 7.5" TFT display (resolution 640 × 480 pixels)
- Integrated CNC keyboard with 75 keys (layout as for the SINUMERIK full CNC keyboard)
- With USB interface at the front
- Version with membrane keys

SINUMERIK OP 010



- Operator panel 483 mm wide, 10.4" TFT display (resolution 640 × 480 pixels)
- Integrated CNC keyboard
- With USB interface for a memory stick at the front
- Version with **membrane keys**
- Separate machine control panel

SINUMERIK OP 010S



- Operator panel 310 mm wide, 10.4" TFT display (resolution 640 × 480 pixels)
- Mechanical keys
- With USB interface for a memory stick at the front
- Separate CNC keyboard and machine control panel

OP 010C



- Operator panel 483 mm wide, 10.4" TFT display (resolution 640 × 480 pixels)
- Integrated CNC keyboard
- With USB interface for a memory stick at the front
- Version with mechanical keys
- Separate machine control panel

OP 012



- Operator panel 483 mm wide, 12" TFT display (resolution 800 × 600 pixels)
- Membrane keys
- Integrated mouse
- Touchpad
- With USB interface for a memory stick at the front

SINUMERIK OP 015A



- Operator panel 380 mm wide, 15" TFT display (resolution 1024 × 768 pixels)
- Version with membrane keyboard with 62 keys
- With USB interface at the front
- Integrated mouse

SINUMERIK OP 015 black



- Operator panel 396 mm wide, 15.6" TFT display (resolution 1366 × 768 pixels)
- Capacitive keyboard with 64 keys
- Capacitive display area for gesture operation (touch operation)

Note: see also Chapter AUTOHOTSPOT

SINUMERIK OP 019



- Operator panel 483 mm wide, 19" TFT display (resolution 1280 × 1024 pixels)
- Version with membrane keys, gloved operation also possible
- Capacitive sensor equipment for fast key operation
- Integrated key disable as protection against incorrect operation
- USB 2.0 connector socket for console installation
- Separate CNC keyboard and machine control panel










SINUMERIK OP 019 black



- Operator panel 46.99 cm wide, 18.5" TFT display (resolution 1366 × 768 pixels)
- Permits the distributed installation of the operator panel front and the controller
- Capacitive display area for gesture operation

Note: see also Chapter AUTOHOTSPOT

SINUMERIK blackline plus

SINUMERIK blackline plus				
Panel size	15"	19"	22"	24"
				
Width	398 mm	464 mm	529 mm	585 mm
SIMATIC ITC Industrial Thin Client	X	X	X	(X)*
SIMATIC IFP Industrial Flat Panel (monitor)	X	X	X	X
SIMATIC IPC 477E Industrial PC (Integrated Panel PC)	X	X	X	X
NCU 710 / 720 / 730	X	X	X	X
SINUMERIK MCP 398 + EM				
SINUMERIK ONE MCP	--			

*) available as a customer-specific version

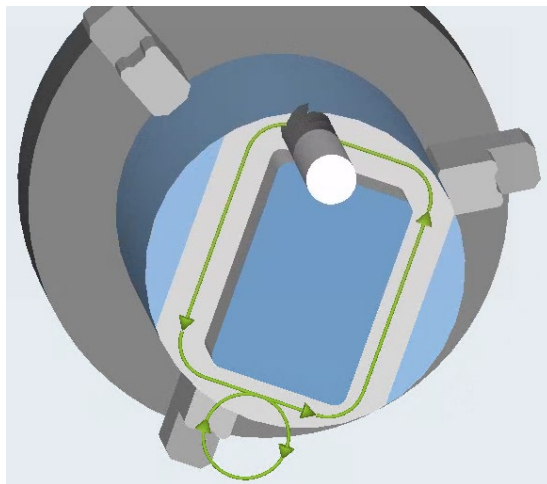
CNC operation with SINUMERIK Operate

3.1 Animated Elements

<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

<input checked="" type="checkbox"/>	SINUMERIK 840D sl
	Basic configuration

To illustrate which parameters affect what in machining operations, SINUMERIK Operate offers a new input support function with animated element sequences.



Benefits



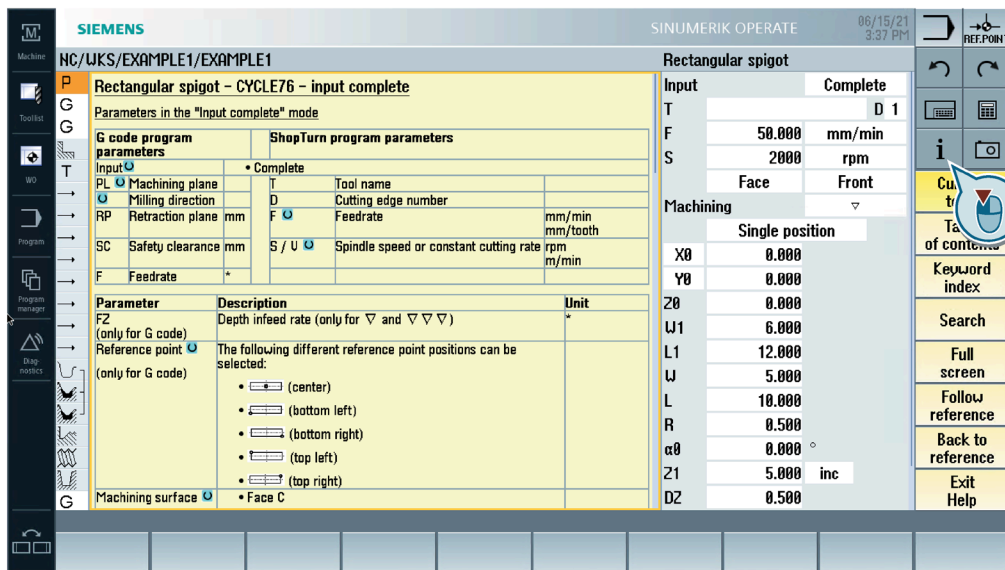
- Process reliability during the setup
- Increased reliability at program input by easily understood depiction of selection options
- This results in improved efficiency and so increased availability of the machine

3.2 Onboard documentation

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sI
	Basic configuration

For each input field in the operating screens, SINUMERIK Operate automatically displays help in the form of a "cursor text". Further information is provided in the form of a complete context-sensitive help system with many useful details and graphics.






Benefits



- Programming on the machine without a handbook
- Help button to toggle between the editor and help screens

3.3 Multitouch operation

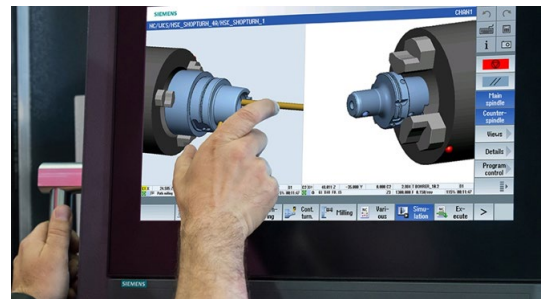
3.3.1 Multi-touch operation, basic configuration

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration






	SINUMERIK 840D sl
	Basic configuration

With the appropriate operator panel fronts, SINUMERIK Operate can also be operated with multitouch gestures. Multitouch operation is possible for the SINUMERIK 840D sl with the operator panel fronts SINUMERIK OP 015 black line or SINUMERIK OP 019 black line and for the SINUMERIK 828D PPU 290.4, vertical.

- Intelligent gesture operation with touchpanels, also with work gloves
- Capacitive touch for industrial use
- Palm detection
- Detection of liquids and contaminations



Extract from the multitouch operation gestures:




				
Tap with two fingers Call the shortcut menu, e.g. copy, paste	Tap and hold Open object to be changed, e.g. NC block	Pan Move graphic contents, e.g. simulation, mold making view	Flick with three fingers Scroll to the start or end of lists or files	Spread Zoom out graphic contents, e.g. simulation, mold making view

Benefit



- Modern and efficient gesture operation of SINUMERIK Operate – rugged and reliable, even in harsh industrial environments

3.3.2 Multitouch operation with sidescreen

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

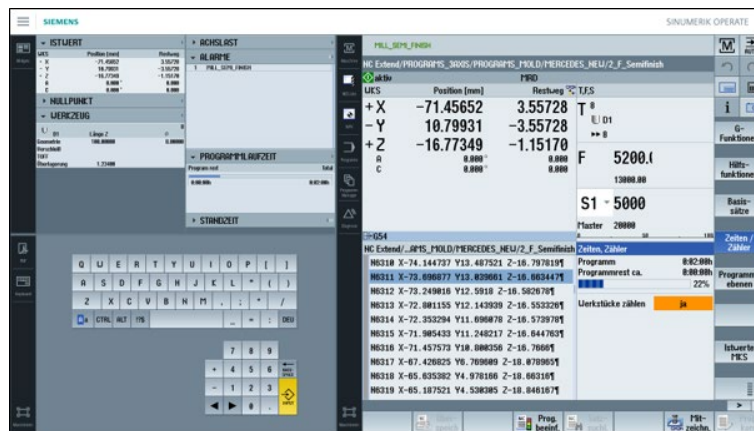
	SINUMERIK 840D sl
	Basic configuration

With sidescreen you can integrate widgets and pages. The additional windows can be closed and opened and placed either on the left or right side of the screen. The sidescreen can be opened and closed.

You can integrate the following standard widgets.

- NC/PLC variables
- Actual value
- Zero point
- Alarms/messages
- Axis load
- Current tool
- Tool life
- Program runtime

The ABC keyboard, as an alternative to the virtual QWERTY keyboard, or the machine control panel functions can be integrated as pages.



Precondition: Only for Panels with a resolution of 1366x768 or a full HD resolution of 1920x1080

Benefit



- All information in view in every operating situation and thus permanent control of the machine status.

3.3.3 SINUMERIK Operate Display Manager

<input type="checkbox"/>	SINUMERIK 828D SW24x	<input type="checkbox"/>	SINUMERIK 828D SW26x	<input type="checkbox"/>	SINUMERIK 828D SW28x
	not available		not available		not available

<input checked="" type="checkbox"/>	SINUMERIK 840D sl				
	Option: P81				

With the Display Manager, the machine operator has the possibility to individually adapt the user interface to machines and individual requirements.



Partitioning of the display area into three or four areas.

Example:

- 1: SINUMERIK Operate
- 2: Standard widgets
- 3: Applications (PDF, keyboard, etc.)
- 4: Virtual keyboard (optional)



- Direct switching between left and right orientation
- Sidescreen widgets can continue to be used in the Display Manager
- Customized Windows applications
- Machine control panel/virtual keyboard
- Temporarily maximizing the display area




Precondition: only for Panels with a full HD resolution of 1920x1080


Benefit




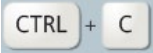


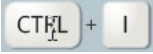
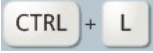


- Effective use of large screens with individually configurable contents.

3.4 Shortcuts

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

Shortcuts are available for many menu operations in SINUMERIK Operate. A small extract follows:

	Select all (editor functionality)
	Copy
	Paste
	Cut
	Calculation of the time from/to line/block
	Language selection
	Maximum simulation speed
	For screenshots (storage location: commissioning (keyword) → System data → HMI data → Logs → Screenshots)

Benefit



- Shortcuts in SINUMERIK Operate avoid the need for complicated menu operations and provide functions not previously expected from a CNC

CNC operation in manual mode (JOG)

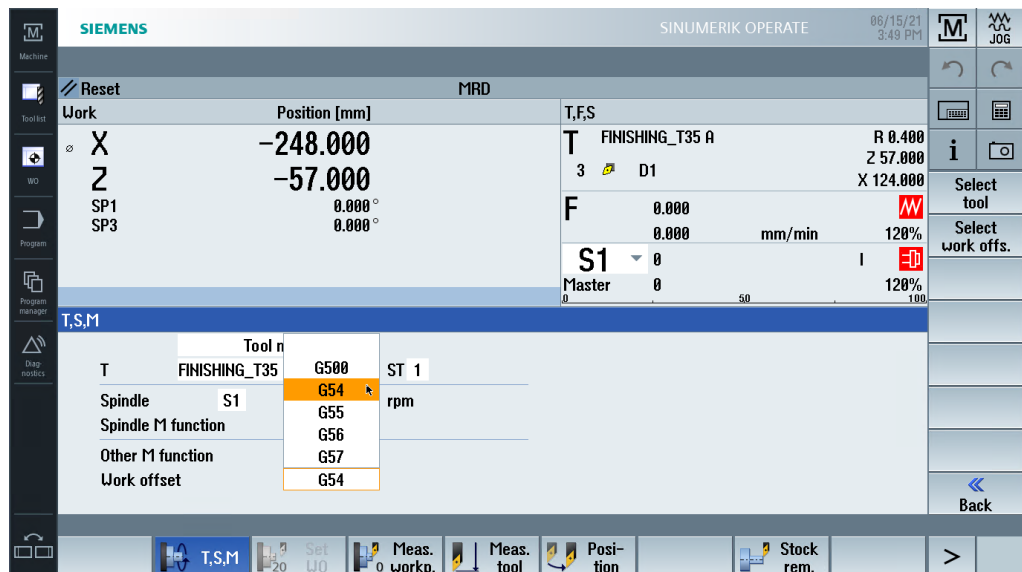
4.1 TSM universal cycle

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

A universal cycle is available in the setup for the most commonly used machine functions:

- Tool change, also involving sister tools, with direct access via the tool table (T)
- Spindle speed and direction (S)
- M functions (M)
- Activation of work offsets
- Definition of the gearbox stage



Benefit



- Take over and change in tools directly from the tool table

4.2 Work offsets

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

The following work offsets are possible:

- **Settable work offsets:**
It is possible to enter as many as 100 work offsets (G54 to G57, G505 to G599), offset coordinates, angles and scaling factors.
- **Programmable work offsets:**
The programmable work offsets allow you, for example, to work with different work offsets for repetitive machining operations at different positions on the workpiece.
- **External work offsets:**
Axis-related linear work offsets can also be activated via the PLC user software.

