

**SIEMENS**

Control system overview  
for machine tools' sales people

## **SINUMERIK Operate - Turning**

SINUMERIK 840D sl / SINUMERIK 828D

Edition

07/2020

[www.siemens.com/sinumerik](http://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 4.8

**07/2020**

A5E41992653B AB




#### 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.

 <b>DANGER</b>
indicates that death or severe personal injury <b>will</b> result if proper precautions are not taken.
 <b>WARNING</b>
indicates that death or severe personal injury <b>may</b> result if proper precautions are not taken.
 <b>CAUTION</b>
indicates that minor personal injury can result if proper precautions are not taken.
<b>NOTICE</b>
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:

 <b>WARNING</b>
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 **SINUMERIK 828D** and **SINUMERIK 840D sl** with **SINUMERIK Operate V4.8** 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
Gretler	Markus	Switzerland	markus.gretler@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
Messner	Erwin	Vietnam	erwin.messner@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

	<b>Preface .....</b>	<b>3</b>
<b>1</b>	<b>Compact overview .....</b>	<b>9</b>
<b>2</b>	<b>System overview.....</b>	<b>13</b>
2.1	SINUMERIK 828D .....	14
2.1.1	SINUMERIK 828D .....	14
2.1.2	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
2.2.3	Operator panel equipment.....	22
<b>3</b>	<b>CNC operation with SINUMERIK Operate .....</b>	<b>23</b>
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
<b>4</b>	<b>CNC operation in manual mode (JOG).....</b>	<b>29</b>
4.1	TSM universal cycle.....	29
4.2	Work offsets .....	30
4.3	Measure workpiece.....	31
4.4	Measure tool .....	32
4.5	Logging measurement results in JOG .....	33
4.6	Stock removal cycle .....	34
4.7	Positioning cycle .....	35
4.8	Manual machine .....	36
<b>5</b>	<b>Tool management.....</b>	<b>37</b>
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

<b>6</b>	<b>Data management.....</b>	<b>41</b>
6.1	Program manager .....	41
6.2	Ethernet networking.....	42
<b>7</b>	<b>CNC operation in automatic mode (AUTO).....</b>	<b>43</b>
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
<b>8</b>	<b>CNC funtionalities.....</b>	<b>49</b>
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
<b>9</b>	<b>CNC programming methods .....</b>	<b>57</b>
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
<b>10</b>	<b>Workpiece visualization.....</b>	<b>67</b>
10.1	2D simulation .....	67
10.2	3D simulation .....	68



<b>11</b>	<b>CNC technology cycles.....</b>	<b>69</b>
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
11.3.1	Residual material detection during turning.....	73
11.3.2	Residual material detection during milling.....	74
11.4	In-process measuring for workpieces and tools.....	75
<b>12</b>	<b>Complete machining .....</b>	<b>77</b>
12.1	End face machining (TRANSMIT) .....	77
12.2	Peripheral surface machining (TRACYL) .....	78
<b>13</b>	<b>Multi-channel machining.....</b>	<b>79</b>
13.1	Overview .....	79
13.2	programSYNC job list .....	80
13.3	Double editor.....	81
13.4	Balance cutting (stock removal) .....	82
13.5	Simulation .....	83
<b>14</b>	<b>Automation.....</b>	<b>85</b>
14.1	SINUMERIK Integrate Run MyRobot / EasyConnect .....	85
14.2	SINUMERIK Integrate Run MyRobot / Handling .....	86
<b>15</b>	<b>Digitalization .....</b>	<b>87</b>
15.1	Digitalization - Overview .....	87
15.2	Manage MyPrograms .....	88
15.3	Manage MyTools .....	89
15.4	Manage MyMachines .....	90
15.5	Analyze MyPerformance.....	91
15.6	Access MyMachine (AMM) .....	92
15.7	Optimize MyMachining /AC AUTO .....	93
15.8	Analyze MyWorkpiece .....	94
<b>16</b>	<b>Tools and information .....</b>	<b>95</b>
16.1	DXF reader.....	95
16.2	SinuTrain for SINUMERIK Operate.....	96
16.3	CNC4you .....	97

<b>17</b>	<b>Safety functions.....</b>	<b>99</b>
17.1	SINUMERIK Safety Integrated .....	99
17.2	Collision avoidance .....	100
17.2.1	Collision protection Axes Run MyCC /PROT.....	100
17.2.2	Collision Avoidance ECO .....	101
17.2.3	Collision avoidance .....	102
17.2.4	Collision Avoidance ADVANCED.....	103
	<b>Index.....</b>	<b>105</b>

# 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)



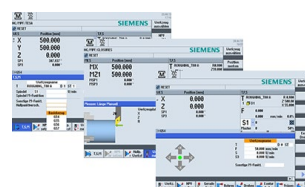
## User-friendliness - effective operation like on a PC

- ✓ 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)



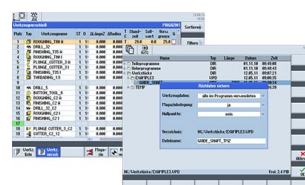
## Setup functions "Intelligent-JOG"

- ✓ 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)



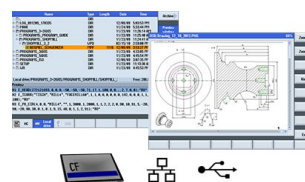
## Tool management - powerful but nevertheless easy to use

- ✓ 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)



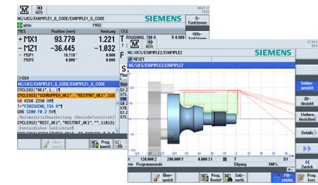
## Data management like on a PC

- ✓ 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)



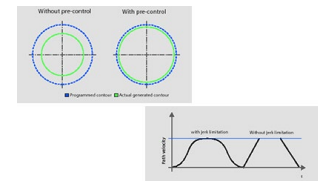
## CNC operation in automatic mode (AUTO)

- ✓ 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)



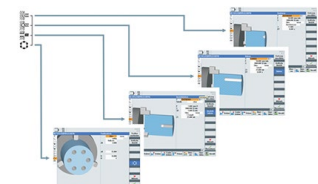
## SINUMERIK CNC performance - the benchmark in all aspects

- ✓ 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)



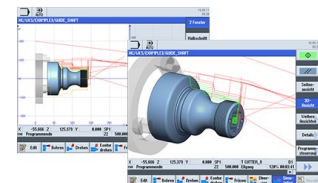
## CNC programming methods - optimally prepared for all production tasks

- ✓ 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)



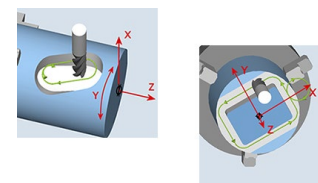
## CNC technology cycles - the little helpers for daily CNC programming

- ✓ 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)



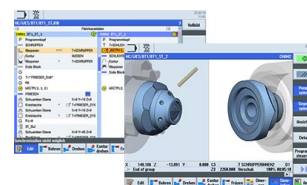
## Complete machining

- ✓ 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 77)



## 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 79)



## 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 85)



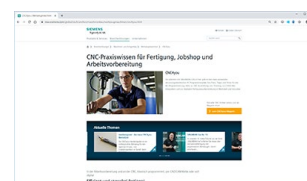
## Digitalization on the shop floor

- ✓ 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 95)



## 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 99)





## 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

### 2.1.1 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



## 2.1.2 Data storage - SINUMERIK 828D

Internal memory			
	828D SW 24x	828D SW 26x	828D SW28
	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
Internal memory can be expanded via option P77 → 100 MB			

External storage
Execution from external storage (EES) (option P75*) Network, USB storage media, compact flashcard
Execution from the CNC expanded user memory (option P77)
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 <sup>1)</sup>			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 75-key CNC keyboard (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

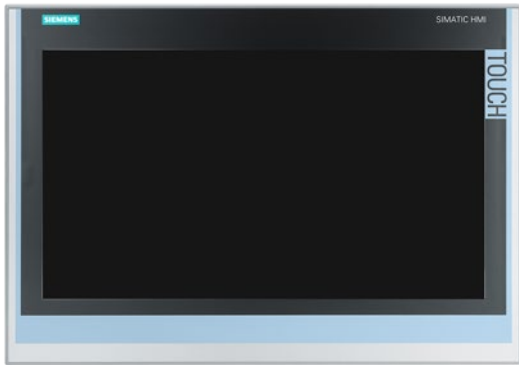
### SINUMERIK OP 019 black



- 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

## SIMATIC Industrial Thin Client



### SIMATIC Industrial Thin Client

- Touch operation
- Connection via Ethernet

#### Versions:

SIMATIC ITC 1200, 12" widescreen TFT display  
(resolution 1280 x 800 pixels)

SIMATIC ITC 1500, 15" widescreen TFT display  
(resolution 1280 x 800 pixels)

SIMATIC ITC 1900, 19" widescreen TFT display  
(resolution 1366 x 768 pixels)

**Note:** see also Chapter AUTOHOTSPOT

### 2.2.3 Operator panel equipment

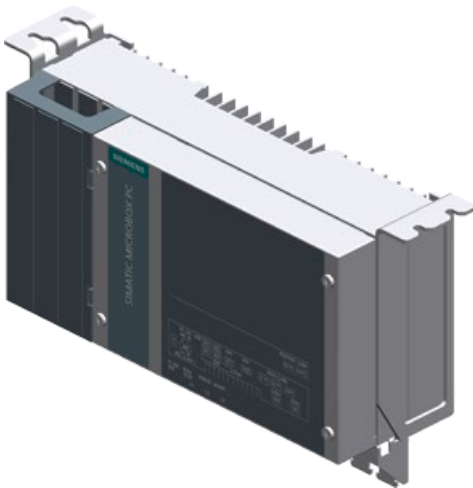
#### PCU 50



If you need a hard disk or supplementary Windows-based software, we offer the PCU 50.x.

