SIEMENS

Overview of controllers for vendors of machine tools

SINUMERIK Operate - Milling

SINUMERIK 840D sl / SINUMERIK 828D

Edition 08/2021

www.siemens.com/sinumerik

SIEMENS

SINUMERIK

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

Control system overview for machine tools' sales people

Valid for:

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

08/2021
A5E41992599B AD

Legal information

Warning notice system

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

⚠**DANGER**
indicates that death or severe personal injury **will** result if proper precautions are not taken.

⚠**WARNING**
indicates that death or severe personal injury **may** result if proper precautions are not taken.

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

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

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

Qualified Personnel

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

Proper use of Siemens products

Note the following:

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

Trademarks

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

Disclaimer of Liability

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

Siemens AG
Digital Industries
Postfach 48 48
90026 NÜRNBERG
GERMANY

A5E41992599B AD
Ⓟ 08/2021 Subject to change

Preface

Scope of validity

This document provides you with an overview of the range of functions included in **SINUMERIK 828D** and the **SINUMERIK 840D sl** with **SINUMERIK Operate V4.9** for milling 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 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
E-mail:	xxx@machinemanufacturer.com

Service

Phone:	+49 xxx xxx
Fax:	+49 xxx xxx
E-mail:	xxx@machinemanufacturer.com

Homepage:

http://www.machinemanufacturer.com

SINUMERIK user support worldwide contact

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

Homepage:

For further information please visit ...

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

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

Table of contents

Compact overview

1

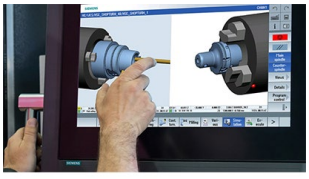
Siemens Machine Tool Systems - a strong partner for the machine tool world ...

Siemens Machine Tool Systems portfolio

The SINUMERIK product family provides perfect solutions for all machine concepts – from price-optimized CNC entry-level machines, to standardized machine concepts, all the way to modular premium machine concepts. (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 25)

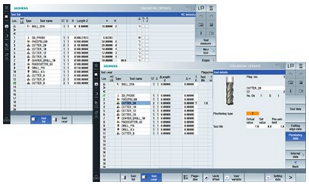


Setup functions "Intelligent-JOG"

Functions for setting up the machining process are of central importance in small-batch production with universal milling machines. SINUMERIK Operate sets standards for these "functions of daily life". Thanks to an intelligent JOG mode and intuitive tool management, all of the typical setup functions feature interactive, graphical support. (Page 31)

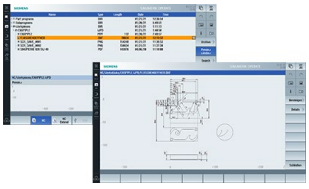
Tool management - powerful but nevertheless easy to use

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 43)



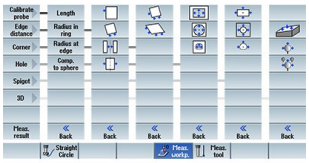
Data management like on a PC

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 49)



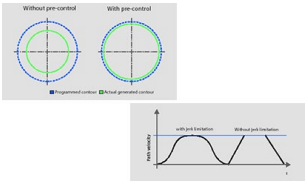
CNC operation in automatic mode (AUTO)

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 51)



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 59)

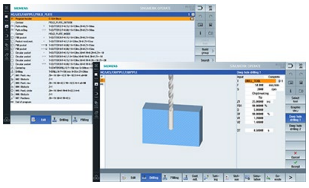


Freeform surface machining - the stress test for every CNC

The machining of freeform surfaces means processing of extremely large quantities of CNC sets in the shortest possible time. Modern CNCs offer special functions to meet this challenge. (Page 67)

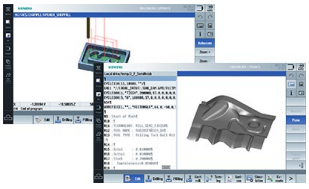
CNC programming methods - optimally prepared for all production tasks

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 73)