Work offset - active [°]		X	Z	SP1	SP3
Act.value Machine		0.000	0.000	0.000	0.000
G500		0.000	0.000	0.000	0.000
Total WO		0.000	0.000	0.000	0.000
Tool: FINISHING_T35 R		124.000	57.000		
Work actual value		-248.000	-57.000	0.000	0.000

Benefits



- Flexible machining thanks to a large number of settable work offsets
- User-friendly understandable display of the number of work offsets

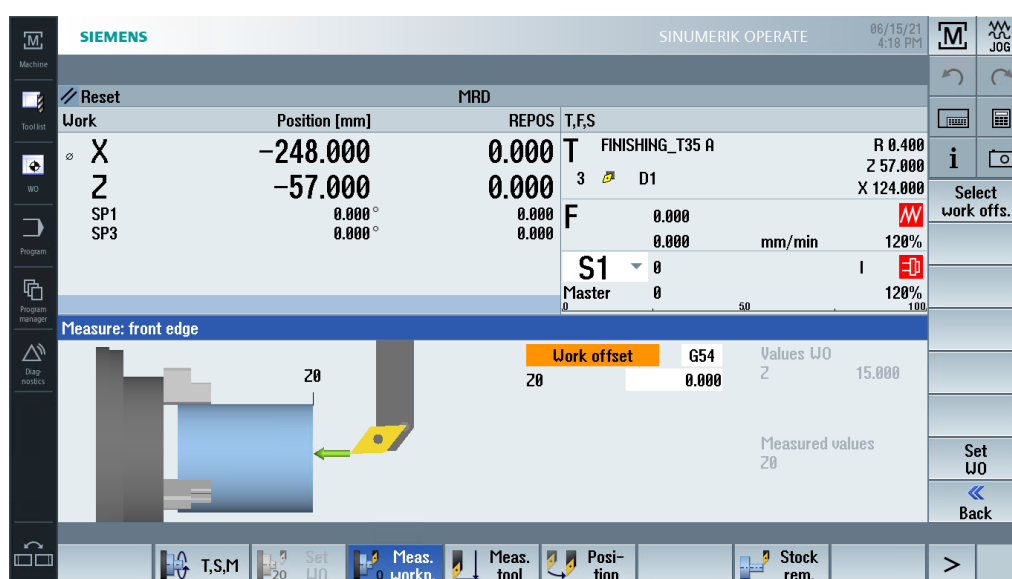
4.3 Measure workpiece

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

The workpieces can be measured as follows:

- Reference tool



Benefit



- Time saving due to user-friendly determination of the workpiece zero

4.4 Measure tool

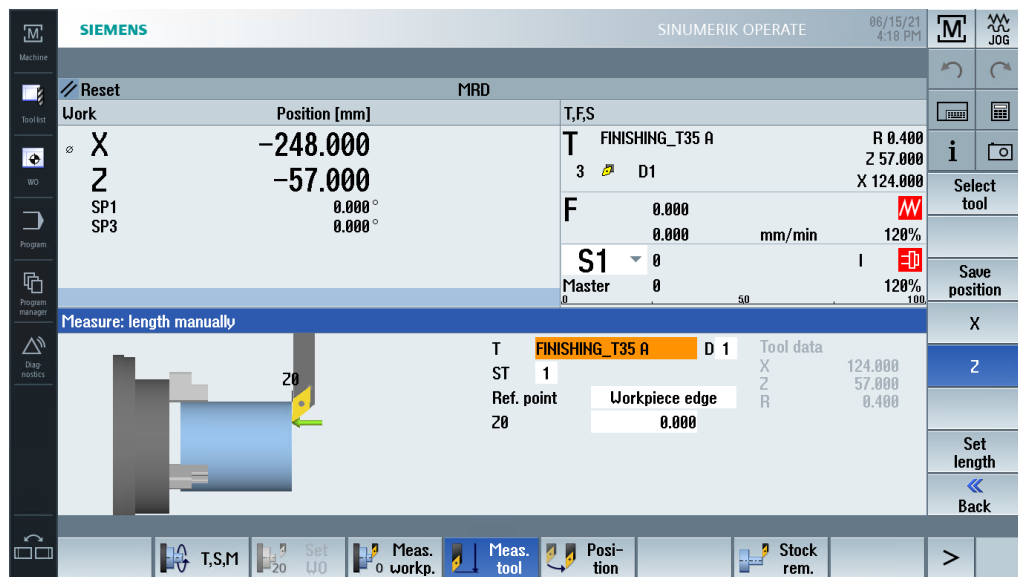
<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration
<input checked="" type="checkbox"/>	SINUMERIK 840D sl				
	Basic configuration				

The tool compensation value can be directly determined in the machine set-up.

The following variants are supported:

- Touch chuck
- Determine lengths via reference diameter
- Tool measuring probe (tooleye) or magnifier

The measurement results can be output in a measurement report (see AUTOHOTSPOT).



Benefit



- User-friendly functions for determining the tool dimensions directly in the machine

4.5 Stock removal cycle

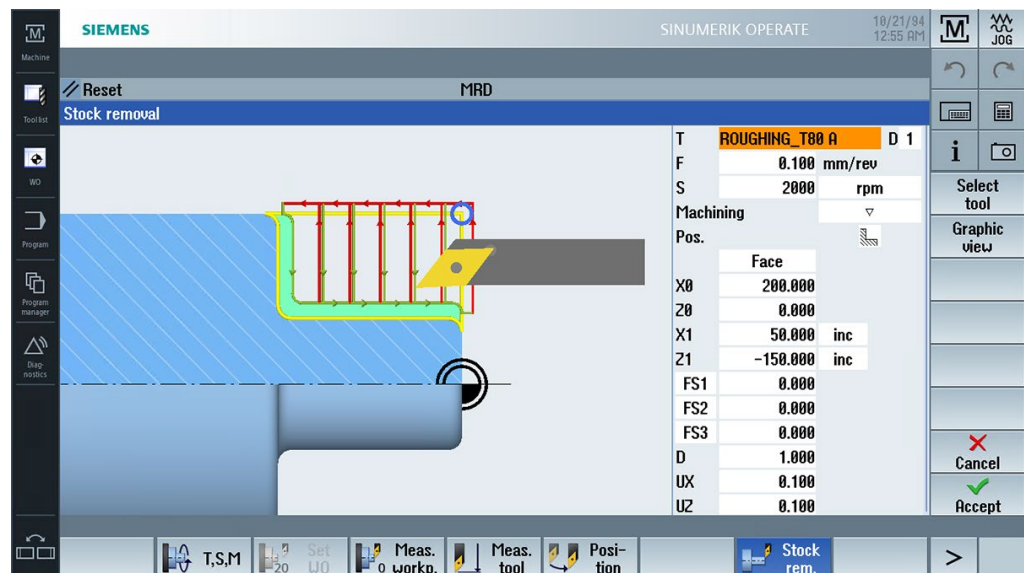
<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

<input checked="" type="checkbox"/>	SINUMERIK 840D sl
	Basic configuration

A comfortable stock removal cycle is available in the set-up mode. Soft collet chucks can, for example, be turned with this cycle.

The following parameters can be specified:

- Roughing or finishing
- Undercut for soft collet chucks



Benefit



- Preparation of workpiece or collet chuck without having to create a part program

4.6 Positioning cycle

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

The machine axes can be positioned directly via input screens in the setup:

- Linear axes / spindles
- Feedrate / rapid traverse

The screenshot shows the SINUMERIK OPERATE manual mode (JOG) screen. The top bar displays 'SIEMENS' and 'SINUMERIK OPERATE'. The main area is divided into several sections:

- Reset**: A button to reset the machine.
- Work**: A table showing the current position of the X and Z axes. The X-axis is at -248.000 mm and the Z-axis is at -57.000 mm. The target position is set to 0.000 mm for both axes.
- Position [mm]**: A table showing the current position of the X and Z axes. The X-axis is at -248.000 mm and the Z-axis is at -57.000 mm.
- T.F.S**: A table showing the feedrate and rapid traverse settings. The feedrate is set to 0.000 mm/min and the rapid traverse is set to 120% of the feedrate.
- Target position**: A table showing the target position for the X and Z axes. The target position is set to 0.000 mm for both axes.
- Diagram**: A diagram of the X-Z coordinate system showing the current position and target position.
- Rapid traverse**: A button to activate rapid traverse.

Benefit



- Simple axis positioning without manual input, directly over the dialog screen

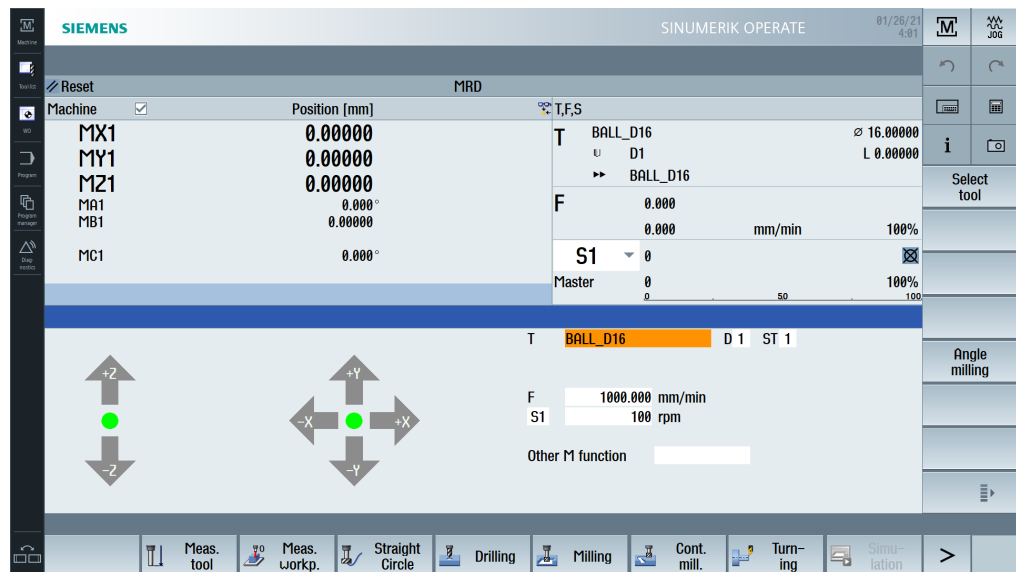
4.7 Manual machine

<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Option: P17		Option: P17		Option: P17
<input checked="" type="checkbox"/>	SINUMERIK 840D sl				
	Option: P17				

The Manual Machine function is part of the ShopMill/ShopTurn option package. This allows you to perform all important machining operations in the manual machine operating area without needing to create a specific part program.

The following functions are available:

- Measure tool
- Traversing axes
- Setting the work offset (WO)
- Setting the endstop
- Turning a straight line / circle
- Drilling, including centering, deep-hole drilling, tapping
- Milling, including face milling, pocket, multiple edge spigot






Benefits




- Simple and intuitive operation of cycle-controlled turning machines

Tool management

5.1 Tool table

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

Tools with their complete operating data can be managed in the tool list.

- Tools are assigned to the desired magazine locations with the load function.
- For each tool, you can store the following data:
 - Tool type (rougher, finisher, engraver, drilling and milling tools)
 - Tool name in plain text (ex.: "ROUGHING_80DEGREES")
 - Max. of 9 cutting edges per tool
 - Tool length and cutting plate geometry
 - Nose angle for drills or number of teeth for milling tools
 - Direction of spindle rotation and coolant (level 1 and 2)
- Direct transfer of the tool from the list in the program or for measurement
- Using the settings, for example, you can activate the graphic magazine display
- Reading tools from a file or archiving to a file

Benefits



- All tool data at a glance
- Simple and secure handling via unmistakable tool names

5.2 Monitoring of tool life and workpiece count

<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

<input checked="" type="checkbox"/>	SINUMERIK 840D sl
	Basic configuration

You can use SINUMERIK Operate to monitor the service life of your tools and the number of exchanges. You can give your tools meaningful names instead of cryptic numbers. You will come to appreciate this convenience when you read the CNC program, if not before.

- Monitor cutting time (T) in minutes or number of exchanges (C)
- Prewarning limit for timely preparation of new tools

The screenshot shows the SINUMERIK Operate interface for tool wear monitoring. The main window is titled 'Tool wear' and displays a table of tool data. The table has columns for 'Loc.', 'Type', 'Tool name', 'ST', 'D', 'ΔLength X', 'ΔLength Z', and 'ΔRadius'. The table lists 13 tools, including 'ROUGHING_T80 A', 'DRILL_32', 'FINISHING_T35 A', 'ROUGHING_T80 I', 'PLUNGE_CUTTER_3 A', 'PLUNGE_CUTTER_3 I', 'FINISHING_T35 I', 'THREADING_1.5', 'CUTTER_8', 'DRILL_5', 'BUTTON_TOOL_8', 'FINISHING_T35_R', and 'PLUNGE_CUTTER_3P'. The right side of the screen shows a 3D model of a tool and a 'Monitoring type' bar chart. The bottom of the screen has a navigation bar with icons for 'Tool list', 'Tool wear', 'Magazine', 'Work offset', 'User variable', 'Setting data', and 'Back'.

Benefits



- Reduction of machine standstill times via tool monitoring
- Support of tool life monitoring or job time monitoring as standard

5.3 Replacement tools

<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Option: M78		Option: M78		Option: M78

<input checked="" type="checkbox"/>	SINUMERIK 840D sl
	Basic configuration

If needed, you can also manage replacement tools with SINUMERIK Operate. Tools with the same name are created as replacement tools. The replacement tools are identified with an incrementing number in the ST column.

Benefit



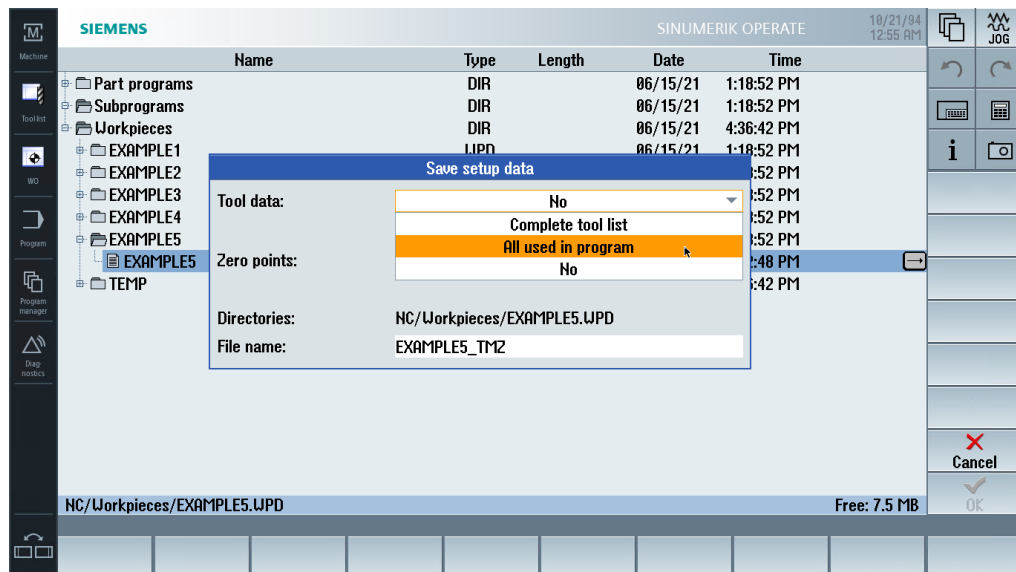
- Automatic exchange of identical tools for unmanned operation

5.4 Setup data

<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Option: P16		Option: P16		Option: P16

<input checked="" type="checkbox"/>	SINUMERIK 840D sl
	Basic configuration

Part programs can be saved complete with set-up data like tool data and zero points.