- Windows 7 operating system
- Up to 40 GB for data (part programs, documentation, other data)
- Additional PCI slots
- Additional CF card slot
- DVI interface

#### SIMATIC IPC



Panel PC variant up to 19" panels for multitouch operation:

- Compact and rugged
- Solid State Drive (SSD)
- Passive cooling
- Specific configuration for SINUMERIK



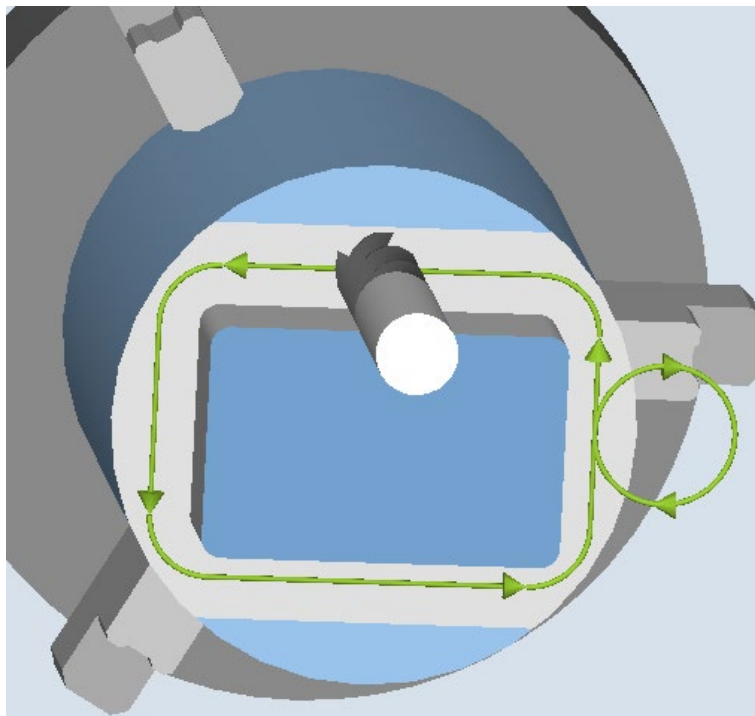
## CNC operation with SINUMERIK Operate

### 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

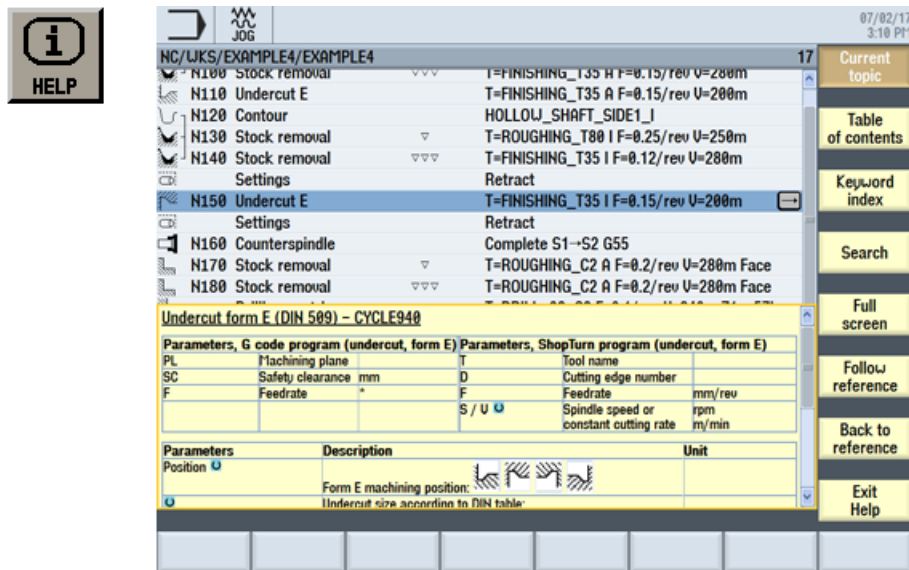
## 3.2 Onboard documentation

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

	SINUMERIK 840D sl
	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

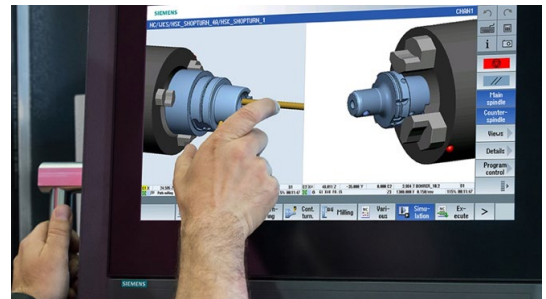
### 3.3.1 Multi-touch operation, basic configuration

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Basic configuration		Basic configuration		Basic configuration






	<b>SINUMERIK 840D sl</b>
	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:




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

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Basic configuration		Basic configuration		Basic configuration

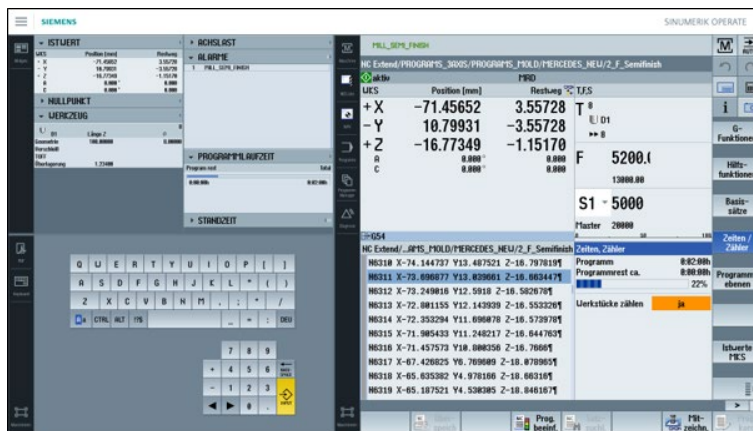
	<b>SINUMERIK 840D sl</b>
	Basic configuration

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.

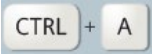
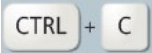



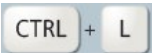


## 3.4 Shortcuts

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Basic configuration		Basic configuration		Basic configuration

	<b>SINUMERIK 840D sl</b>
	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

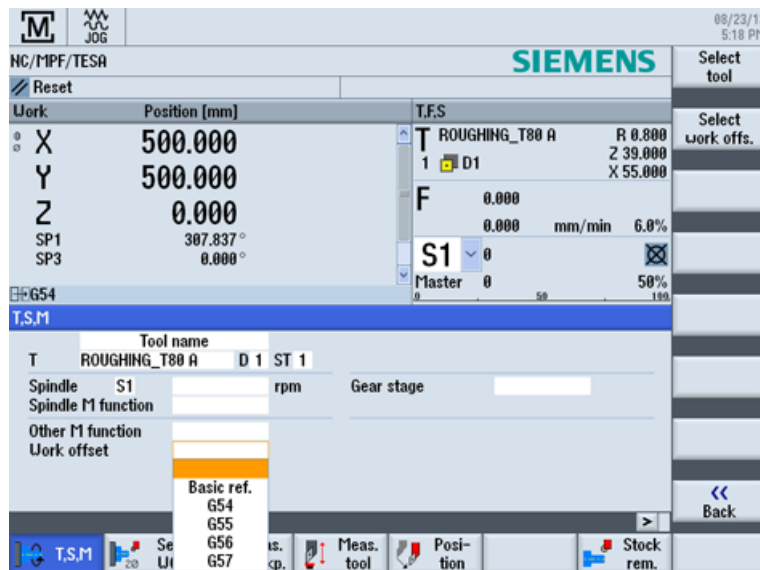
<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

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

A 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

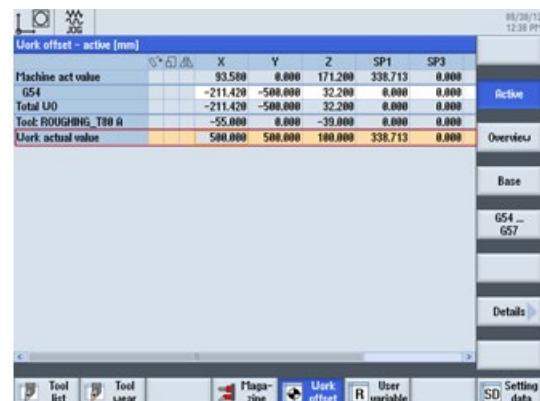
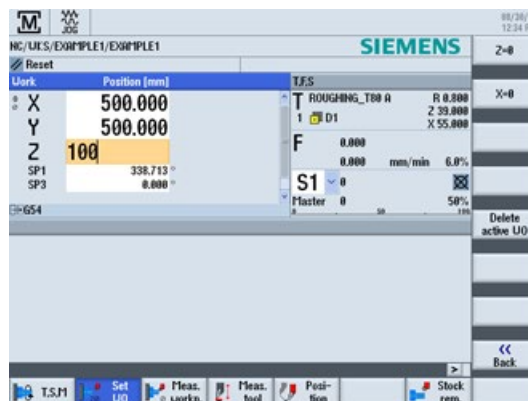
<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

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

The 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.



### 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

<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 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 Logging measurement results in JOG (Page 33)).