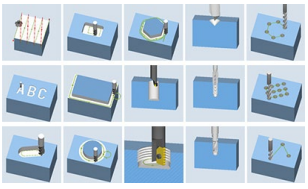
Workpiece visualization - more safety through simple and fast control

Realistic 2D and 3D simulation and the mold-making quick view offer reliability regarding programming and quotation pricing. (Page 83)



CNC technology cycles - the little helpers for daily CNC programming

Irrespective of whether you use programGUIDE or ShopMill – in either case the full range of technological cycles, position patterns and geometries is available to you. (Page 87)

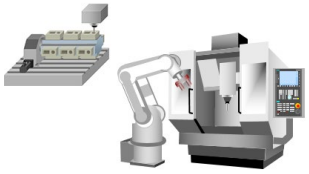


Complete machining - CNC performance in any machining plane, with any tool orientation

Powerful kinematic transformations enable machining in any desired plane or with any tool orientation – without restrictions in the calculation of tool offsets and without compromising on ease of operation and programming. (Page 97)

Automation - the fully automated workpiece flow

Different automation concepts, tailor-made for the respective milling machine concept, automate the workpiece flow and thus increase the economic efficiency in production. (Page 105)

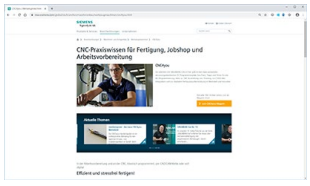


Digitalization on the shop floor

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 109)

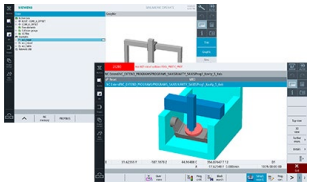
Tools and information

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 125)



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 129)



System overview

2

SINUMERIK 828D and SINUMERIK 840D sl, the easily understandable and intuitive SINUMERIK Operate programming interface, and the SINUMERIK MDynamics milling technology packages provide a tailored solution for all CNC milling machines and machining centers used worldwide.

SINUMERIK Operate

The characteristic features of SINUMERIK Operate:

- ShopMill and ShopTurn are integrated into the SINUMERIK Operate user interface
- Intuitive and clear operation and programming, including Animated Elements
- Display in the modern Windows style
- Powerful functions covering all aspects of setting up, programming, tool and program management

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

SINUMERIK MDynamics

SINUMERIK MDynamics - optimally equipped for all milling applications - from tool making and jig construction, including the machining of free-form surfaces, through to the milling of structural parts:

- Powerful CNC hardware and intelligent CNC functions for a cost-effective package price
- Very simple to operate
- NX CAD/CAM and SINUMERIK MDynamics provide an integrated, optimally matched CAD/CAM/CNC process chain
- Technology know-how concerning milling in all industry sectors, e.g. automotive, aerospace or workshop manufacturing

SINUMERIK 828D and SINUMERIK 840D sl with SINUMERIK MDynamics so offer a control configuration that covers all required application areas for using the milling technology without subsequent commissioning effort:

- Easy-to-use interface for all machine functions
- DIN/ISO programming on the machine
- Graphic programming
- Measuring functions for workpieces and tools

2.1 SINUMERIK 828D

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

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

Software variant 28x

- Up to 8 axes/spindles
- Up to 2 machining channels (T, M, G)
- 768 tools, 1536 cutting edges
- 10 MB user memory
- Additionally up to 2 auxiliary axes

Software variant 26x

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

Software variant 24x

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

You can find further information in catalog NC 82

Benefits

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

2.1.1 Data storage - SINUMERIK 828D

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

* Option P75 not available for SW 24x

2.1.2 Handheld unit

Mini handheld unit

You can install the mini handheld unit pictured below in setup mode.