Benefit




- Time savings when you save the part programs

Data management

6.1 Program manager

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

The program manager provides an optimum overview of the directories and programs, and very easy-to-use file handling similar to Windows Explorer.




- Plain text names with as many as 24 characters for directories and files
- Manage subdirectories on external storage media, local drives and on the NC
- Store and display files of any type (e.g. *.png, *.pdf, *.dxf, *.xml)
- Manage and open DXF files
- Display all storage media in the program manager (with details of the storage capacity), including the network drives.
- Part programs can be edited on all media


Benefits



- Easy and open exchange of data between the various storage media and the network
- User-friendly data handling in typical PC style with copy, paste, rename, etc.
- Preview window permits quick identification of programs without having to open them

6.2 Ethernet networking

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: P01		Option: P01		Option: P01

	SINUMERIK 840D sI
	Option: P17

The SINUMERIK controls are prepared for networking via Ethernet (TCP/IP) (RJ45 connection).

- The data transfer rate is 10/100 Mbit/s.
- Remote access to the control via the RCS Commander, e.g. for commissioning and remote diagnostics
- Access to the network drives is available directly from the program manager. No additional software is required on the server.




Benefits




- **Cost-effective and simple connection via Ethernet (TCP/IP) to Windows PCs**
- **No software required on the servers**

CNC operation in automatic mode (AUTO)

7.1 Block search

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

A block search may be executed in machine status RESET, e.g. after a program interruption or to specifically return to machining. The program data is prepared in such a way that all relevant parameters (tool, work offsets, M functions, etc.) are available when accessing the program.

The following search variants are available:

- selectively to the point of interruption, still possible after Power Off for G code program
- to arbitrary CNC blocks in DIN/ISO programs
- in arbitrary subprogram levels for DIN/ISO programs
- in ShopTurn machining step programs
- in position patterns for machining step programming

You can individually configure the block search:




- with calculation/without calculation
- with approach/without approach


Benefits



- Time-saving and secure restart at any program point, as no editing of the part program is required
- An extremely quick block search is also available for large part programs through the "External block search without calculation" function; overstore, if necessary

7.2 Program control

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

You can influence the program sequence in the AUTO and MDI modes. The following options are available to do this:




- **PRT** – No axis motion
The program is completely executed with the axes stationary, e.g. for the program test.
- **DRY** – Dry run feedrate
The traversing velocities programmed in conjunction with G1, G2, G3, CIP and CT are replaced by a defined dry run feedrate.
- **RG0** – Reduced rapid traverse
You define the reduced rapid traverse in the settings for automatic operation.
- **M01** – Programmed stop 1
The processing of the program stops at every block in which supplementary function M01 is programmed. In this way you can check the intermediate result when machining a workpiece.
- **SKP** - Skip block
Skip blocks are skipped during machining.
- **MRD** - Displaying measurement result
The display of the measurement result can be enabled or disabled during program execution.

Benefits






- **Secure positioning of new part programs**
- **Continue machining quickly after interruptions**

7.3 Execution from external storage devices

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

You can select, edit and execute part programs directly on the CF card, USB stick, hard disk or via the network.

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	not available		Option: P75		Option: P75

	SINUMERIK 840D sl
	Option: P75

The "Execution from external storage (EES)" option provides the following advantages over the basic configuration:

- Uniform syntax for the subprogram call, independent of the storage location of the subprogram. This reduces syntax errors for the subprogram call.
- Part programs can be edited without NC reset.
- The size of the memory available on the machine can be expanded economically with external media. The size of the part programs is limited only by the capacity of the external data storage.

Benefits



- Quick and easy access to part programs on external storage media
- Block search for large programs on external storage media

7.4 Basic block display

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration
	SINUMERIK 840D sl				
	Basic configuration				

The individual traversing blocks are displayed as DIN/ISO commands during execution of machining steps or machining cycles.

The basic block display guarantees an especially high process security while running-in programs in single block mode.

This function is available for programGUIDE (screenshot on left) and also for ShopTurn (screenshot on right).



Benefit

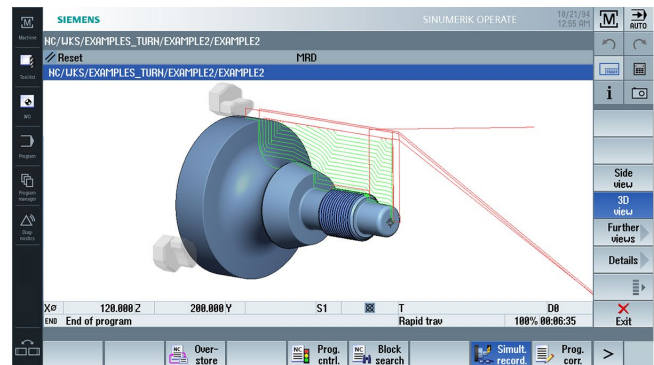
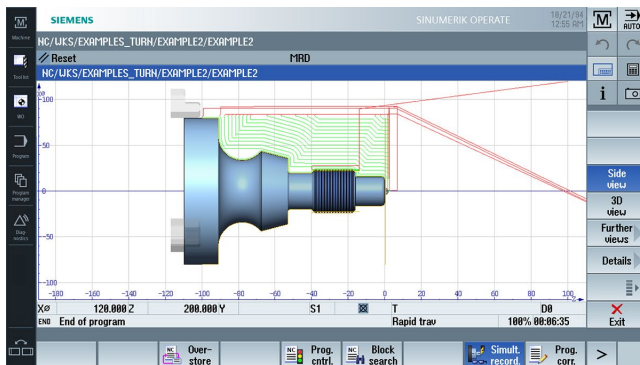


- Optimal control of the program execution, even in complex sequences or machining cycles, especially in single block mode

7.5 Simultaneous recording

<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Option: P22		Option: P22		Option: P22
<input checked="" type="checkbox"/>	SINUMERIK 840D sl				
	Option: P22				

During machining, the tool paths can be simultaneously recorded on the display of the controller in side view, front view, two window view or in 3D view. Workpiece depiction and views correspond to the graphic simulation.







Benefit



- Machining can also be monitored in a complex machine room

7.6 Logging measurement results in automatic operation

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

In automatic operation, you can output the measurement results as measuring log. You can configure the output. The following settings are some of those possible:

- Display mode: autom. 8 s, NC start, for alarm
- Log type: standard log, user log
- Log format: text format (*.txt), table format (*.csv)
- Log data: new (discard old log data), append (append to old log data)
- Log storage: storage directory (complete path)

You can then open the measuring log in the program management at the configured storage path. The measuring log contains data that includes:

- Date and time when the log was written
- Measuring method
- Correction target
- Setpoints, measured values and differences

Note: Irrespective of the user interface language, the measuring logs are output in English.

Benefit



- Simple logging of measured values in log files

CNC functionalities

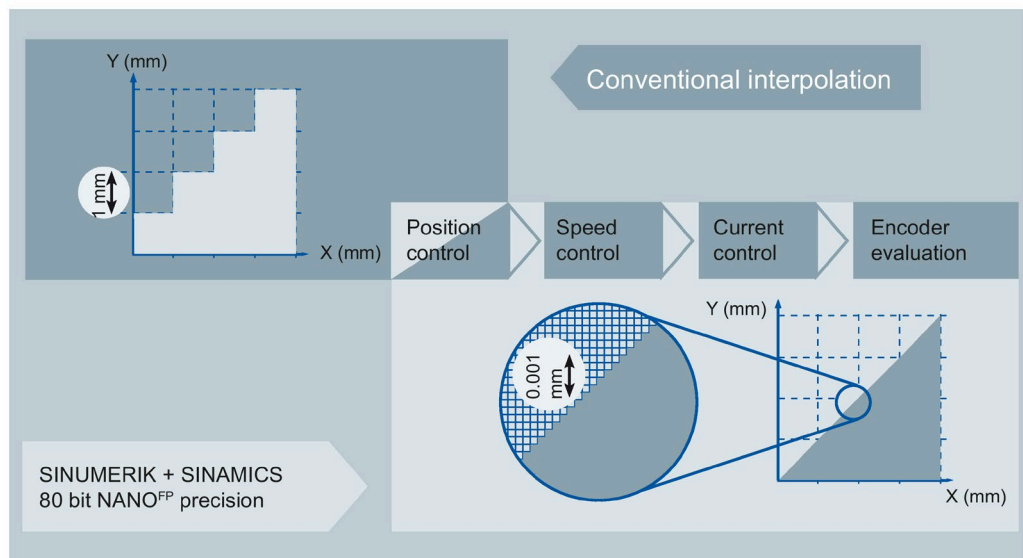
8.1 80-bit NANO floating-point accuracy

✓	SINUMERIK 828D SW24x	✓	SINUMERIK 828D SW26x	✓	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

✓	SINUMERIK 840D sl
	Basic configuration

The accuracy of the workpiece is determined by more factors than just the mechanical characteristics of the machine. The CNC also contributes to a critical degree towards the precision of the workpieces. SINUMERIK Operate offers many CNC functions for this purpose.

The SINUMERIK controls and the SINAMICS drive calculate with 80-bit NANO floating-point accuracy. This enables a calculation accuracy much less than a nanometer. This exactness is available not only for closed loop position control but also for closed-loop power and speed control and also for sensor evaluation of the drive.






Benefit



- Maximum precision in the workpiece results due to extremely high calculation accuracy

8.2 Block change times

8.2.1 SINUMERIK 828D

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

The following table shows the minimum block change times with compressors, depending on the PPU used:


PPU 270.4/PPU 271.4/PPU 290.4		
SW24x	SW26x	SW28x
~9 ms	~6 ms	~6 ms

Benefit



- Minimum block change times for the associated performance versions

8.2.2 SINUMERIK 840D sl

	SINUMERIK 840D sl
	Basic configuration

The following table shows typical block change times depending on the deployed NCU:




NCU 710.3B PN	NCU 720.3B PN	NCU 730.3B PN
1.2 ms	0.5 ms	0.3 ms

Benefit



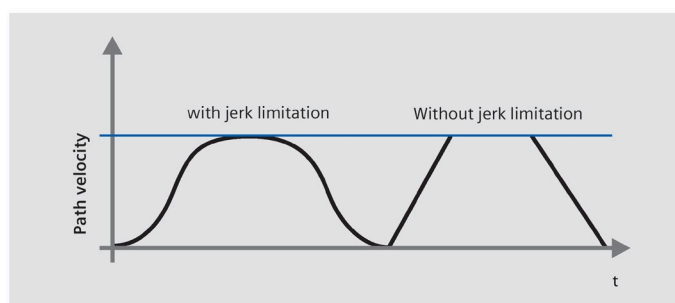
- Minimum block change times for the associated performance versions

8.3 Jerk limitation

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

The control calculates a steady acceleration profile instead of jumps in acceleration. This enables jerk-free speed characteristics for the involved path axes. The jerk limitation can also be directly activated in the part program with the »SOFT« NC language command.



Benefits



- Longer machine lifespan through protection of the mechanical components
- Higher path accuracy through softer acceleration

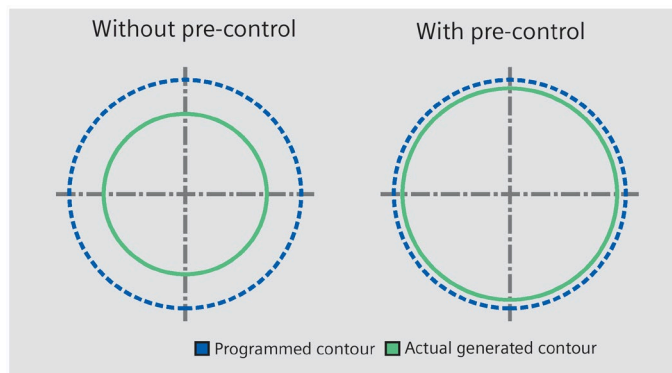
8.4 Dynamic feedforward control

✓	SINUMERIK 828D SW24x	✓	SINUMERIK 828D SW26x	✓	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

✓	SINUMERIK 840D sl
	Basic configuration

Inaccuracies in the resulting workpiece contour due to following errors can practically be eliminated using dynamic feedforward control FFWON. The result is excellent machining precision even at high path speeds. This is clarified with a circularity test on the machine.

Example:



Benefit



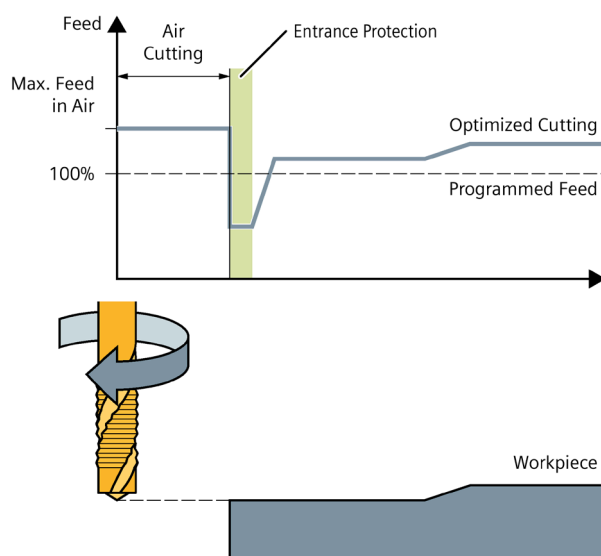
- Higher path accuracy through compensation of contouring errors

8.5 Adaptive Control & Monitoring (ACM)

✓	SINUMERIK 828D SW24x	✓	SINUMERIK 828D SW26x	✓	SINUMERIK 828D SW28x
	Option: via SISW		Option: via SISW		Option: via SISW
✓	SINUMERIK 840D sl				
	Option: via SISW				

Adaptive Control & Monitoring (ACM) monitors the current cutting conditions in real-time and automatically adjusts the feedrate to the optimum speed.

- If an overload is detected, ACM reduces the feedrate and can trigger an alarm to stop the machine.
- Detection of tool breakage to prevent consequential damage.