### Benefit



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

## 4.5 Logging measurement results in JOG

<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

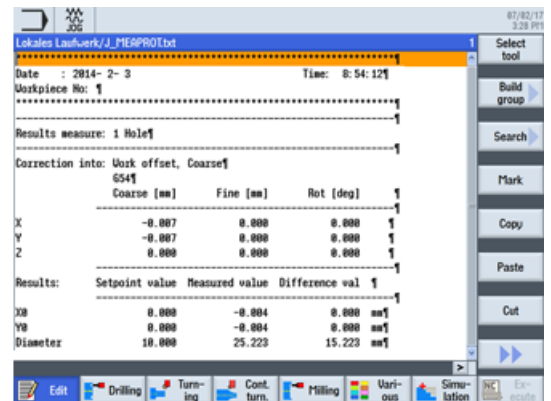
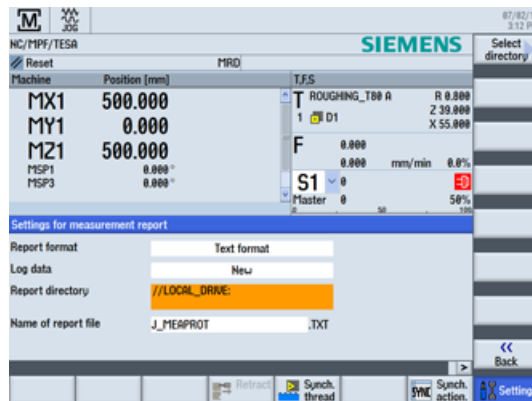
The results for measuring in JOG can be logged. The standard log contains the measurement results of the most recently performed measuring method.

The function is available as turning technology for the tool measurement.

Text format or table format can be selected for the output format

The measuring log comprises the following data:

- Date and time when the log was written
- Log name with path details
- Measuring method
- Correction target
- Setpoints, measured values and differences



### Benefit



- Simple logging of measured values in log files

## 4.6 Stock removal cycle

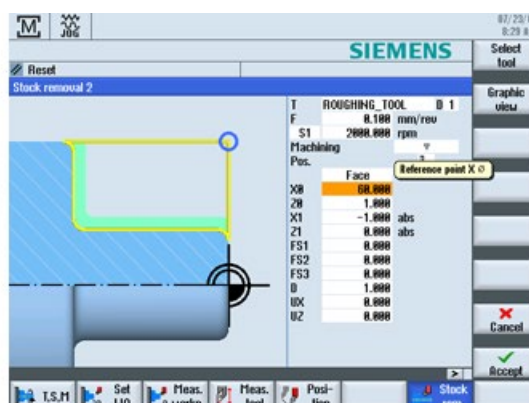
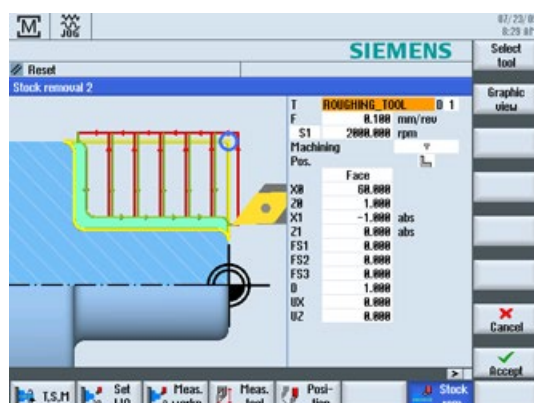
<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

<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



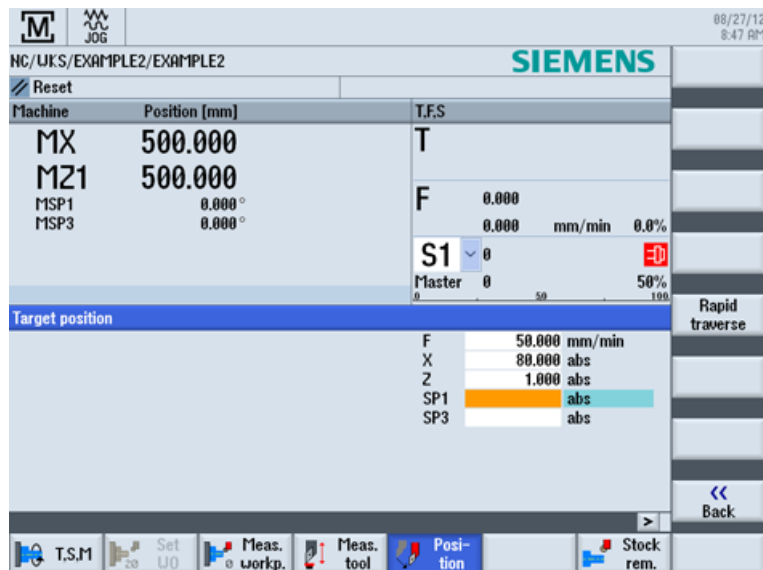
- Preparation of workpiece or collet chuck without having to create a part program

## 4.7 Positioning 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 machine axes can be positioned directly via input screens in the setup:

- Linear axes / spindles
- Feedrate / rapid traverse



### Benefit



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

## 4.8 Manual machine

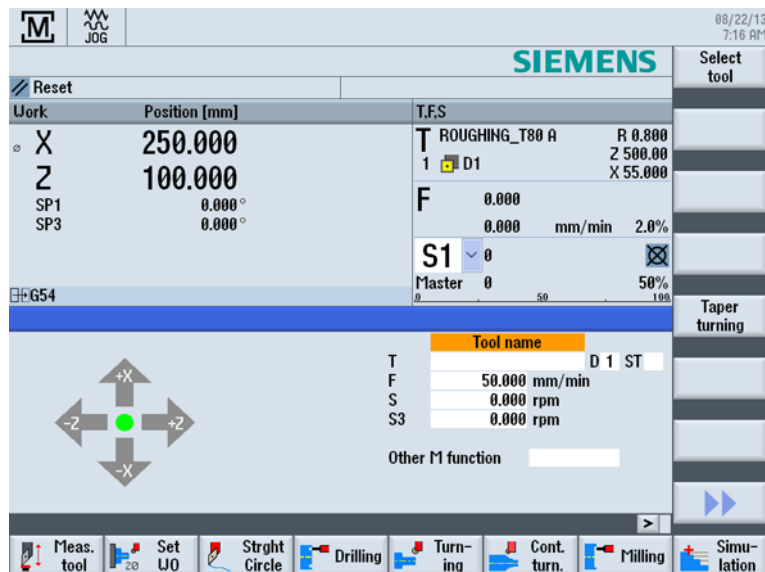
<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Option: P17		Option: P17		Option: P17

<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

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Basic configuration		Basic configuration		Basic configuration

	<b>SINUMERIK 840D sl</b>
	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

Loc.	Type	Tool name	ST	D	ΔLength Z	ΔRadius	T	C	Tool life	Set val	Prewar limit	D
1		ROUGHING_T80 A	1	1	0.000	0.000	T		29.0	0.0	25.0	
2		DRILL_32	1	1	0.000	0.000						
3		FINISHING_T35 A	1	1	0.000	0.000						
4		ROUGHING_T80 I	1	1	0.000	0.000						
5		PLUNGE_CUTTER_3 A	1	1	0.000	0.000						
6		PLUNGE_CUTTER_3 I	1	1	0.000	0.000						
7		FINISHING_T35 I	1	1	0.000	0.000						
8		THREADING_1.5	1	1	0.000	0.000						
9												
10		DRILL_5	1	1	0.000	0.000	C		40	0	35	
11		BUTTON_TOOL_8	1	1	0.000	0.000						
12		ROUGHING_C2 A	1	1	0.000	0.000						
13		FINISHING_C2 A	1	1	0.000	0.000						
14		DRILL_32_C2	1	1	0.000	0.000						
15		ROUGHING_C2 I	1	1	0.000	0.000						
16		FINISHING_C2 I	1	1	0.000	0.000						
17												
18		PLUNGE CUTTER_3_C2	1	1	0.000	0.000						
19		CUTTER_C2_12	1	1	0.000	0.000						

### Benefits



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



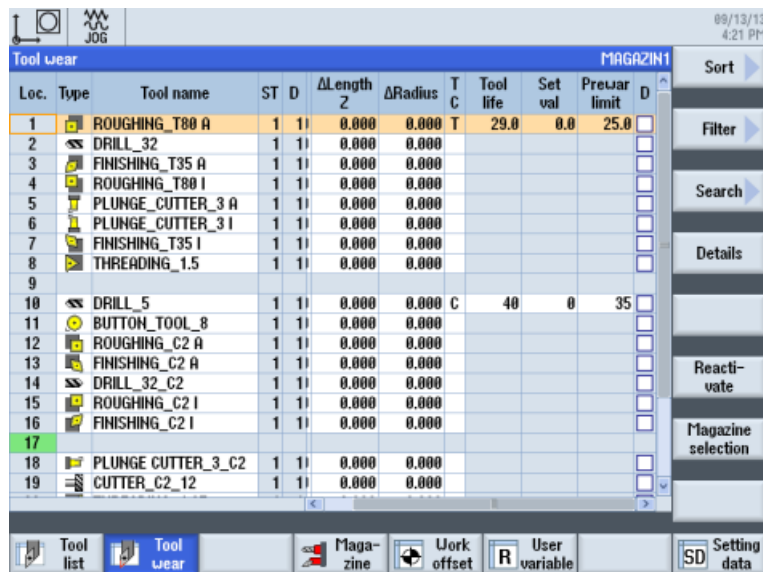
## 5.3 Replacement tools

<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.