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 expanded functionality for digitalization and integration in automation concepts**

2.2.1 Data storage - SINUMERIK 840D sl

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

1) HMI user memory, alternative to PCU

2.2.2 Panels

SINUMERIK OP 08T

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

SINUMERIK OP 010

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

SINUMERIK OP 010S

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

OP 010C

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

OP 012

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

SINUMERIK OP 015A

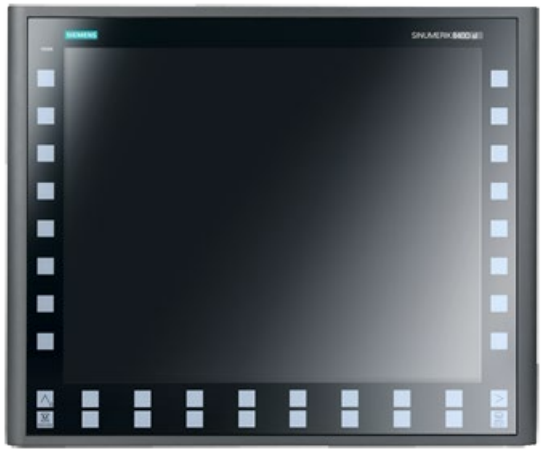
- Operator panel 380 mm wide,
  15" TFT display
  (resolution 1024 × 768 pixels)
- Version with membrane keyboard with 62 keys
- With USB interface at the front
- Integrated mouse

SINUMERIK OP 015 black

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

Note: see also Chapter Multi-touch operation, basic configuration (Page 27)

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 Multi-touch operation, basic configuration (Page 27)

SINUMERIK blackline plus

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

*) available as a customer-specific version

2.2.3 Handheld units

SINUMERIK HT 2

The SINUMERIK HT 2 handheld terminal permits the manual operation of machine tools if you need to remain mobile during operation (e.g. for setup activities). It has been developed specifically with the focus on easy handling, ruggedness and to address the actual requirements met in practice.

SINUMERIK HT 8

The mobile SINUMERIK HT 8 handheld terminal combines the functions of an operator panel and a machine control panel in a single device.

- Fully graphic 7.5" TFT color display
- Mobility for operator control and monitoring
- Operation via touch screen, membrane keys and touch pen
- Emergency stop button and 2 enabling buttons for left-handed and right-handed operators
- Simple insertion or removal during operation
- Rugged, compact and ergonomically designed

SINUMERIK HT 10

All application-specific functions are displayed on the touch-sensitive display of the SINUMERK Handheld HT 10. A particular function is initiated by touching the appropriate location in the display with the finger. The front glass is hardened.

- 10" multi-touch display with LED backlight
- Override rotary switch with 19 positions
- Handwheel with 50 pulses/revolution
- There are 7 keys on the front of the HT 10:
  - -
  - +
  - RAPID
  - U (User button)
  - CYCLE STOP
  - CYCLE START
  - RESET
- Emergency stop button and acknowledgment button (3-position switch)

Mini handheld unit

The following mini-handheld unit is suitable for the machine set-up:

CNC operation with SINUMERIK Operate

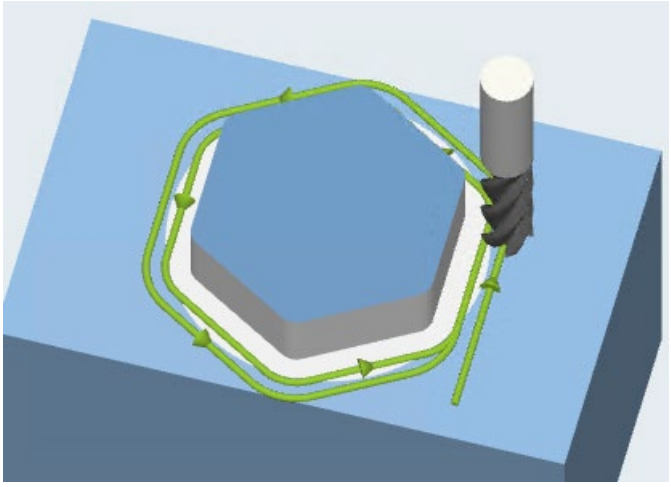
3

3.1 Animated elements

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

✓	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. For instance, the difference between chip breakage and chip removal when drilling or the precise probe sequence for a corner measurement can be shown.