SINUMERIK 828D - Hardware solution

The solution consists of two components:

- ACM DAQC unit
- PTM Power Transducer Module with Current Sensor

Besides the hardware you need the software component "ACM-HMI". This is installed on an external PC/laptop computer/IPC. The software can be sourced via SIEMENS Industry Software (SISW).

Adaptations to the PLC user program are required for ACM.

SINUMERIK 840D sl - Software solution

The solution consists of two main components:

- Real-time component:
Compile Cycle Run MyCC /IMD to access the necessary data
- HMI component:
SINUMERIK Operate, based on Run MyHMI /3GL

Optional: With the "Cross-operational actions" option, the synchronous action between the compile cycle and the HMI is executed automatically.

Note

Contact SIEMENS Industry Software (SISW) for further questions regarding the products and licenses.




> Contact SISW (<https://new.siemens.com/global/en/products/automation/systems/cnc-sinumerik/digitalization/manufacturing.html>)

Benefit



- **ACM boosts productivity, extends the machine and tool life, and ensures a stable production process.**

8.6 Contour handwheel

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: M08		Option: M08		Option: M08

	SINUMERIK 840D sl
	Option: M08

With the contour handwheel function, the handwheel has a velocity-generating effect in AUTO and MDI operating modes on all programmed traversing movements of the path and synchronized axes.

A feedrate specified by the part program becomes ineffective, and a programmed velocity profile is no longer valid. The feedrate, in mm/min, results from the handwheel pulses as based on pulse weighting (machine data) and the active increment.

The handwheel's direction of rotation determines the direction of travel:

- Clockwise: In the programmed direction of travel, even beyond block boundaries
- Counter-clockwise: Opposite to the programmed direction of travel, continuation beyond the start of the block is prevented

Benefit



- **Used with conventional turning machines for setup/scratching.**
- **More user-friendly operation of the machine in setup mode**

CNC programming methods

SINUMERIK Operate provides the following programming methods for selection:

DIN-ISO programming with programGUIDE

CNC text editor with programGuide cycle support, and DIN-ISO and readable CNC high-level language commands for mid-sized and large series

The wide choice of technology cycles and the ease of parameterization allows you to reduce the programming time.

ShopTurn - Machining step programming

with graphical interactive CNC machining step editor and CNC programming without DIN-ISO knowledge for small series.

Machining operations such as stock removal, grooving or thread cutting are shown in ShopTurn in the form of worksteps. In this way CNC programs – even for complex machining operations – are very compact and easily read. Associated sequences are automatically interlinked and can be assigned any position patterns.

ShopTurn offers you the shortest programming times even for highly demanding machining tasks. The parameter input is supported by Animated Elements.




Benefit



- **Whether you use programGUIDE or ShopTurn – in either case the full range of technological cycles, position patterns and geometries is available to you**

9.1 programGUIDE DIN/ISO and SINUMERIK high-level language

9.1.1 Introduction

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

Below is an overview of the characteristic functions of programGUIDE and SINUMERIK CNC programming. This includes:

- DIN/ISO editor
- Languages
- programGUIDE input support

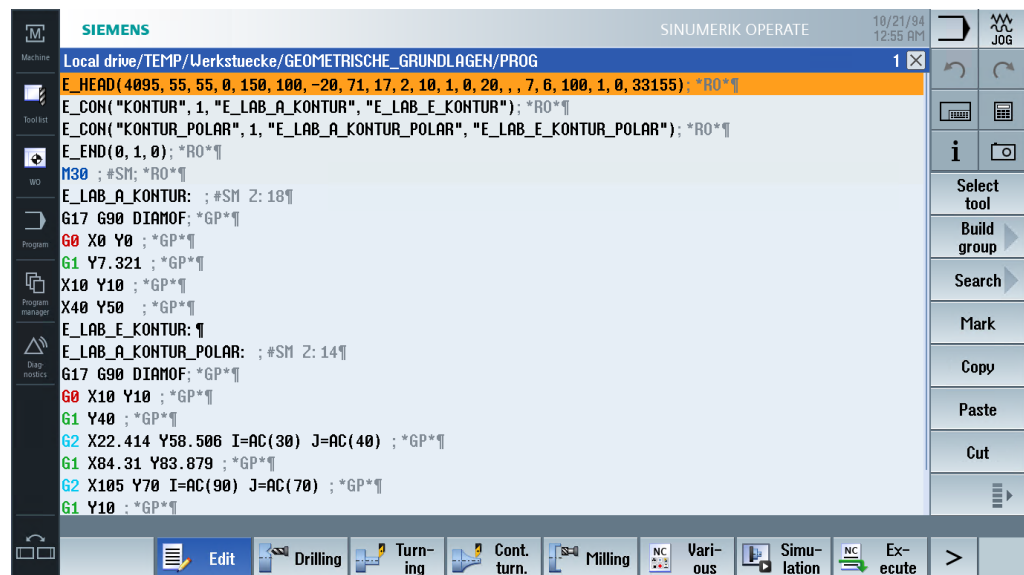
These functions belong to the basic configuration of SINUMERIK Operate.

9.1.2 Program editor

A line-oriented program editor is available to you for DIN/ISO programming. The editor enables you to input CNC language commands directly or to edit them. Thereby, the complete range of CNC functions are available for the most complex machining.

The following functions are included in the program editor:

- Contour calculator
- Tool selection directly from tool list
- Support screens for standard machining and measuring cycles
- "Copy", "Insert" and "Cut" key group
- "Find", "Replace" and "Replace All" character string
- The syntax is highlighted in various colors (comments, NC blocks, etc.)
- Renumbering a program
- Direct execution from any NC program block (block search)
- Jump to program start or program end



Benefits



- Time saving by using a powerful editor when programming
- Even large part programs allow extremely fast editing in MB size

9.1.3 Languages

The CNC Interpreter of the SINUMERIK 828D and the SINUMERIK 840D sl can also process more complex CNC commands, in addition to DIN 66025 standard commands. The commands are presented in clearly readable form.

The following commands are available:

- **G-code**
G-code in accordance with DIN 66025 and in ISO dialect mode
- **G functions**
G0, G1, G2, G71 ...
- **Language commands (extended G functions)**
CIP, SOFT, BRISK, FFWON ...
- **Frame operations (programmable work offsets)**
The workpiece coordinate system can be shifted, scaled, mirrored or rotated with the commands TRANS, SCALE, MIRROR, ROT.
- **R parameters (arithmetic parameters)**
300 predefined R parameters are available as arithmetic parameters (floating-point format).
- **User variables**
Users can define their own variables by name and type.
- **System variables**
System variables can be read/written in all programs. They enable access to work offsets, tool offsets, axis positions, measurement values, control conditions etc.
- **Arithmetic operations**
The following arithmetic operations are available to combine the variables:
arithmetic operations + - * / sin, cos, exp, etc.
logical operations == <> >=, etc.
- **Program control structures**
BASIC-style language commands are available for flexible programming of the user cycles:
IF-ELSE-ENDIF, FOR, CASE ...

Benefits

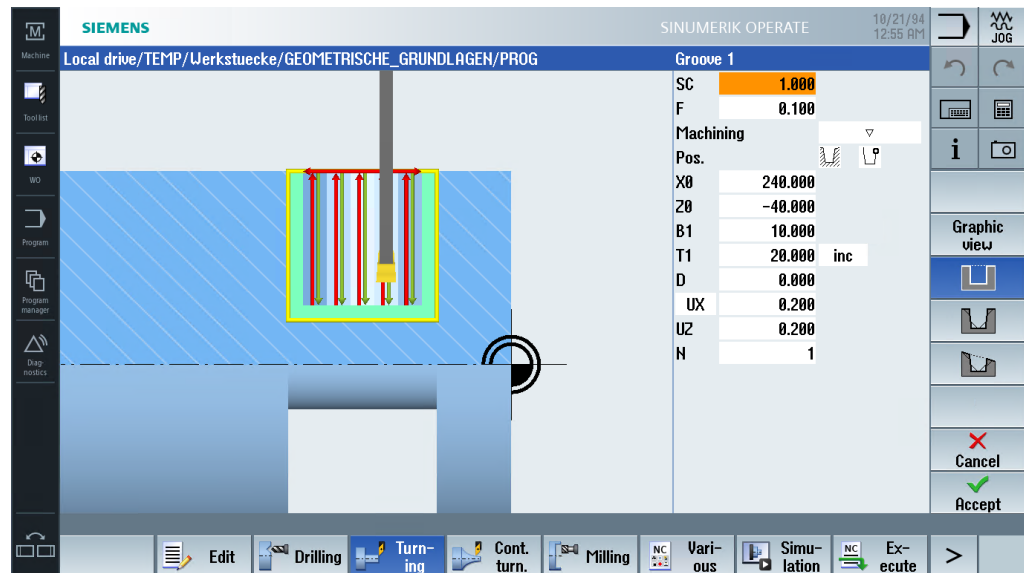


- Established programming according to DIN 66025
- Unbeatable range of commands for flexibility and time saving while programming

9.1.4 programGUIDE input support

The cycle support is an extension of the highly flexible DIN/ISO programming. The input screens are based on the ShopTurn cycles input screens, so as to ensure optimum continuity.

The calls for tool, feedrate and spindle speed can of course also be input in the DIN/ISO editor.






Benefits



- Existing DIN/ISO part programs with cycles can continue to be used
- Minimum learning requirements due to the continuity of the input support

9.2 ShopTurn machining step programming

9.2.1 Introduction

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: P17		Option: P17		Option: P17

	SINUMERIK 840D sl
	Option: P17

The following information provides you with an overview of the characteristic functions of ShopTurn. This includes:

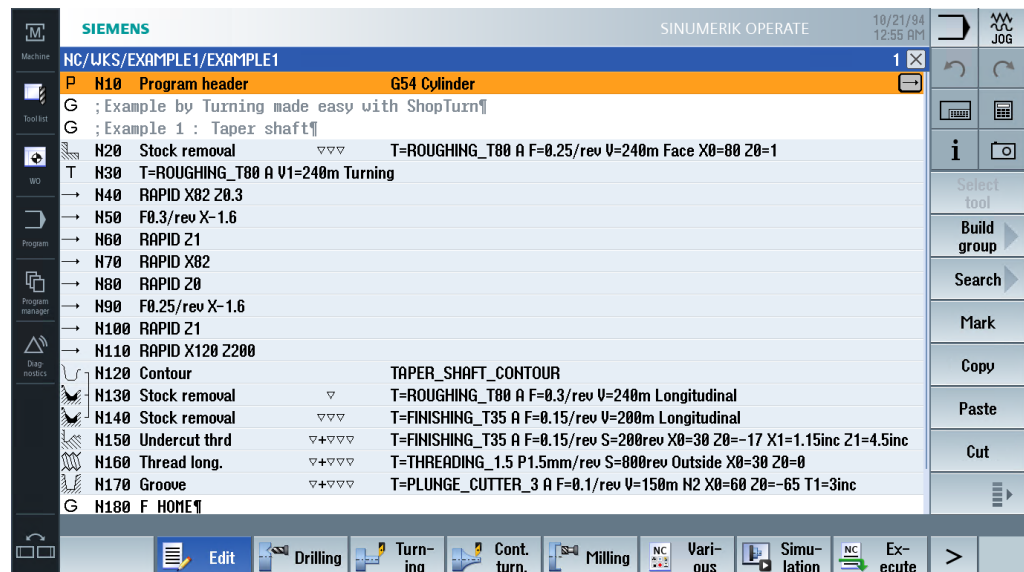
- Sequence editor
- Interlinking of sequences
- Graphic view (broken-line graphics)

These functions are part of the machining step programming options package in ShopTurn.

9.2.2 Sequence editor

The graphical programming is performed via a graphic interactive sequence editor. Each program line represents a technological sequence (such as: face turning, centering, drilling, tapping) or geometric data required for the sequences (position patterns or contours). Graphical programming offers, in comparison to DIN/ISO programming, a compact and comprehensible program overview.

Entering individual sequences requires no knowledge of DIN/ISO. All required technical and geometric parameters are entered in screen forms. Simple, intuitive programming with sequences can always be expanded very flexibly by inputting DIN/ISO blocks and control functions.



Benefits

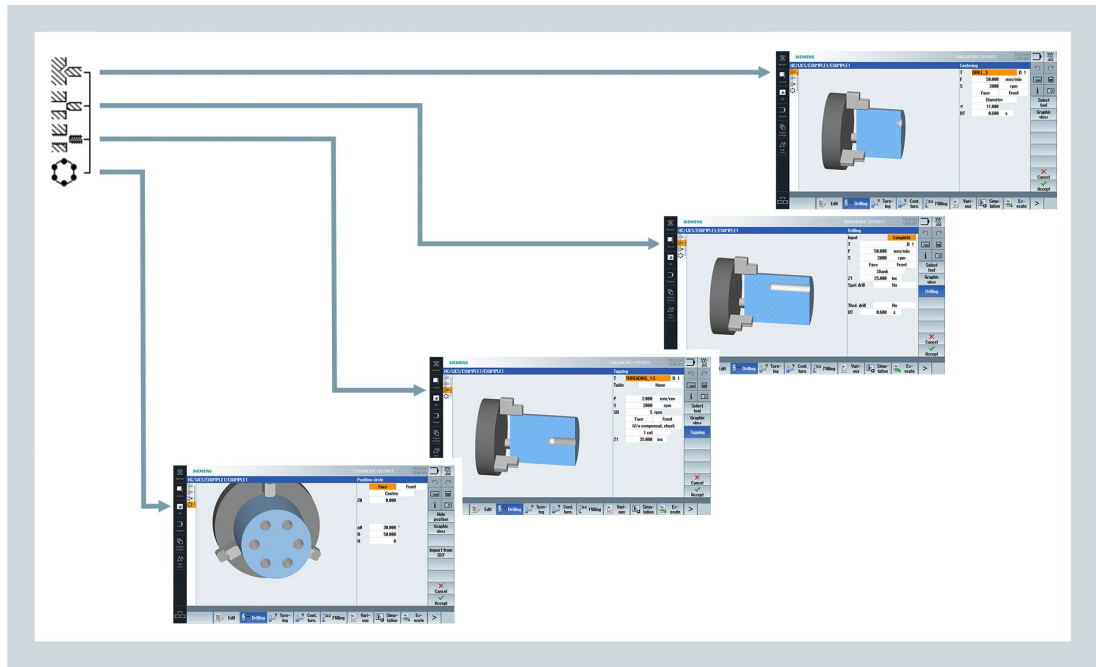


- Intuitive program input, without knowledge of DIN/ISO or Operating Manual
- Compact, clearly arranged machining programs
- Reducing the programming time by graphical input screens and copying / inserting machining steps

9.2.3 Interlinking of sequences

In ShopTurn, associated sequences are interlinked with each other. The interlinked sequences are performed consecutively at the appropriate contours or pattern positions.