Loc.	Type	Tool name	ST	D	ΔLength Z	ΔRadius	T	C	Tool life	Set val	Pre wear limit	D
1		ROUGHING_T80 A	1	1	0.000	0.000	T		29.0	0.0	25.0	
2		DRILL_32	1	1	0.000	0.000						
3		FINISHING_T35 A	1	1	0.000	0.000						
4		ROUGHING_T80 I	1	1	0.000	0.000						
5		PLUNGE CUTTER_3 A	1	1	0.000	0.000						
6		PLUNGE CUTTER_3 I	1	1	0.000	0.000						
7		FINISHING_T35 I	1	1	0.000	0.000						
8		THREADING_1.5	1	1	0.000	0.000						
9												
10		DRILL_5	1	1	0.000	0.000	C		40	0	35	
11		BUTTON_TOOL_8	1	1	0.000	0.000						
12		ROUGHING_C2 A	1	1	0.000	0.000						
13		FINISHING_C2 A	1	1	0.000	0.000						
14		DRILL_32_C2	1	1	0.000	0.000						
15		ROUGHING_C2 I	1	1	0.000	0.000						
16		FINISHING_C2 I	1	1	0.000	0.000						
17												
18		PLUNGE CUTTER_3_C2	1	1	0.000	0.000						
19		CUTTER_C2_12	1	1	0.000	0.000						

### Benefit



- Automatic exchange of identical tools for unmanned operation

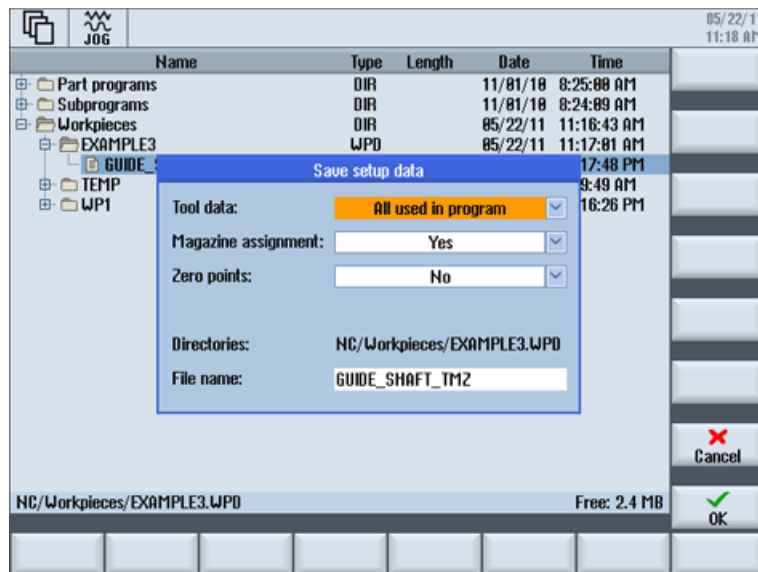
## 5.4 Setup data

<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

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Basic configuration		Basic configuration		Basic configuration

	<b>SINUMERIK 840D sl</b>
	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

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Option: P01		Option: P01		Option: P01

	<b>SINUMERIK 840D sl</b>
	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

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Basic configuration		Basic configuration		Basic configuration

	<b>SINUMERIK 840D sl</b>
	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

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Basic configuration		Basic configuration		Basic configuration

	<b>SINUMERIK 840D sl</b>
	Basic configuration

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

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	not available		Option: P75		Option: P75

	<b>SINUMERIK 840D sl</b>
	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



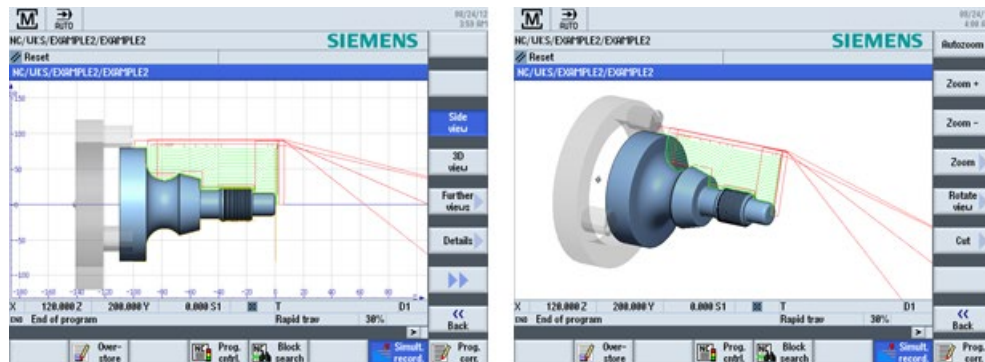
- Optimal control of the program execution, even in complex sequences or machining cycles, especially in single block mode



## 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

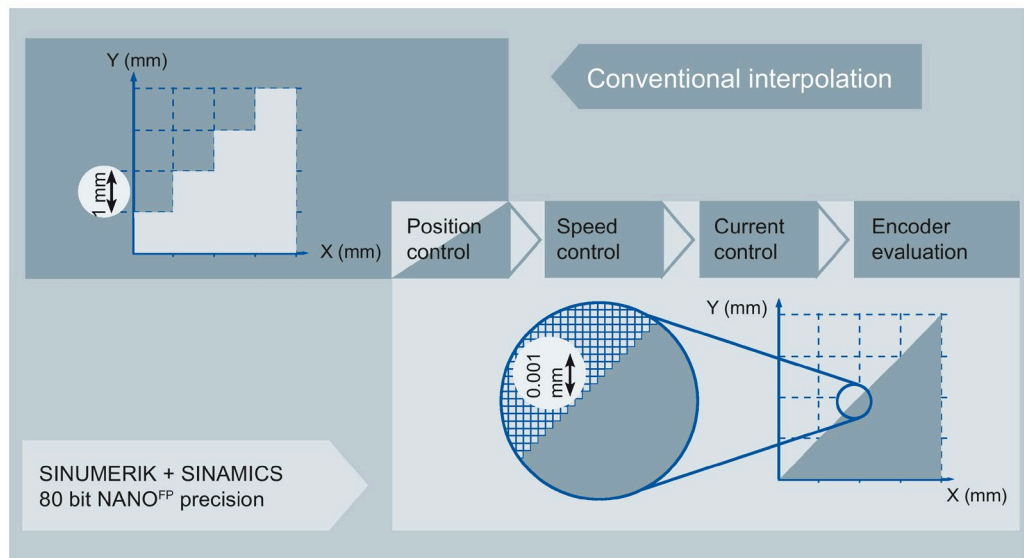
✓	SINUMERIK 828D SW24x	✓	SINUMERIK 828D SW26x	✓	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

✓	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

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	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

	<b>SINUMERIK 840D sl</b>
	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



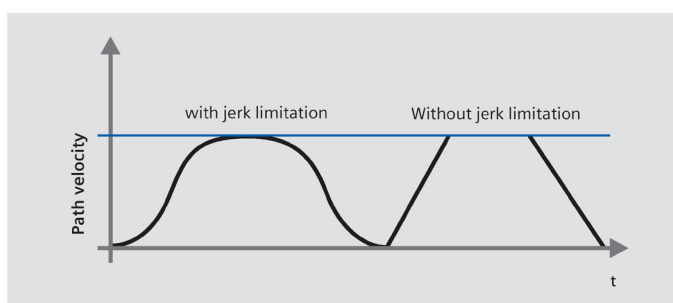
- Minimum block change times for the associated performance versions

## 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

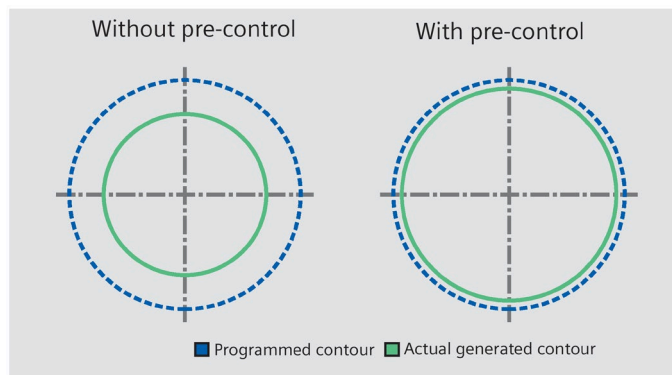
✓	SINUMERIK 828D SW24x	✓	SINUMERIK 828D SW26x	✓	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

✓	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

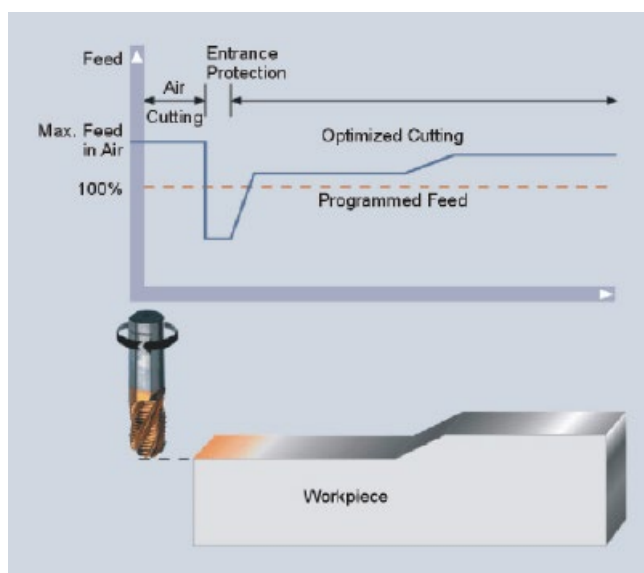
## 8.5 Adaptive Control & Monitoring (ACM)

✓	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.