In the following example, the sequences centering, drilling and tapping are applied to 4 holes on the pitch circle pattern position.



Benefit



- Reduced programming time due to linking of machining steps

9.2.4 Graphical view

While programming, the previously entered sequences will be continuously displayed to scale. A simulation is not required for this. The switching between the machining step program and the broken-line graphics is performed with the "Graphics View" softkey or the "Ctrl+G" shortcut.

- Turning view
- Front face and peripheral side



Benefit



- Increased reliability at program input by quickly checking the contour, without having to start a simulation run

Workpiece visualization

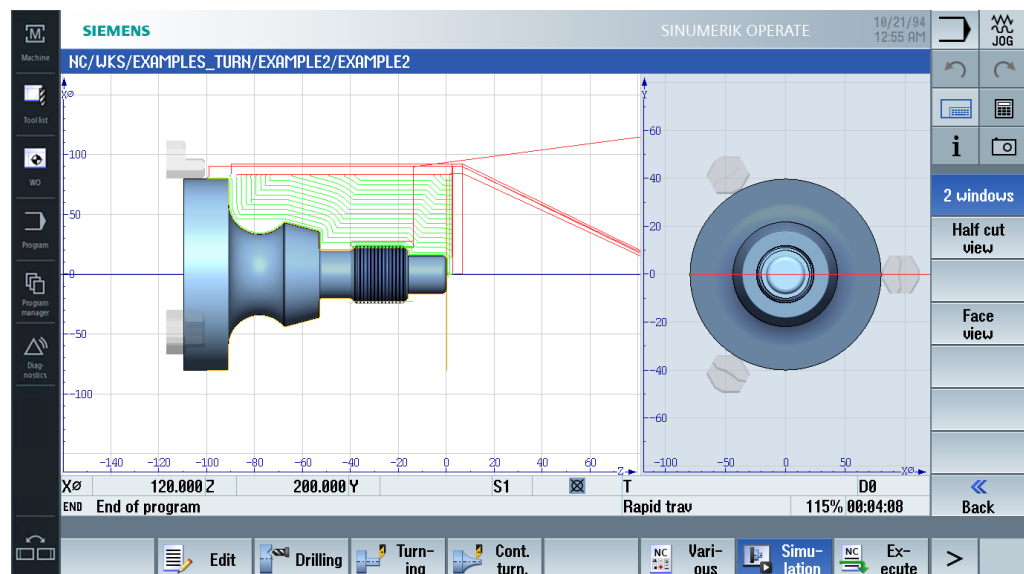
10.1 2D simulation

<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

<input checked="" type="checkbox"/>	SINUMERIK 840D sl
	Basic configuration

SINUMERIK Operate offers with 2D simulation the facility to make optimum and reliable preparations for machining workpieces, such as by detecting collisions. Calculating the machining time also supports optimum calculation of tooling costs.

- Use of the real geometry values of the tools mounted in the machine
- Simulation in side view, front view or two window view
- Simulation can be interrupted at any time, and the speed is controllable



Benefits



- Maximum process reliability through simulation using real geometry values
- Perfect clarity by showing the workpiece dimensions with a scale
- Parallel simulation (background simulation) is possible in conjunction with the NCU 720 and NCU 730, i.e. simulating a part program while another part program is being simultaneously machined.

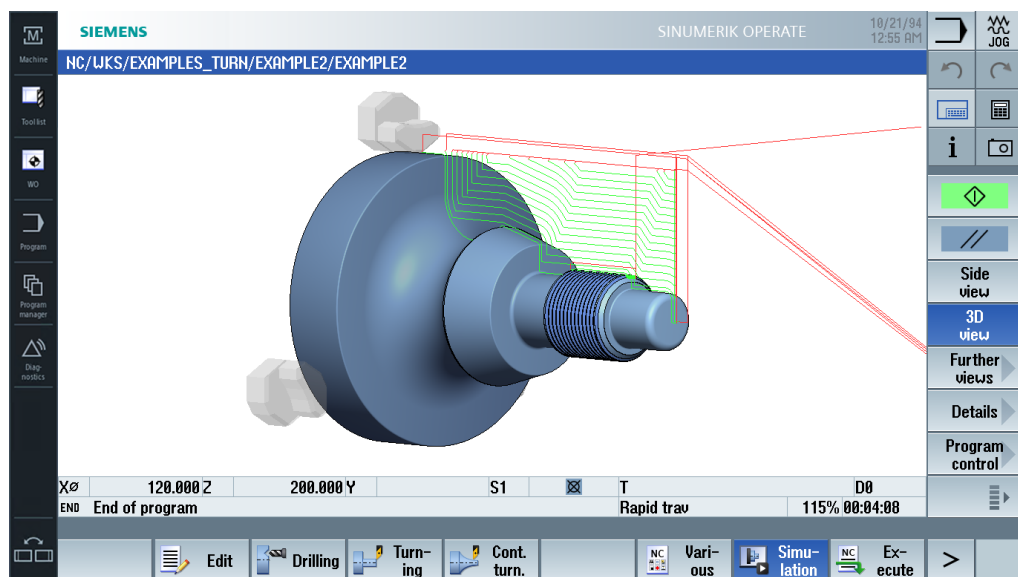
10.2 3D simulation

<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Option: P25		Option: P25		Option: P25

<input checked="" type="checkbox"/>	SINUMERIK 840D sl
	Option: P25

SINUMERIK 3D workpiece simulation offers you optimum assistance and reliability in programming and in quotation costing.

- Reliability:
realistic 3D volume model, with zoom to details and free rotation of the viewing angle
- Support:
 - Simulation speed controllable
 - Single block operation and start / stop available at any time
- Checking:
Automatic calculation of machining time

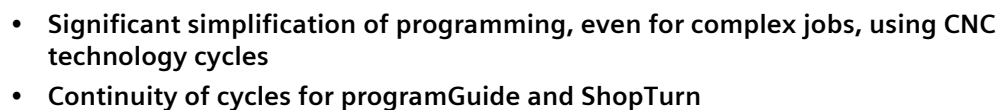


Benefits



- Particularly realistic simulation through representation of the tool
- Optimum help and reliability in programming and in quotation costing
- Parallel simulation (background simulation) is possible in conjunction with the NCU 720 and NCU 730, i.e. simulating a part program while another part program is being simultaneously machined.

Irrespective of whether you use programGUIDE or ShopTurn – in either case the full range of technological cycles, position patterns and geometries is available to you.



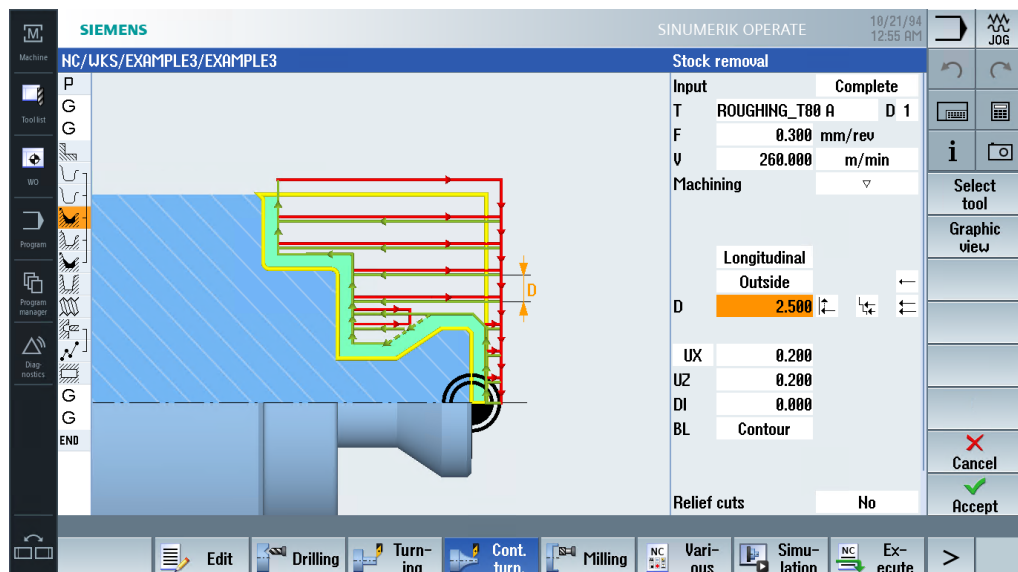
11.2 Highlights of machining cycles

11.2.1 Stock removal along contour with blank contour

<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

<input checked="" type="checkbox"/>	SINUMERIK 840D sl
	Basic configuration

With the intelligent contour stock removal cycle, free contours can be processed in a variety of ways:



- Processing any contour calculator geometry
- Cylindrical blank, freely-defined blank, blank as allowance of finished-part contour
- Longitudinal / face / contour-parallel roughing on outside and inside
- Processing sloping contours (relief cuts)
- Consideration of tool's setting and plate angle
- Grooving any contours on outside, inside or end face
- ...

Benefits



- Effective processing through orientation to the actually existing material
- Lower risk of accident and better chip disposal through feed interruption

11.2.2 Engraving cycle

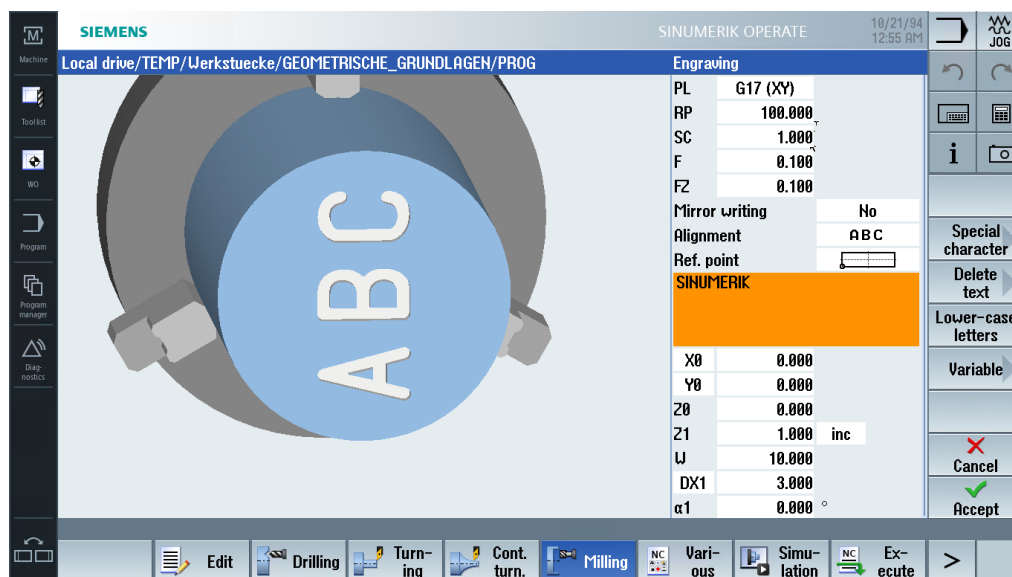
<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

<input checked="" type="checkbox"/>	SINUMERIK 840D sl
	Basic configuration

The engraving cycle is used to engrave a text on a workpiece along a line or arc. You can enter the text as fixed text or assign it via a variable as variable text.

Examples of variable texts:

- Date and time
The values for the date and time are read from the CNC.
- Quantity
The "Quantity" variable is available as a pre-defined user variable
- Numbers
When outputting numbers (e.g. measurement results), you can select the output format (digits before and after the point) of the number to be engraved.
- Text
Instead of entering a fixed text in the engraving text field, you can specify the text to be engraved via a text variable (e.g. `_VAR_TEXT="ABC123"`).



Benefits



- Reduction of set-up times by complete machining on one machine
- Simple program input of engraving

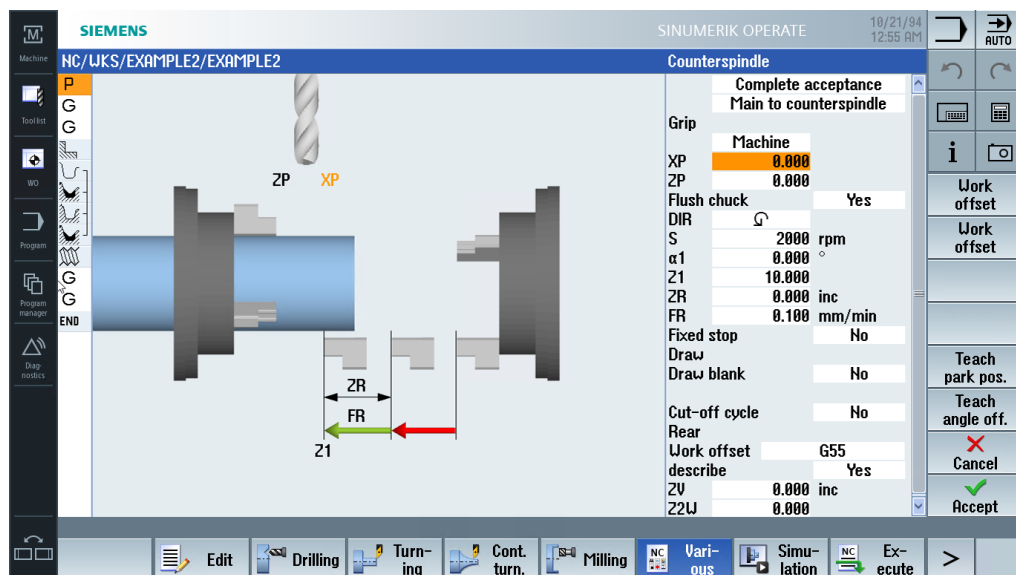
11.2.3 Counterspindle cycle

<input type="checkbox"/>	SINUMERIK 828D SW24x	<input type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	not available		not available		Option: M75

<input checked="" type="checkbox"/>	SINUMERIK 840D sl
	Basic configuration ¹⁾

1) The function is only available in conjunction with ShopTurn/ShopMill machining step programming (option: P17).

SINUMERIK Operate enables the use of a fully-functional counterspindle. The main spindle and counterspindle can be operated under conditions of angular synchronism.



DIN/ISO programming

The commands for spindle synchronization and the axis movements for workpiece transfer can be programmed as DIN/ISO language commands.

Machining step programming

A user-friendly counterspindle cycle is conveniently available for spindle synchronization and axis movements for workpiece transfer.

Benefits



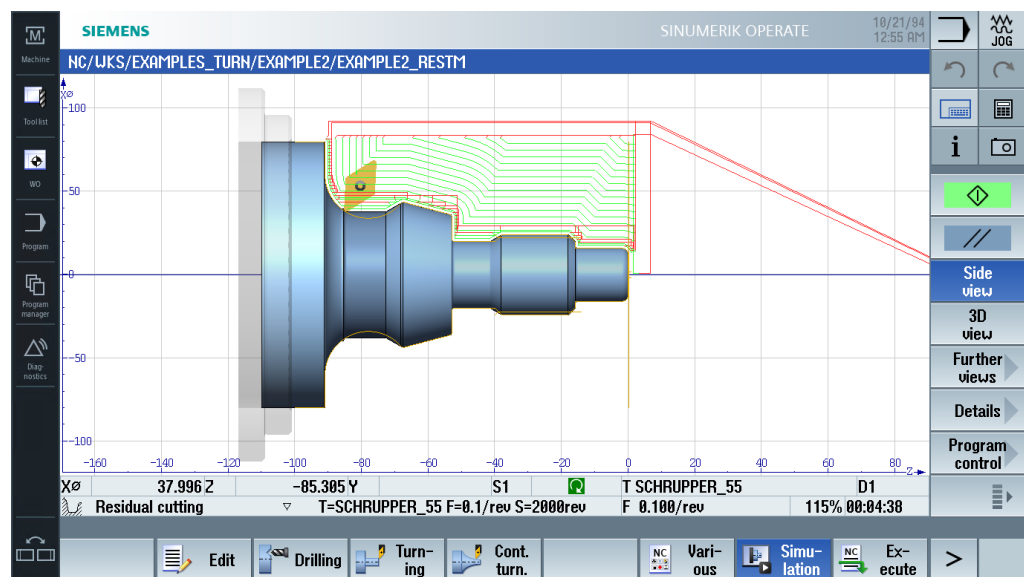
- Simple and secure programming of all counterspindle functions
- High quality of workpieces by workpiece transfer in synchronous spindle mode

11.3 Residual material detection for contour cycles

<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Option: P13		Option: P13		Option: P13

<input checked="" type="checkbox"/>	SINUMERIK 840D sl
	Option: P13

Contour areas which do not permit machining by tools with large plate angles are automatically recognized in the stock removal cycle. The operator can rework these areas using a suitable tool with a smaller plate angle.



Benefit



- Time saving through avoiding idle cuts during residual stock removal

11.4 In-process measuring for workpieces and tools

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: P28		Option: P28		Option: P28

	SINUMERIK 840D sl
	Option: P28

For measurement tasks in automatic operation, powerful measuring cycles are available both within the sequence and in DIN/ISO programming. Input screens with dynamic help displays are used for convenient entry of the measuring parameters.

The following cycles are available for workpiece measurement:

Calibrate probe	Length						
Measurem. turning	Radius in ring						
Edge distance	Radius at edge						
Corner	Comp. to sphere						
Hole							
Spigot							
3D							
Meas. result	<< Back	<< Back	<< Back	<< Back	<< Back	<< Back	<< Back

	Straight Circle		Meas. workp.		Meas. tool
--	-----------------	--	--------------	--	------------

The following measurement variants are available for tool measurement:

- Calibration of the tool probe
- Determination of the tool length of turning tools and drills
- Determination of length/radius/length and radius of milling tools on a turning machine

The following measuring tasks can be made:

- Automatic value correction for tool geometry or work offset
- Display of measurement results
- Logging of measurement results

Benefits



- Reliable quality of the manufactured parts by automatic measurement in the machine
- Fast programming for complex measuring tasks thanks to input screens with graphic support

Complete machining

12.1 End face machining (TRANSMIT)

<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Option: M27		Option: M27		Option: M27

<input checked="" type="checkbox"/>	SINUMERIK 840D sl
	Option: M27

Drilling and milling can be performed on the end face of workpieces in the main and counterspindle with ShopTurn.

The part program is easily created in a right-angle coordinate system with the end face transformation TRANSMIT (C axis mode) .

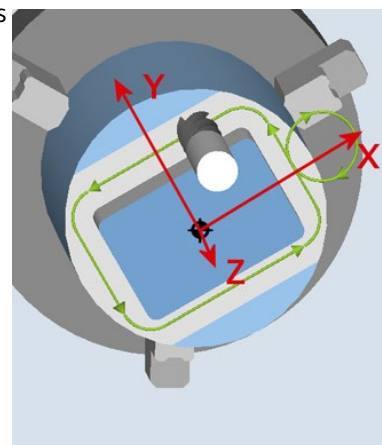
The path movements are conducted with the linear axes X / Z and the rotary axis C.

Machine without Y axis

- Machining with TRANSMIT

Machine with Y axis

- Machining with Y axis
- Machining with TRANSMIT



Benefit



- Full functional range for drilling and milling on the end face

12.2 Peripheral surface machining (TRACYL)

✓	SINUMERIK 828D SW24x	✓	SINUMERIK 828D SW26x	✓	SINUMERIK 828D SW28x
	Option: M27		Option: M27		Option: M27

✓	SINUMERIK 840D sl
	Option: M27

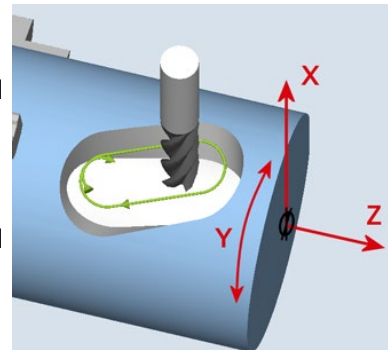
Using the peripheral surface transformation TRACYL, drilling and milling machining can be executed on the peripheral surface of workpieces in the main and counterspindle.

Machine without Y axis

- Any drill holes on the peripheral surface
- Any milling without slot wall offset on the peripheral surface

Machine with Y axis

- Any drill holes on the peripheral surface
- Any milling without slot wall offset on the peripheral surface
- Any milling with slot wall offset on the peripheral surface
- Grooving on parallel walls of the peripheral surface with milling radius correction






Benefits



- Full functional range for drilling and milling on the peripheral surface
- Reduction of set-up times by complete machining on one machine

Multi-channel machining

13.1 Overview

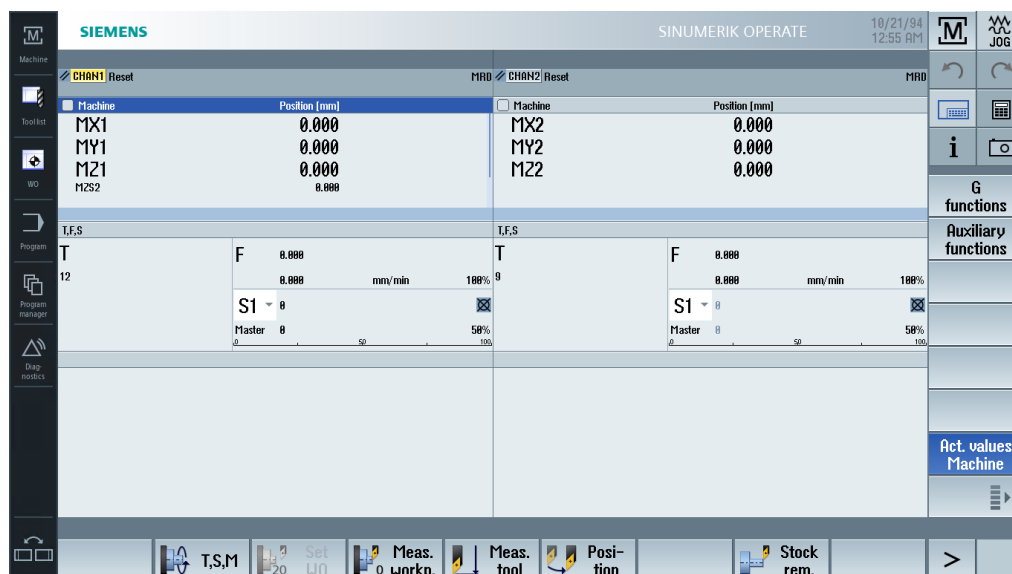
	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	not available		not available		Option: P05

	SINUMERIK 840D sl
	Option: P05

SINUMERIK Operate offers numerous functions to support you with programming and production of multi-channel machining processes.

With programSYNC, you can easily synchronize, optimize and visualize programs for multi-channel machining processes.

For the machine basic screen you can choose between single-channel and multi-channel view. The active channel is highlighted in color.



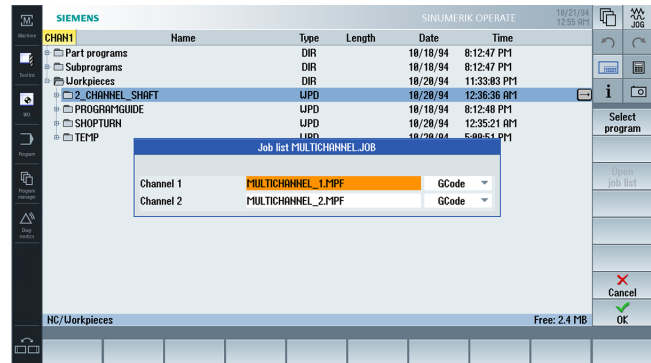
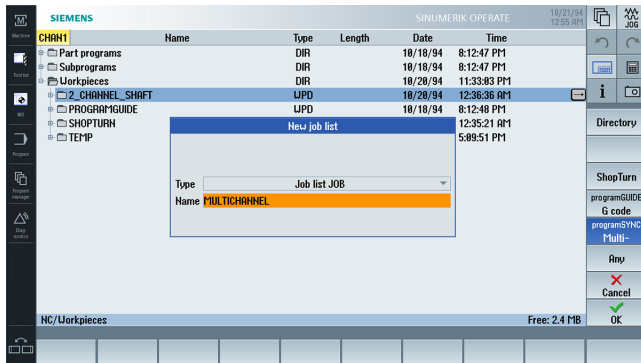
Benefit



- SINUMERIK supports the easy handling of complex machines.

13.2 programSYNC job list

In programSYNC multi-channel, the programs for processing of the respective channels are managed in job lists. In the job list, you assign arbitrary ShopTurn or G code programs to the respective channels.



Benefit

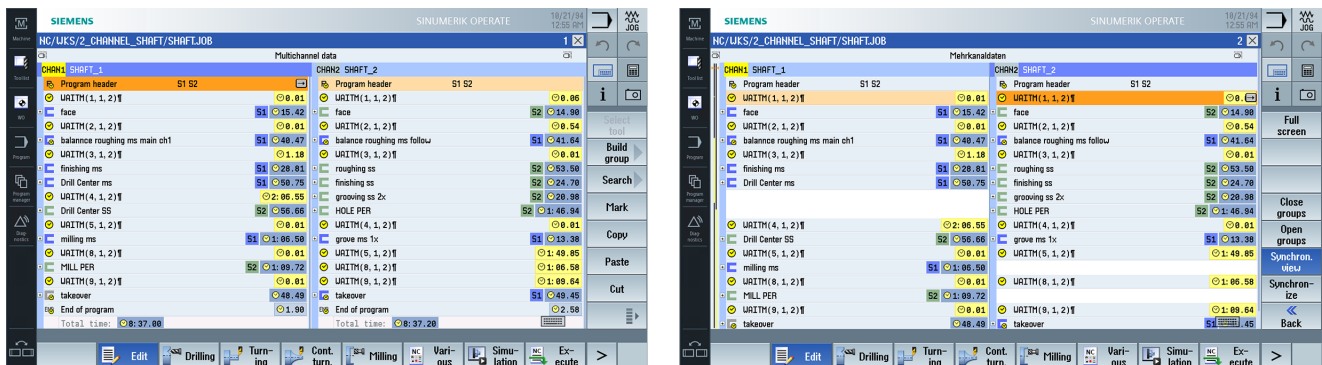


- Simple program management in Windows Explorer style

13.3 Double editor

The double editor facilitates the creation of the programs for the respective channels.

- You structure the programs by means of blocks. These can be expanded and collapsed for a clear representation.
- In the double editor, you can program the chronological sequence and check the wait marks through the synchronized view.
- Through the automatic time evaluation, you can further optimize the multi-channel program in the double editor. If required, you can transfer individual machining processes to other channels to create a time-optimized program.



Benefit



- Easy creation of time-optimized programs through synchronization of wait marks and determination of the processing time of the respective blocks

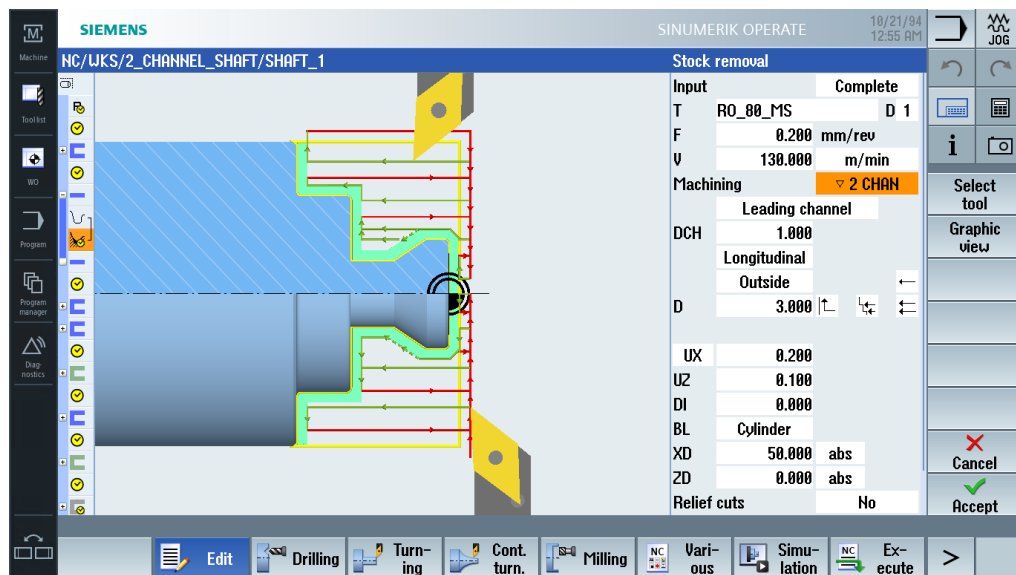
13.4 Balance cutting (stock removal)

With the multi-channel contour stock removal cycle (CYCLE952), complex 4-axis turning can be programmed directly on the machine without a CAD/CAM system. The contour and stock removal parameters can simply be entered in the master channel. The CNC sequences are created completely automatically by the contour stock removal cycle. With just two additional parameters, machining with a tool can be extended to highly productive balance cutting with two tools.