ACM 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 option "Cross-operational actions" the synchronous action between the compile cycle and the HMI is executed automatically.

### Benefit






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






## 8.6 Contour handwheel

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Option: M08		Option: M08		Option: M08

	<b>SINUMERIK 840D sl</b>
	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

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Basic configuration		Basic configuration		Basic configuration

	<b>SINUMERIK 840D sl</b>
	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

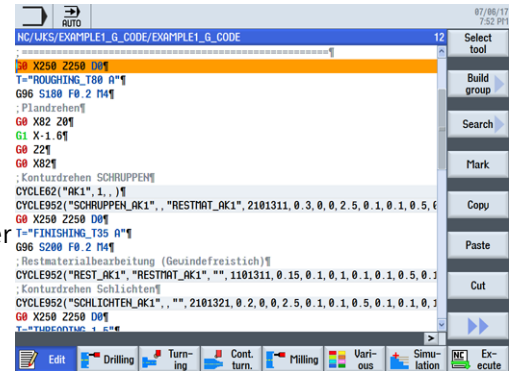
These functions belong to the basic configuration of SINUMERIK Operate.

## 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

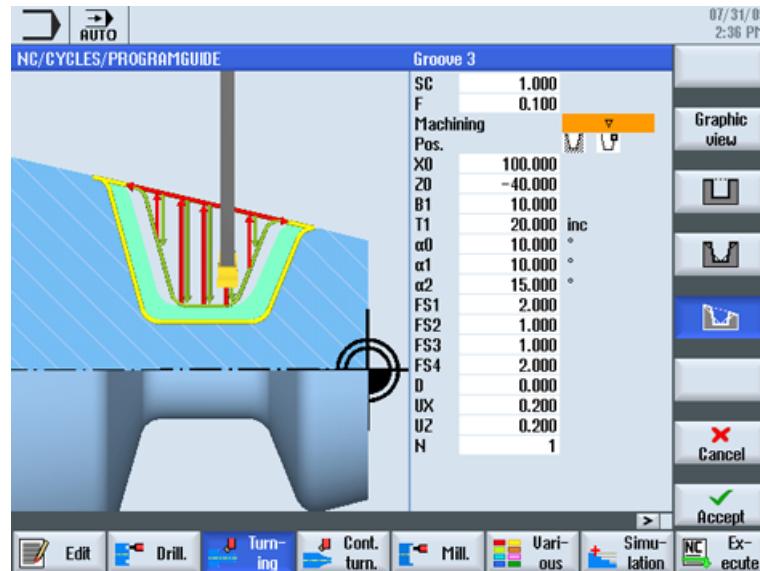


- **Established programming according to DIN 66025**
- **Unbeatable range of commands for flexibility and time saving while programming**

### 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

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Option: P17		Option: P17		Option: P17

	<b>SINUMERIK 840D sl</b>
	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)

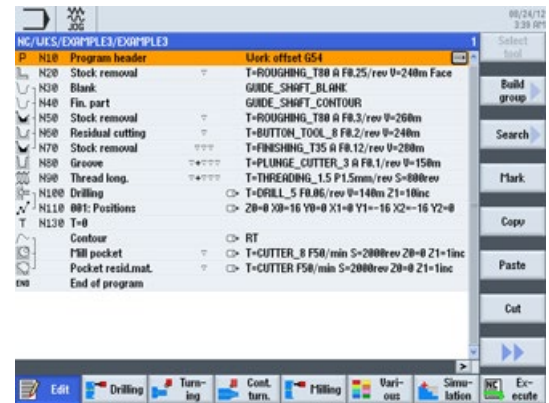
These functions are part of the machining step programming options package in ShopTurn.



## 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

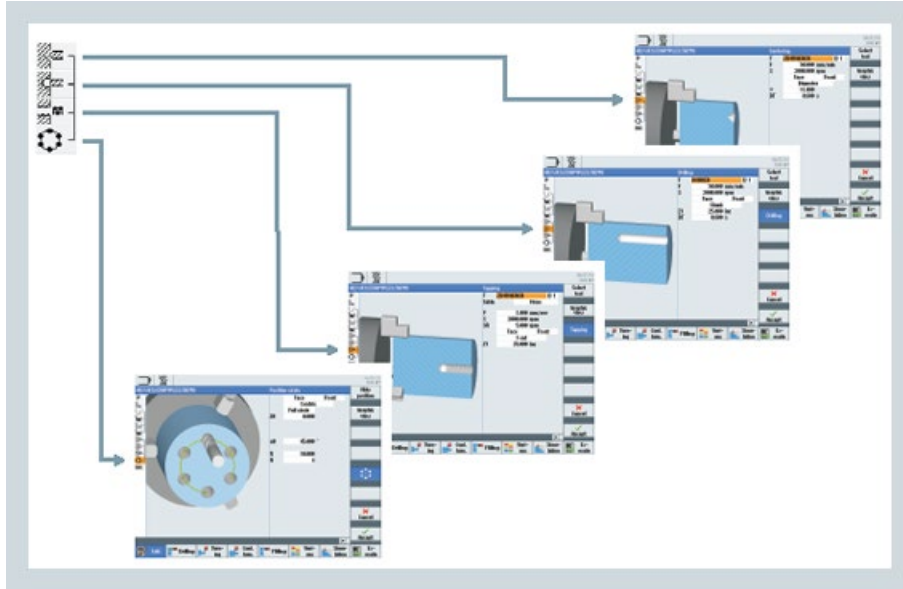


- Intuitive program input, without knowledge of DIN/ISO or Operating Manual
- Compact, clearly arranged machining programs
- Reducing the programming time by graphical input screens and copying / inserting machining steps

### 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

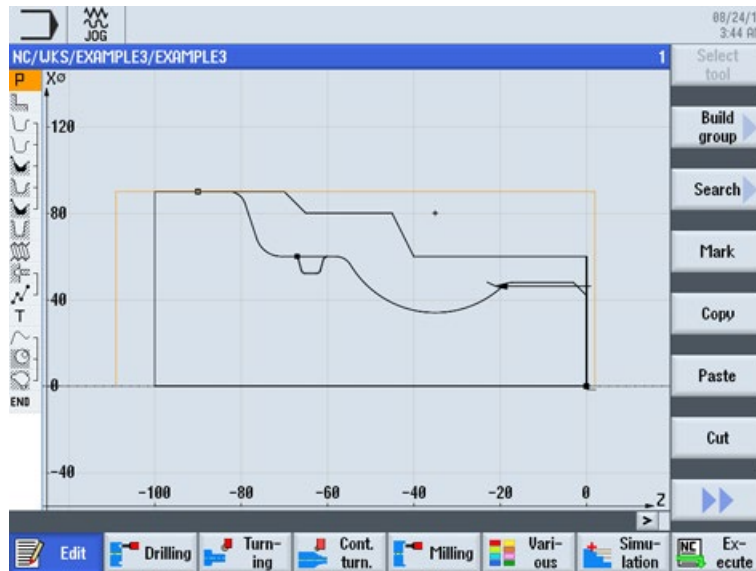


- Reduced programming time due to linking of machining steps

### 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

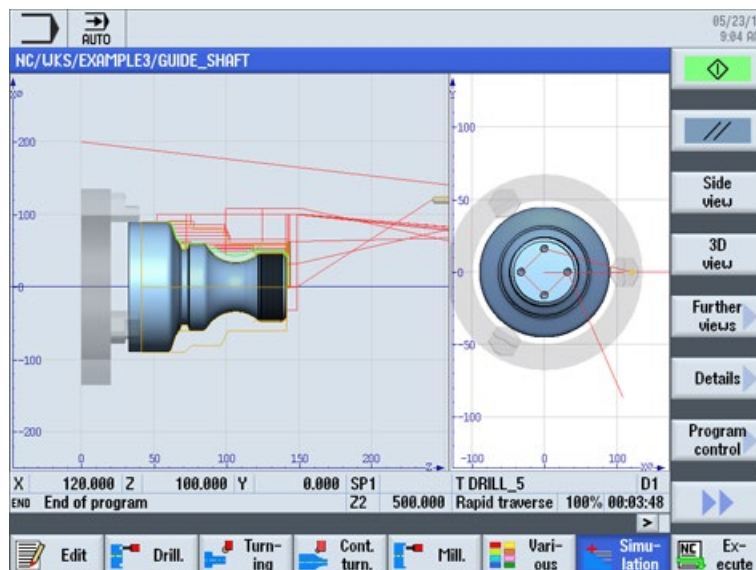
### 10.1 2D simulation

<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



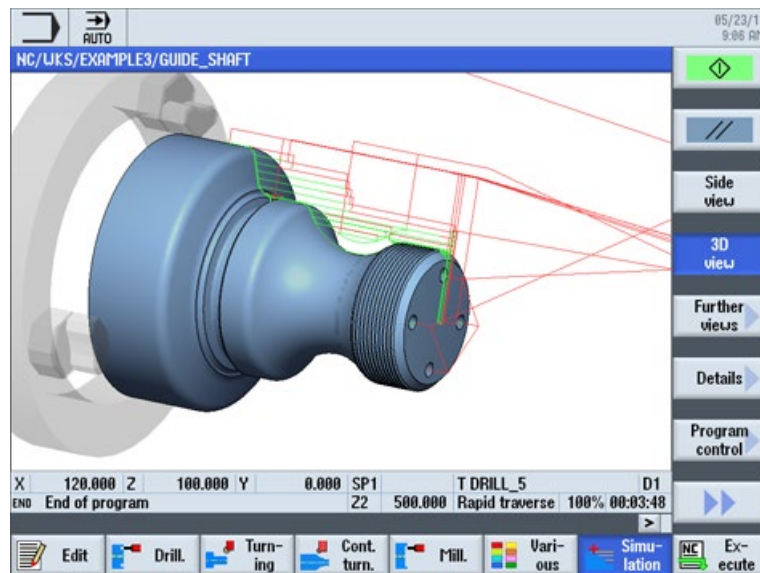
- Maximum process reliability through simulation using real geometry values
- Perfect clarity by showing the workpiece dimensions with a scale
- Parallel simulation (background simulation) is possible in conjunction with the NCU 720 and NCU 730, i.e. simulating a part program while another part program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

## CNC technology cycles

### 11.1 CNC technology cycles for programGuide and ShopTurn

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.