With the multi-channel stock removal cycle, contours can be machined in a variety of ways. Powerful functions facilitate the workpiece machining:

- Automatic detection of residual material ensures an optimum cut segmentation
- Automatic feedrate interruption breaks the chips evenly and removes them

Note: 4-axis stock removal with CYCLE952 is available on multi-channel turning machines.



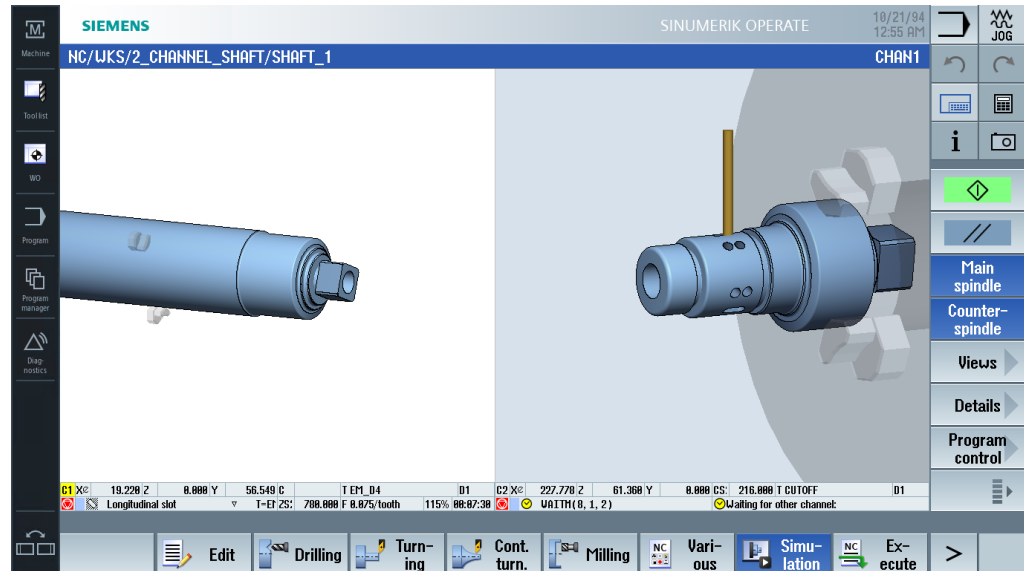
Benefit



- Cycle support enables efficient programming of complex machining tasks for multi-channel turning machines
- Higher throughput of workpieces per machine while maintaining a high level of machining accuracy

13.5 Simulation

For the simulation, you can select, among other things, machining on the main spindle and counterspindle and choose between different views, including 3D view.






Benefit



- With the workpiece simulation, SINUMERIK offers optimum help and safety for programming - even during parallel machining

Automation

14.1 SINUMERIK Run MyRobot /EasyConnect

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

The prepared Run MyRobot / EasyConnect configuring interface enables handling robots to be connected to machine tools.

- Prepared NC/PLC interface in accordance with VDMA/VDW 34180
- Prepared CNC diagnostic screen




Note:

The robot is normally connected to the CNC by the machine manufacturer or a system integrator.

Benefit

- The prepared Run MyRobot / EasyConnect configuration interface provides a universal and multi-vendor interface for the low-effort automation of machine tools.

14.2 SINUMERIK Run MyRobot /Handling

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: via SISW *)		Option: via SISW *)		Option: via SISW *)

*) ab SINUMERIK RUN MyRobot /Handling V4.0

	SINUMERIK 840D sl
	Option: via SISW

The Run MyRobot / Handling option enables a robot to be operated, programmed and diagnosed for handling tasks with SINUMERIK Operate.

- Operation, teaching and programming of the robot in the familiar CNC programming environment.
- Minimum training effort, because fully integrated in SINUMERIK Operate.
- Efficient loading and unloading of a machine by direct programming in a control system.

Note

The robot is connected to the CNC by the machine tool manufacturer or a recommended system integrator*).




*) For details, please contact your local Siemens office.

Benefit



- Run MyRobot / Handling offers the integration of handling robots in machine tools with the best-possible user-friendliness thanks to the familiar CNC look-and-feel.

14.3 SINUMERIK Run MyRobot /Direct Handling

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	not available		not available		not available

	SINUMERIK 840D sl
	Option: via SISW

The Run MyRobot /Direct Handling option is a tailor-made package for the integration of handling robots into the SINUMERIK control system:

The package includes all the necessary options for operating a handling robot with a special axis:

- No separate robot control necessary since the robot arm is controlled directly by the CNC
- Programming is done via G-code
- No training overhead since operation is entirely via SINUMERIK Operate

Benefit



- Run MyRobot /Direct Handling gives you the option to increase the spindle times of the machines.
- Robot automation made easy, especially for flexible production.

Digitalization

15.1 Digitalization - Overview

The portfolio of the Siemens CNC Shopfloor Management Software covers the entire value chain in production – from product design all the way to actual production and service.

Digitalization offers a wide range of opportunities to increase productivity, reduce costs, and improve quality.

You can optimize your production in four specific areas – even with a full-fledged hardware and software landscape.




- Order preparation and execution
 - Manage MyResources /Tools (Page 89)
 - Manage MyResources /Programs (Page 90)
- Efficiency and flexibility in production
 - Analyze MyPerformance (in line) (Page 91)
 - Analyze MyPerformance /OEE Monitor (MindSphere) (Page 92)
 - Analyze MyPerformance /OEE Tuning (MindSphere) (Page 93)
 - Manage MyMachines (Page 93)
 - Manage MyMachines /Remote (Page 94)
- Machine availability
 - Analyze MyMachine /Condition (SINUMERIK Edge) (Page 100)
 - Optimize MyMachining /Trochoidal (SINUMERIK Edge) (Page 97)
 - Optimize MyMachining /AC AUTO (Page 95)
- Improved machining processes
 - SINUMERIK Edge (Page 96)
 - Analyze My Workpiece /Capture (SINUMERIK Edge) (Page 98)
 - Analyze My Workpiece /Monitor (SINUMERIK Edge) (Page 99)
 - Analyze My Workpiece /Toolpath (SINUMERIK Edge) (Page 101)

Note

If you have any questions about the applications, please contact our CNC digitalization experts at SIEMENS Industry Software (SISW). You will receive information about which apps you can use to optimize your manufacturing processes and information about the ordering process and licensing.

> Contact (<https://new.siemens.com/global/en/products/automation/systems/cnc-sinumerik/digitalization/manufacturing.html>)

15.2 Manage MyResources /Tools

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: via SISW		Option: via SISW		Option: via SISW

	SINUMERIK 840D sl
	Option: via SISW

Manage MyResources /Tools permits centralized management of tools.




- Factory-wide management of tools:
Tool planning and stock management for tools and components
- Tool stock during magazine assignment:
Overview of the current tool data, setup dialog for loading and unloading the physical tool
- Data exchange and evaluation:
Standard interface to measuring devices for tool presetting, provision of tool history


Benefit



- Increased transparency through cross-machine availability of tool information
- Increased efficiency through identification of optimization potentials

15.3 Manage MyResources /Programs

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: via SISW		Option: via SISW		Option: via SISW

	SINUMERIK 840D sl
	Basic configuration: single machine Option: networked machine, via SISW

Manage MyResources /Programs supports you in managing NC packages.




- Secure data transmission to and from the machine, i.e. no manual data handling for the machine operator
- Package life cycle management
- Restoring of older revisions


Benefit



- Secure handling of data
- Transparency of the NC package status

15.4 Analyze MyPerformance (in line)

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: via SISW		Option: via SISW		Option: via SISW

	SINUMERIK 840D sl
	Option: via SISW

Analyze MyPerformance enables a targeted analysis of weak points and thus the optimization of production.

- Acquisition of machine data to calculate key parameters:
Key parameters provide information about the state of the plant and allow optimization potential to be identified.
- Acquisition of alarms/messages:
Avoiding disruptions to production by supporting maintenance and deriving preventive maintenance measures.
- Provision of the widest range of evaluations and analyses:
 - Display of the average duration of disruptions and their percentage of the planned machine usage time.
 - Weak point analysis by showing the effects on upstream and downstream stations.

Benefits



- Improved productivity
- Increased machine availability
- Enhanced transparency of the production status

15.5 Analyze MyPerformance /OEE Monitor (MindSphere)

	Control-independent
	Option: via SISW

Analyze MyPerformance /OEE Monitor for analyzing performance in production with machine tools:

- Analyze MyPerformance /OEE Monitor creates maximum transparency of machine states and production data, thus enabling maximum productivity in the production environment.
- Analyze MyPerformance /OEE Monitor calculates the overall equipment efficiency (OEE) and provides important indicators for measures to increase efficiency.
- By automatically acquiring machine data and providing a user dialog for planning machine utilization and adding quality data, Analyze MyPerformance /OEE Monitor provides all the information required to optimize production.

Benefits



- Improved productivity
- Reduction of production costs
- Enhanced transparency of the production status

15.6 Analyze MyPerformance /OEE Tuning (MindSphere)

	Control-independent
	Option: via SISW

With Analyze MyPerformance /OEE Tuning on MindSphere, you can increase productivity and reduce production costs:




- Analyze MyPerformance /OEE Tuning creates maximum transparency of machine states and production data, thus enabling maximum productivity in the production environment.
- Analyze MyPerformance /OEE Tuning calculates the overall equipment efficiency (OEE) and provides important indicators for measures to increase efficiency.
- By automatically acquiring machine data and providing a user dialog for planning machine utilization and adding quality data, Analyze MyPerformance /OEE Tuning provides all the information required to optimize production.

Benefits



- Reduction of production costs based on insights into machine performance
- Optimized planning to ensure delivery reliability
- High transparency of machine utilization

15.7 Manage MyMachines

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: via SISW		Option: via SISW		Option: via SISW

	SINUMERIK 840D sl
	Option: via SISW

Manage MyMachines visualizes numerous operating and plant-specific data of machine tools or individual machine components for production, as well as service and maintenance.


- Possibility to combine critical machine data for a meaningful analysis
- Data acquisition from time series and easy creation of rules and threshold values
- Determination of the machine utilization




Benefit



- Increased availability, utilization and efficiency of machine tools.

15.8 Manage MyMachines /Remote

	SINUMERIK 840D sl			
	Option: via SISW			

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: via SISW		Option: via SISW		Option: via SISW

Manage MyMachines /Remote permits global and secure remote control and monitoring of machine tools connected to MindSphere via Manage MyMachines.




- For critical situations and preventive maintenance, OEM service organizations have immediate access to a comprehensive toolbox for remote diagnostics and troubleshooting.
- If you need support from service specialists such as internal experts or the machine manufacturer, you can provide real-time access to your HMI. All connections to and from a machine tool via the Internet are encrypted.
- Manage MyMachines /Remote meets all security guidelines for remote access to industrial machines.

Benefits



- Faster problem solving and higher machine availability
- Improved service response time and quality

15.9 Optimize MyMachining /AC AUTO

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	not available		not available		not available

	SINUMERIK 840D sl
	Option: via SISW

Optimize MyMachining /AC AUTO monitors the cutting conditions in real time and automatically optimizes the feedrate. By adjusting the feedrate, Optimize MyMachining /AC AUTO minimizes production time and avoids tool breakage.

- Automatic feedrate adjustment to the spindle load
- Feedrate reduction in case of tool overload and impact on material
- Simple and fast configuration

Precondition:

- Run MyCC /IMD
- Run MyHMI /3GL

Optional:

With the "Cross-operational actions" option, the synchronization between the compile cycle and the HMI is executed automatically.




Benefit



- Optimize MyMachining /AC AUTO system for production optimization gives the CNC machine the ability to feel by dynamically adjusting the feedrate!

15.10 SINUMERIK Edge & Applications

15.10.1 SINUMERIK Edge

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	not available		not available		not available

	SINUMERIK 840D sl
	Option: via SISW

SINUMERIK Edge enables new capabilities for the machine tool, for immediate processing of high-frequency data volumes – directly where they arise.

By decoupling data processing tasks and automation, safe machine operation is always guaranteed. At the same time, customer-specific applications run on the SINUMERIK Edge – for example, to ensure workpiece quality and increase machine availability and machine productivity.




The cloud-based services of the SINUMERIK Edge make it possible to distribute updates and new applications within a very short time. Entire machine parks can thus follow shorter innovation cycles – with maximum efficiency.

Benefits



- Enables the storage and transmission of high-frequency data
- Reaction-free: no load on the NCU
- Use and development of further applications on SINUMERIK Edge

15.10.2 Optimize MyMachining /Trochoidal

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	not available		not available		not available

	SINUMERIK 840D sl
	Option: via SISW

Optimize MyMachining /Trochoidal extends the existing vortex milling functionality by using the most advanced algorithms running on the SINUMERIK Edge.

Based on the best possible optimized tool path and dynamic machine data, Optimize MyMachining /Trochoidal adapts programs – for more productive and tool-friendly machining of grooves [as well as pockets in the future].




This SINUMERIK Edge application for optimizing the NC programs is accessed directly on the controller. Its use is therefore particularly intuitive.

Benefits



- Extends tool life and increases productivity
- Extension of the productive use of older machines – due to reduced mechanical load (e.g. bearings)
- Optimized process operations

15.10.3 Analyze MyWorkpiece /Capture

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	not available		not available		not available

	SINUMERIK 840D sl
	Option: via SISW

Measurement data from SINUMERIK are stored in a structured way using Analyze MyWorkpiece /Capture:




- Generic acquisition of high-frequency data, storage in protected file
- Access to stored data via Siemens applications such as Analyze MyWorkpiece /Toolpath
- Access to data from different Edge devices as well as different programs via one instance of Analyze MyWorkpiece /Toolpath

Benefit



- Analyze MyWorkpiece /Capture provides high-frequency data as the basis for a variety of use cases

15.10.4 Analyze MyWorkpiece /Monitor

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	not available		not available		not available

	SINUMERIK 840D sl
	Option: via SISW

Analyze MyWorkpiece /Monitor provides you with the following functions:




- Monitoring of process tags and comparison against the reference model
- Provision of information on the quality produced
- Flexible and job-based monitoring, e.g. for selected NC programs/tools
- Feedback on quality and traceability documentation for each workpiece, e.g. based on KPIs as indicators

Benefit



- Documentation of workpiece and process quality

15.10.5 Analyze MyMachine /Condition

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	not available		not available		not available

	SINUMERIK 840D sl
	Option: via SISW

Analyze MyMachine /Condition enables users to generate a machine tool's mechanical fingerprint so that potential deviations can be identified at an early stage, machine failures prevented, and machine operation optimized.