#### Benefits



- Significant simplification of programming, even for complex jobs, using CNC technology cycles
- Continuity of cycles for programGuide and ShopTurn

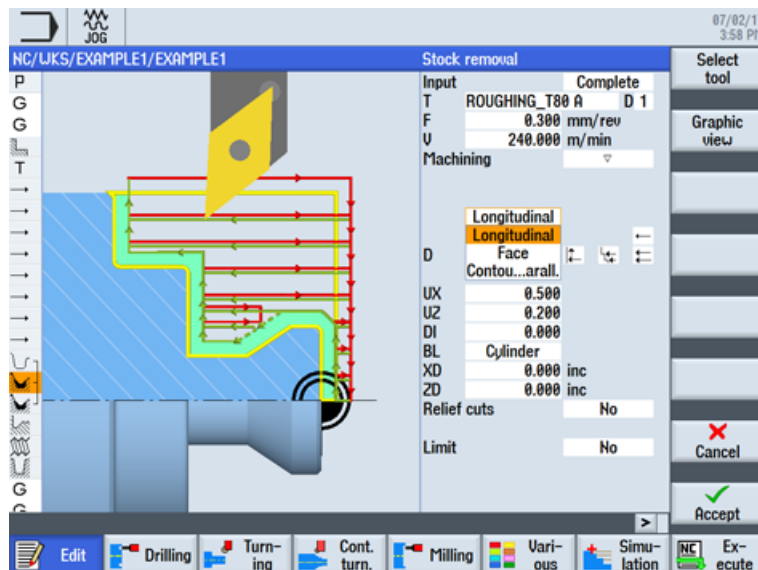
## 11.2 Highlights of machining cycles

### 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

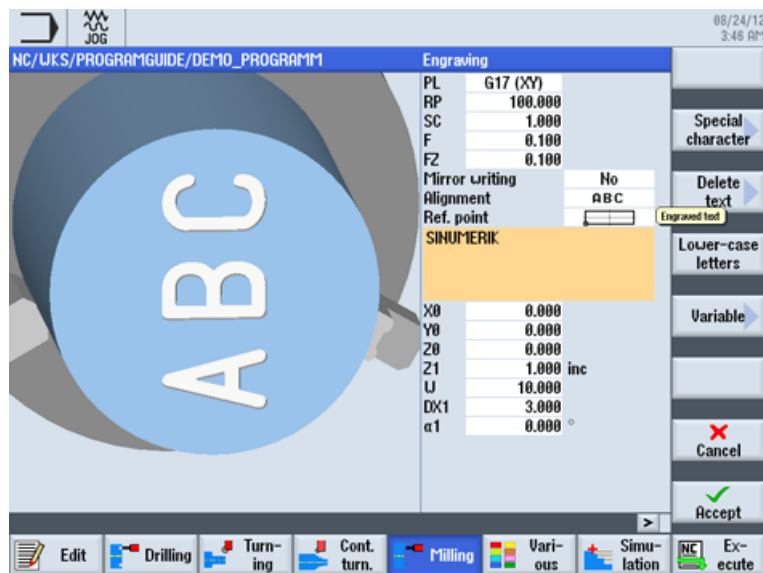
<input checked="" type="checkbox"/>	SINUMERIK 828D SW24x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW26x	<input checked="" type="checkbox"/>	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

<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

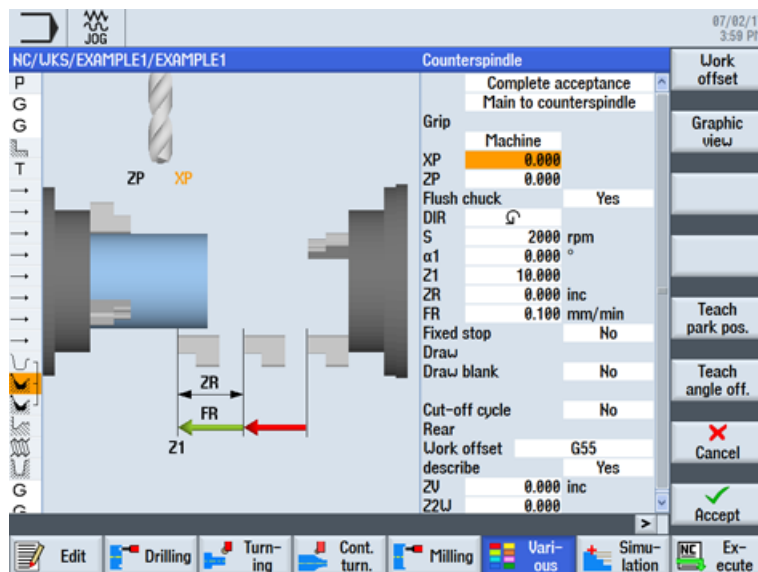
### 11.2.3 Counterspindle cycle

<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 <sup>1)</sup>

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

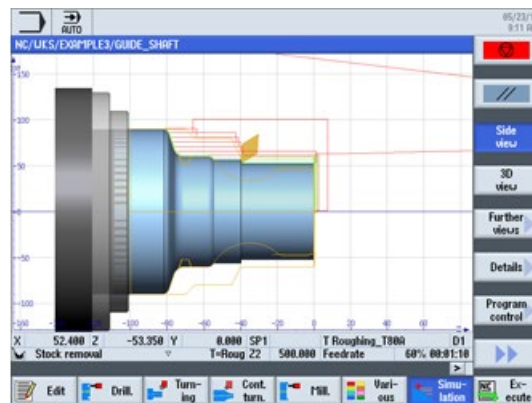
## 11.3 Residual material detection for contour cycles

### 11.3.1 Residual material detection during turning

<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

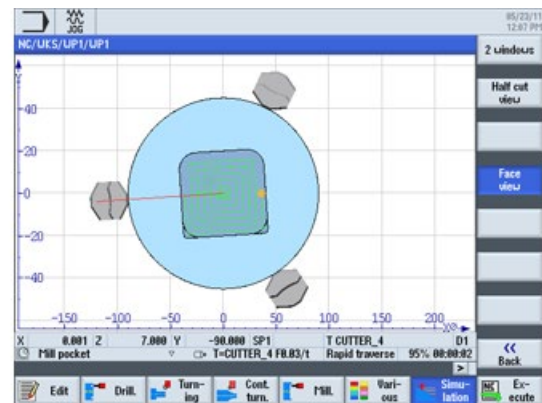
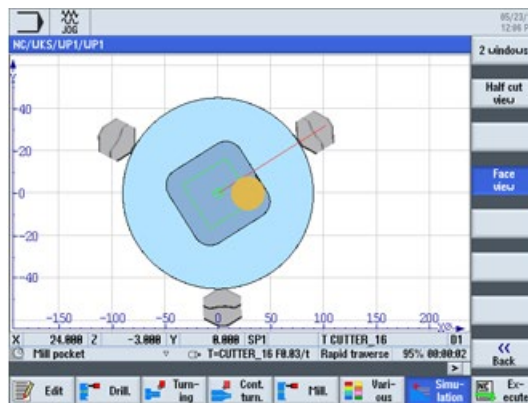
### 11.3.2 Residual material detection during milling

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

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

Contour ranges which do not permit milling with large diameters are automatically identified in the cycle for contour pockets and contour pins. These areas can be selectively machined with a suitable smaller tool, rather than having to use this tool for the entire contour pocket or pin.

If you mill several pockets and wish to avoid unnecessary tool changeovers, remove stock from all the pockets first and then remove the residual material. In this case, you must enter the tool used for removing the residual material from the pocket in the "TR reference tool" parameter.



#### Benefits



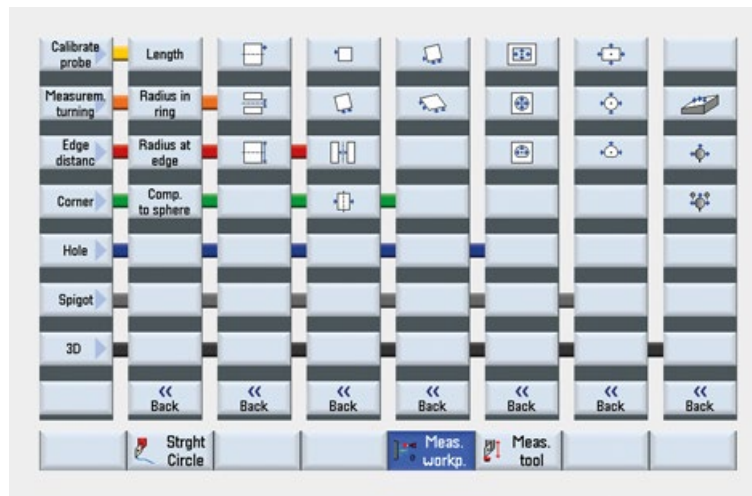
- Shorter machining times through the use of a large tool for the substantial part of the stock removal and a smaller tool for the remaining residual material
- Avoidance of non-cutting movements while achieving extremely simple programming

## 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:



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) .

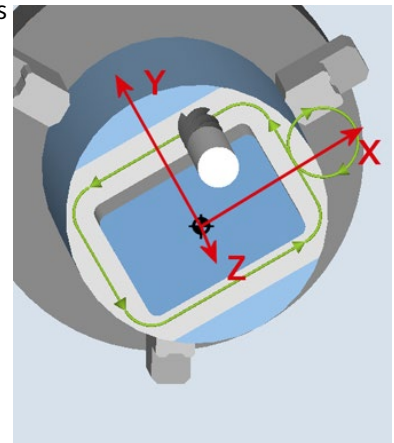
The path movements are conducted with the linear axes X / Z and the rotary axis C.

#### 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				

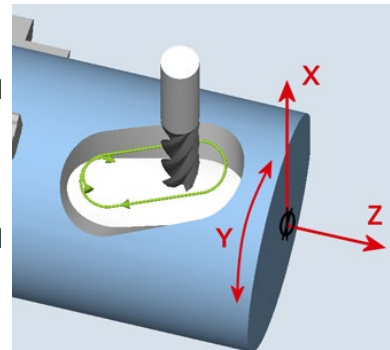
Using the peripheral surface transformation TRACYL, drilling and milling machining can be executed on the peripheral surface of workpieces in the main and counterspindle.

### 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

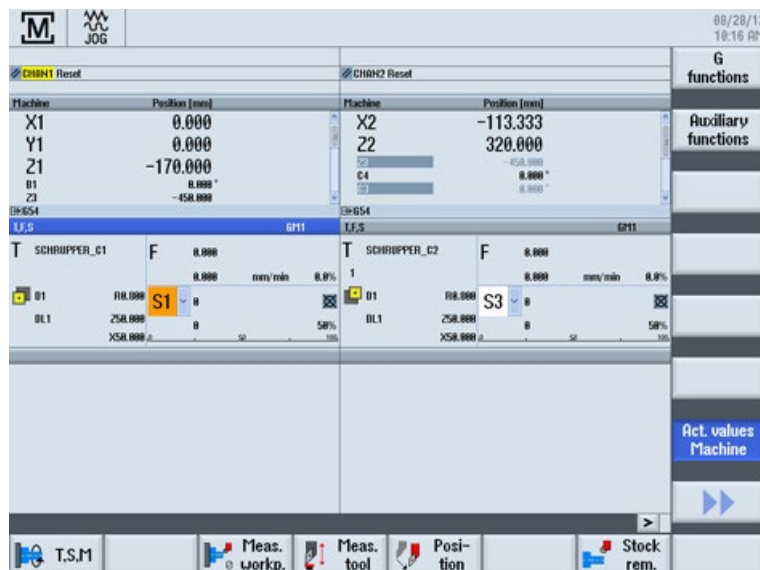
	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	not available		not available		Option: P05

	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.



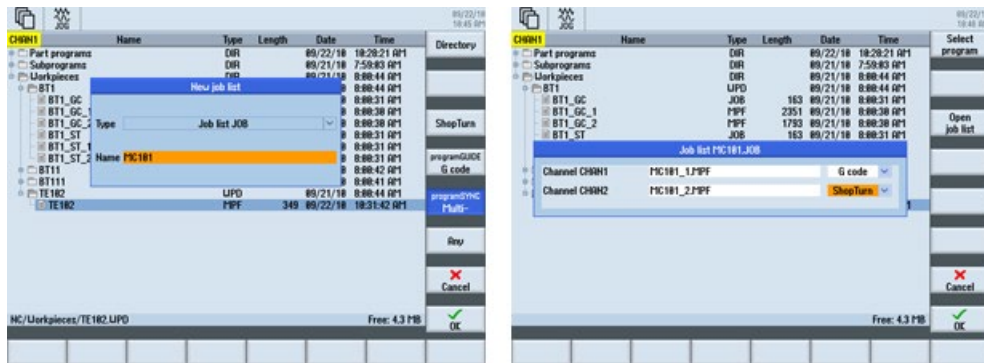
#### Benefit



- 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

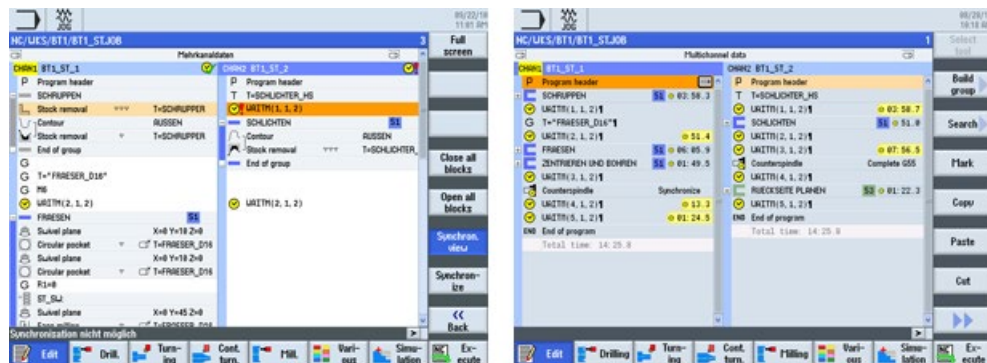


- Simple program management in Windows Explorer style

## 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

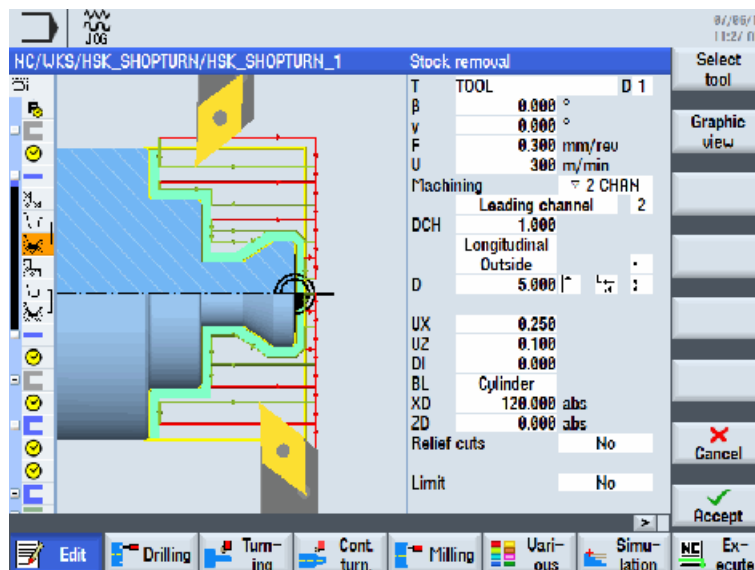
## 13.4 Balance cutting (stock removal)

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.



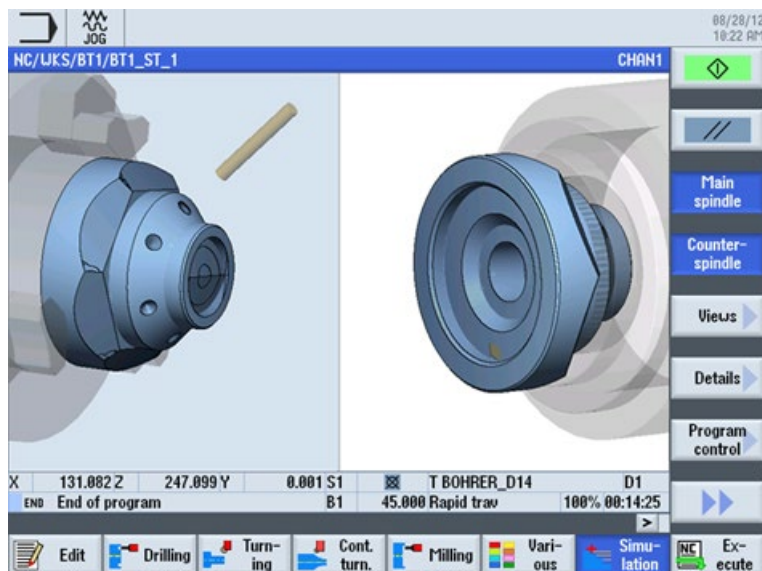
### Benefit



- 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 Integrate 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 permits the connection of handling robots 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 manufacturer-independent interface for the low-effort automation of machine tools.

## 14.2 SINUMERIK Integrate Run MyRobot / Handling

	SINUMERIK 840D sl
	Option: via SISW

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

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.



## 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 MyPrograms (Page 88)
  - Manage MyTools (Page 89)
- Efficiency and flexibility in production
  - Manage MyMachines (Page 90)
  - Analyze MyPerformance (Page 91)
- Machine availability
  - Access MyMachine (Page 92)
  - Optimize MyMachining /AC AUTO (Page 93)
- Improved machining processes
  - Analyze MyWorkpiece (Page 94)

## 15.2 Manage MyPrograms

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

	SINUMERIK 840D sl
	Option: via SISW

Manage MyPrograms permits central management and distribution of CNC programs in machine parks with different CNC types. This reduces the risk of mix-ups, unauthorized changes, crashes, and viruses spread via USB storage devices.