Various parameters can be recorded with the aid of flexibly configurable measurement series:

- Stiffness
- Friction/friction distribution
- Backlash
- Quadrant error
- Signature
- Down-cut
- Frequency response

The measurement results can be visualized and compared to reference values.

Benefits



- **Basis for condition-based maintenance**
- **Documentation and comparison of machine conditions**
- **Improved machine utilization through optimization of settings**

15.10.6 Analyze MyWorkpiece /Toolpath

	Control-independent
	Option: via SISW

Analyze MyWorkpiece /Toolpath can be used in various production stages:

- Analysis of the part program/dynamic files (trace) before starting machining.
- Analysis of dynamic files (trace) after machining with a real machine

Analyze MyWorkpiece /Toolpath provides various analysis functions:

- Analysis of the NC code
- Comparison of different versions of the optimization
- Checking the alignment of the tool
- Checking the dynamic tool paths

Benefits



- Quick localization of errors/tool paths in the NC program
- Comparison of programmed and real NC paths

Tools and information

16.1 DXF reader

<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Option: P56		Option: P56		Option: P56
<input checked="" type="checkbox"/>	SINUMERIK 840D sl				
	Option: P56				

The integrated DXF Reader allows you to accept and extract contours and positions from DXF files.

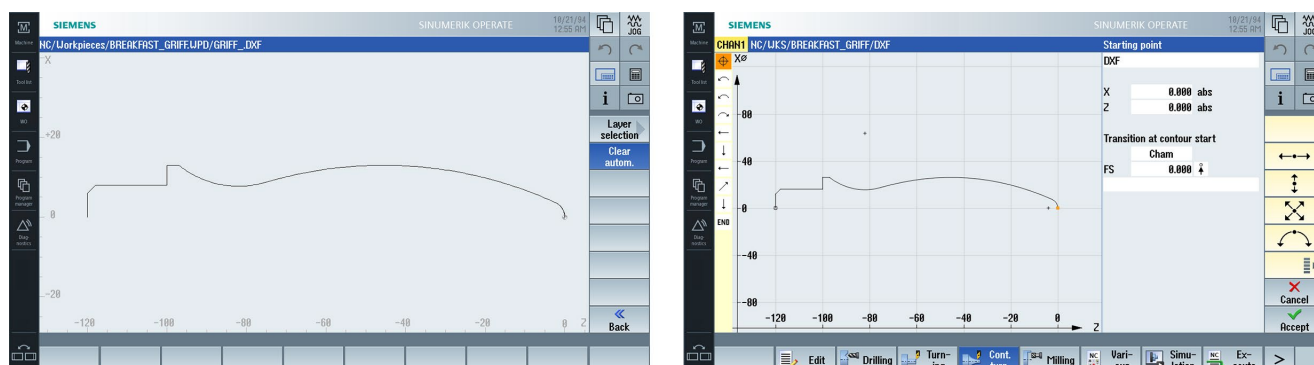
- **DXF Reader in the Program Manager**

With the Program Manager, you can open DXF files in the DXF Reader. You can either clean DXF data automatically or select the desired layer yourself.

- **Import DXF data in the contour calculator**

You can either clean the imported DXF data automatically or select the desired layer yourself.

Cleaned DXF data can be buffered as new DXF file.



- **Import DXF data in position patterns**




You can import the positions from a DXF file for position patterns for the associated technologies.


Benefits



- Time saving for generating the production data
- Avoidance of mistakes and inaccuracies
- Higher workpiece quality

16.2 SinuTrain for SINUMERIK Operate

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: C43, via SISW		Option: C43, via SISW		Option: C43, via SISW

	SINUMERIK 840D sl
	Option: C43, via SISW

SinuTrain for SINUMERIK Operate is a PC-based CNC programming software package based on the original CNC kernel. SinuTrain for SINUMERIK Operate enables identical operation and CNC programming as for SINUMERIK CNCs that are equipped with the SINUMERIK Operate graphical user interface.

SinuTrain for SINUMERIK Operate taps into the following applications:

In work preparation:

- Increased machine availability thanks to work preparation on the CNC programming station and safety by offline verification of the programs
- 1:1 operation and programming as on the machine means no new operating or programming knowledge is required

In training:

- Simple learning and professional training thanks to preconfigured machines and no additional hardware costs
- Learning as on the CNC, with additional tutorials and programming guides

For presentation:

- Present always and everywhere
- Live demonstration of (new) SINUMERIK functions instead of slides

Note

The basic version of SinuTrain for SINUMERIK Operate is available as download in the Internet. More information is available in the Internet at:
www.siemens.com/sinutrain (www.siemens.com/sinutrain)

Note

To order SinuTrain licenses, contact your regional customer service representative or SIEMENS Industry Software (SISW).

> Contact SISW

Benefits

- Controller-identical PC software for training and work preparation with configuration of the real machine on the PC
- Preparation of the part program anywhere without needing a machine
- Prediction of the production time

16.3 CNC4you




On the CNC4you portal, SINUMERIK users can find helpful tips & tricks, SinuTrain downloads, tutorials and more.

CNC4you portal:

<http://www.siemens.de/cnc4you>

Safety functions

17.1 SINUMERIK Safety Integrated

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: refer to the cata- log		Option: refer to the cata- log		Option: refer to the cata- log

	SINUMERIK 840D sl
	Option: refer to the catalog

SINUMERIK Safety Integrated provides integrated safety functions that support the implementation of highly effective personnel and machine protection. The safety functions comply with the requirements of Category 3 as well as Performance Level d according to DIN EN ISO 13849-1 and Safety Integrated Level SIL2 of IEC 61508.

As a consequence, the essential requirements concerning the functional safety can be implemented simply and cost-effectively.

The functional safety for machine tools covers:

- Functions for reliable monitoring of velocity and standstill
- Functions for establishing safe boundaries in work spaces and protected spaces, and for range recognition
- Functions for the safe activation and testing of holding brakes
- Direct connection of all safety-related sensors/actuators and their internal logic combination

Benefits



- **High level of flexibility:** Supports the implementation of practical safety and operating concepts
- **High level of security:** Complete implementation of the safety functions in Category 3/SIL 2
- **Increased availability:** Absence of interference-susceptible electromechanical switching elements
- **High degree of cost effectiveness:** Reduction of the hardware and installation costs; simple commissioning and acceptance

17.2 Collision avoidance

Machine tools are becoming ever faster and more complex. This is also placing more challenging demands on machine operators and programmers.





Operating errors often cause collisions and the associated production outages. This results in standstill times and high repair costs.

Whatever moves in space has the potential to collide. The collision avoidance options ensure optimum protection of moving and static machine components against collisions and prevent major damage.

Note

- The use of collision monitoring requires the availability of the relevant machine data and the associated visualization.
- The options for collision avoidance demand machine-specific enabling. Please contact your sales representative.

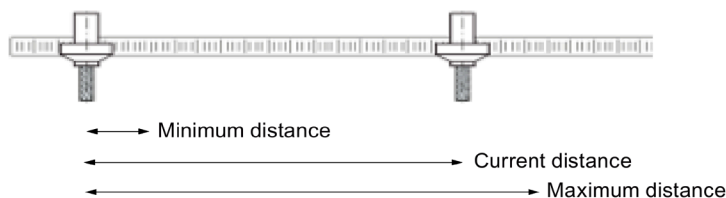
17.2.1 Protect MyMachine /Axis Lock

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	not available		not available		not available
	SINUMERIK 840D sl				
	Option: 6FC5800-0AN06-0YB0				

With the Protect MyMachine option (previously Run MyCC /PROT), you can monitor the minimum and maximum distance between a pair of axes on a shared guide rail.

Braking takes place automatically with a predefined delay.

- Up to 20 axis pairs
- Multi-channel



Benefits



- Low-cost protection for axis pairs.
- Permanent protection through activation of only a few parameters.

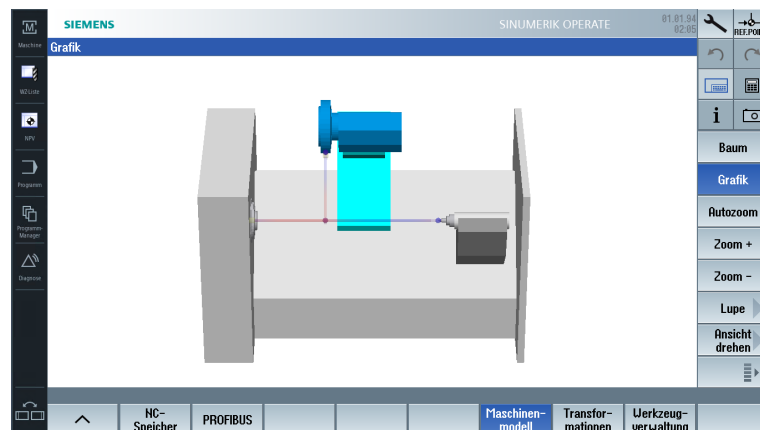
17.2.2 Protect MyMachine /3D Primitives

<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Option: S03		Option: S03		Option: S03

<input checked="" type="checkbox"/>	SINUMERIK 840D sl
	Option: S03

With the Protect MyMachine / 3D Primitives option (previously: Collision Avoidance ECO option), you can monitor the minimum clearance between protection areas. The geometry of the protection areas is defined using protection area elements.

- Up to 17 protection areas
- Up to 34 protection area elements
- Up to 10 collision pairs
- Cube, cylinder, or sphere or truncated cone
- In the modes JOG, MDI, Automatic
- Single-channel



Benefits



- Low-cost entry into the protection of the machine.
- Reduced CPU load of the CNC.

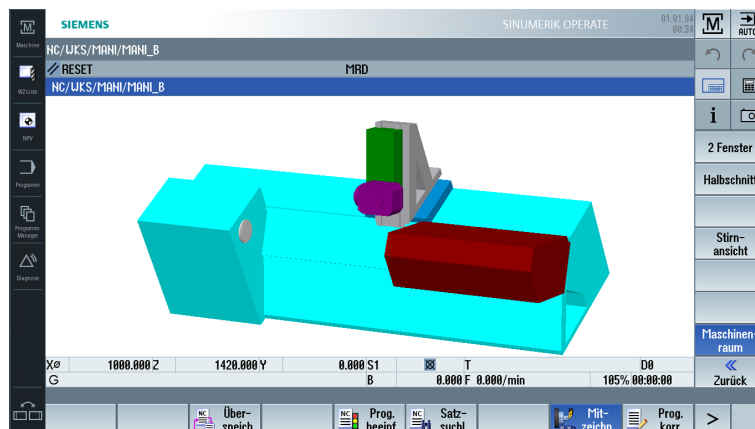
17.2.3 Protect MyMachine /3D STL

<input type="checkbox"/>	SINUMERIK 828D SW24x	<input type="checkbox"/>	SINUMERIK 828D SW26x	<input type="checkbox"/>	SINUMERIK 828D SW28x
	not available		not available		not available

<input checked="" type="checkbox"/>	SINUMERIK 840D sl
	Option: S02

With the Protect MyMachine /3D STL option (previously: Collision Avoidance option), you can monitor the minimum clearance between protection areas. The geometry of the protection areas is defined using protection area elements.

- Like Collision Avoidance ECO
- Up to 500 protection area elements (based on CAD – STL format)
- In the modes JOG, MDI, Automatic







Benefit



- Machine-oriented mapping of complex protection areas possible.

17.2.4 Protect MyMachine /Open

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	not available		not available		not available

	SINUMERIK 840D sl
	Option: S04

The Protect MyMachine /Open option (previously: Collision Avoidance ADVANCED) provides the following functions:

- Data interface for the integration of the Collision Avoidance system from ModuleWorks
- Inclusion of the entire machine model (3D machining area) in collision avoidance
- Collision protection even when using cycles and transformations
- Import/modification of the 3D models of tool, tool holder, clamping device, workpiece and tool adapter (angular head) directly from the CAD/CAM system
- Color highlighting in case of danger of collision enables quick identification of the collision location
- Real-time simulation of material removal
- Predictive collision detection by the CAS system enables controlled stopping or braking of the axes
- Collision detection using the look-ahead function
- JOG, MDI, Automatic modes

Benefit



- Collision monitoring also possible for complex machining operations, such as 5-axis simultaneous milling or turning with B axis.

Index

8

80-bit NANO floating-point accuracy, 49

A

Animated Elements, 23

B

Basic block display, 46

Block change times, (SINUMERIK 828D), (SINUMERIK 840D sl)

Block search, 43

C

CNC technology cycles

 programGuide, 69

 ShopTurn, 69

Collision protection Axes, 108

Contour stock removal cycle, 70

Contour-grooving, 70

D

DIN/ISO language, 60

DIN/ISO programming, 60

E

End face machining, 75

Engraving cycle, 71

F

Feed interruption, 70

Feedforward control, 52

G

G code editor, 59

I

In-process measurement, 74

M

Measure tool, 32

Monitoring of tool life and workpiece count, 38

Multi-channel machining

 Blocks, 79

 Job list, 78

 Machine basic screen, 77

 programSYNC, 78

 Simulation, 81

 Wait marks, 79

O

Online Help, 24

P

Peripheral surface machining, 76

Plunge-turning, 70

Positioning cycle, 34

Program editor, 59

Program manager, 41

Protect MyMachine / 3D Primitives, 109

Protect MyMachine /3D STL, 110

Protect MyMachine /Axis Lock, 108

Protect MyMachine /Open, 111

R

Replacement tools, 39

Residual material detection during turning, 73

S

Safety functions, 107

Safety Integrated, 107

Sequence editor, 63

Simulation

 2D, 67

- 3D, 68
- Processing time, 68
- Simultaneous recording, 47
- SINUMERIK blackline plus
 - SIMATIC IFP, 21
 - SIMATIC IPC, 21
 - SIMATIC ITC, 21
- SINUMERIK CNC4you, 106
- SINUMERIK OP 010, 18
- SINUMERIK OP 010C, 19
- SINUMERIK OP 010S, 18
- SINUMERIK OP 012, 19
- SINUMERIK OP 015 black, 20
- SINUMERIK OP 019, 20
- SINUMERIK OP 019 black, 20
- SINUMERIK OP 08T, 18
- SinuTrain, 104
- Speed control, 51
- Stock removal cycle, 33

T

- Tool management, 37
- TRACYL, 76
- TRANSMIT, 75
- TSM universal cycle, 29