- Can be easily extended with PLM systems (Teamcenter)
- Management of additional production information (e.g. workpiece drawings, clamping instructions) for paperless production


### Benefit



- Efficient network-wide organization and management of CNC programs

## 15.3 Manage MyTools

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

	<b>SINUMERIK 840D sl</b>
	Basic configuration: single machine Option: networked machine, via SISW

Manage MyTools helps you to determine the tool requirement for production orders, mirrors the tool requirement on the magazine assignment of the machine, and supports the tool setup operation.




- Factory-wide management of tools
- Statistical functions for increasing production efficiency
- Link to Teamcenter including utilization of the tool catalogs

### Benefit



- Tool requirements are determined and reflected in the tool inventory at the machine and in the tool warehouse.

## 15.4 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.

## 15.5 Analyze MyPerformance

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Option: via SISW		Option: via SISW		Option: via SISW

	<b>SINUMERIK 840D sl</b>
	Option: via SISW

Analyze MyPerformance calculates the overall equipment efficiency (OEE) and provides important indicators for measures to increase efficiency.

Through the automatic recording of machine data and states, all the data required for optimizing production are provided.




- Detection and analysis of machine states
- Comprehensive analysis options for increasing the Overall Equipment Efficiency (OEE)

### Benefit



- **Transparency about current and even future utilization of the machine park enables on-schedule processing of production orders and contributes to increased efficiency in production.**

## 15.6 Access MyMachine (AMM)

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Option: via SISW		Option: via SISW		Option: via SISW

	<b>SINUMERIK 840D sl</b>
	Option: via SISW

Access MyMachine enables fail-safe remote control and remote monitoring of machine tools worldwide - from simple point-to-point connections in closed networks all the way to a secure Internet connection. Service personnel have access to a wide range of options for fault diagnostics and troubleshooting. This results in faster problem solving and higher machine availability.




- Remote diagnostics in closed networks (AMM Peer to Peer)
- Remote diagnostics via the Internet (AMM Ethernet)
- Unrestricted remote control of the CNC user interface
- Arbitrary file transfer from and to the CNC
- Secure encrypted communication for remote diagnostics via the Internet

### Benefit



- **Faster problem solving and higher machine availability.**

## 15.7 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 option "Cross-operational actions" the synchronous action 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 feed by dynamically adjusting the feedrate!

## 15.8 Analyze MyWorkpiece

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

	SINUMERIK 840D sl
	Option: via SISW

With Analyze MyWorkpiece you can analyze and optimize NC programs and SINUMERIK trace data using modern 3D visualization.

Errors in the NC program are detected at an early stage, which enables optimization by reducing idle times and by prior testing of the workpiece quality by simulation on the machine.

### Benefit



- Analyze MyWorkpiece helps improve productivity and part quality.



## 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.

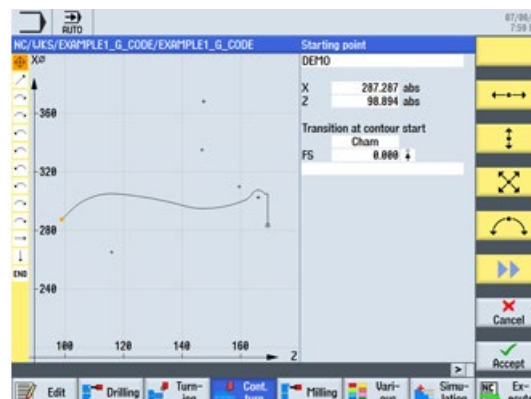
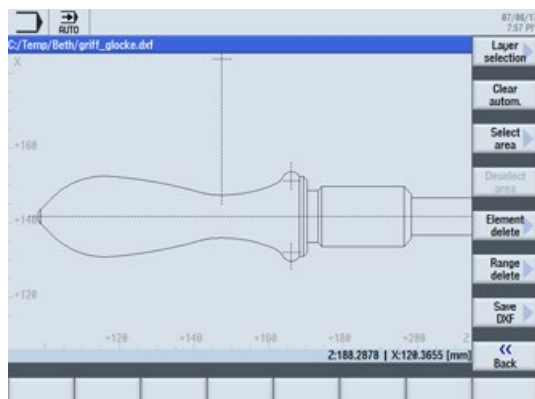
- **DXF Reader in the Program Manager**

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

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Option: C43		Option: C43		Option: C43

	<b>SINUMERIK 840D sl</b>
	Option: C43

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](http://www.siemens.com/sinutrain) ([www.siemens.com/sinutrain](http://www.siemens.com/sinutrain))

### 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

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Option: refer to the cata- log		Option: refer to the cata- log		Option: refer to the cata- log

	<b>SINUMERIK 840D sl</b>
	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.

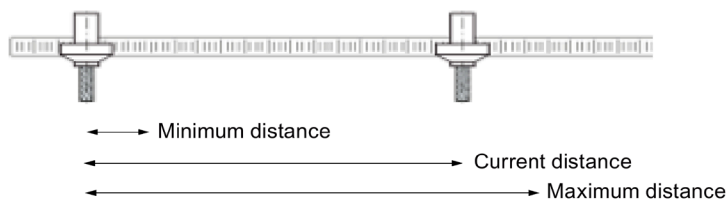
### 17.2.1 Collision protection Axes Run MyCC /PROT

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

With the collision protection Axes 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.

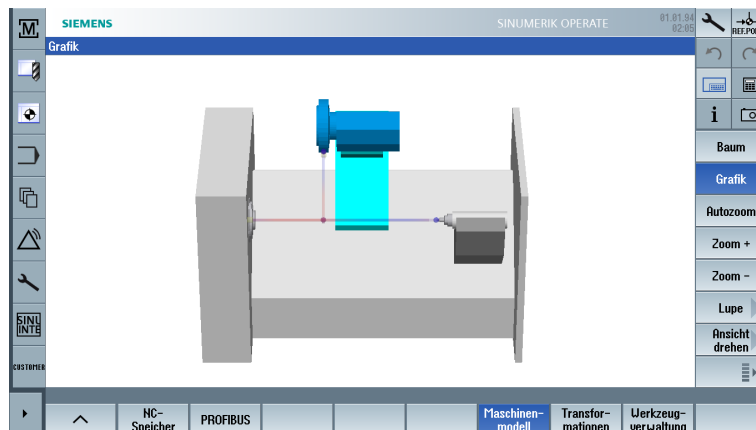
## 17.2.2 Collision Avoidance ECO

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	Option: S03		Option: S03		Option: S03

	<b>SINUMERIK 840D sl</b>
	Option: S03

With the Collision Avoidance ECO option, you can monitor the minimum distance between protection zones. 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 ball
- In the modes JOG, MDI, Automatic
- Single-channel






### Benefits



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

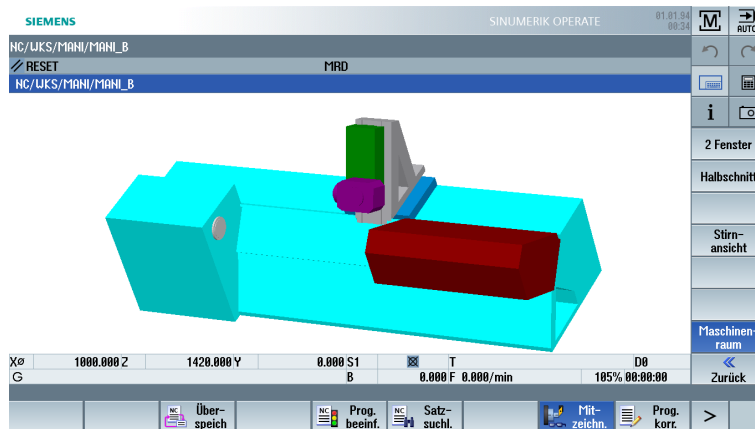
## 17.2.3 Collision avoidance

	<b>SINUMERIK 828D SW24x</b>		<b>SINUMERIK 828D SW26x</b>		<b>SINUMERIK 828D SW28x</b>
	not available		not available		not available

	<b>SINUMERIK 840D sl</b>
	Option: S02

The Collision Avoidance option allows you to monitor the minimum distance 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 Collision Avoidance ADVANCED

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

	SINUMERIK 840D sl				
	Option: S04				

The Collision Avoidance ADVANCED option offers 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 avoidance, 102  
Collision Avoidance ADVANCED, 103  
Collision Avoidance ECO, 101  
Collision protection Axes, 100  
Contour stock removal cycle, 70  
Contour-grooving, 70

## D

Detection of residual material when milling, 74  
DIN/ISO language, 60  
DIN/ISO programming, 60

## E

End face machining, 77  
Engraving cycle, 71

## F

Feed interruption, 70  
Feedforward control, 52

## G

G code editor, 59

## I

In-process measurement, 75

## M

Measure tool, 32  
Monitoring of tool life and workpiece count, 38  
Multi-channel machining  
    Blocks, 81  
    Job list, 80  
    Machine basic screen, 79  
    programSYNC, 80  
    Simulation, 83  
    Wait marks, 81

## O

Online Help, 24

## P

Peripheral surface machining, 78  
Plunge-turning, 70  
Positioning cycle, 35  
Program editor, 59  
Program manager, 41

## R

Replacement tools, 39  
Residual material detection during turning, 73

## S

Safety functions, 99  
Safety Integrated, 99  
Sequence editor, 63  
SIMATIC ITC 1200, 21  
SIMATIC ITC 1500, 21

SIMATIC ITC 1900, 21  
Simulation  
    2D, 67  
    3D, 68  
    Processing time, 68  
Simultaneous recording, 47  
SINUMERIK CNC4you, 97  
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, 96  
Speed control, 51  
Stock removal cycle, 34

## T

Tool management, 37  
TRACYL, 78  
TRANSMIT, 77  
TSM universal cycle, 29