Benefits

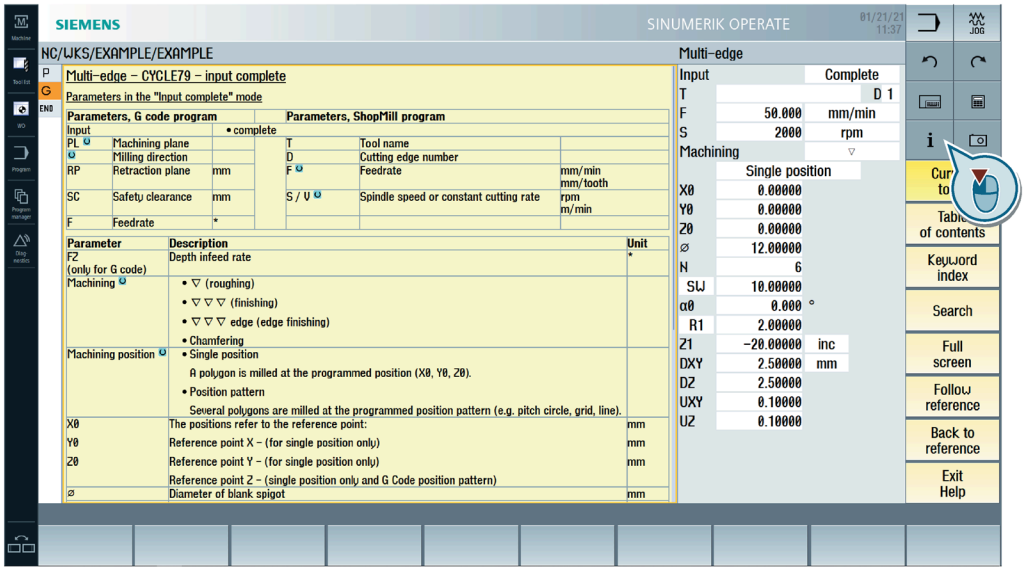
- **Process reliability during the setup**
- **Increased reliability during program input by easily understood depiction of selection options**
- **This results in improved efficiency and increased availability of the machine**

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

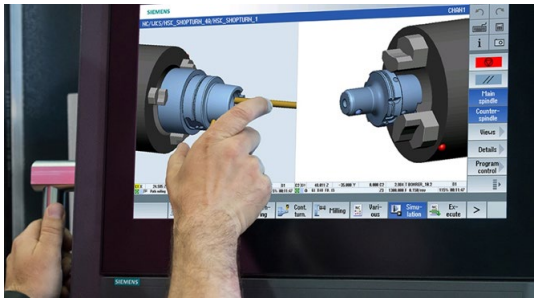
3.3.1 Multi-touch operation, basic configuration

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

☑	SINUMERIK 840D sl
	Basic configuration

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

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



Extract from the multitouch operation gestures:

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

Benefit

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

3.3.2 Multitouch operation with sidescreen

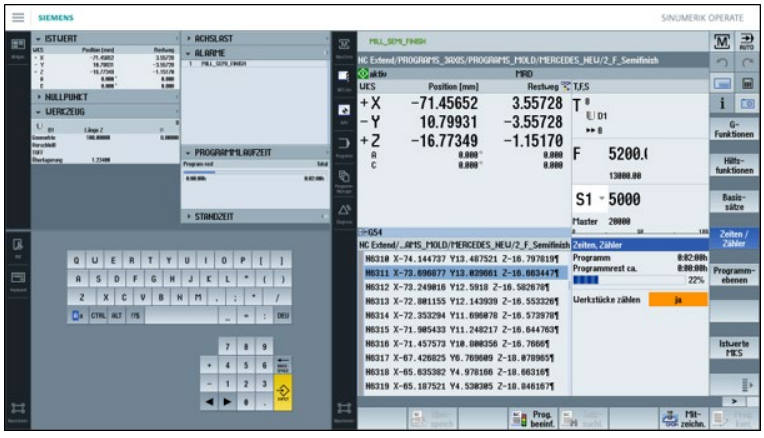
☑	SINUMERIK 828D SW24x	☑	SINUMERIK 828D SW26x	☑	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

☑	SINUMERIK 840D sl
	Basic configuration

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

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

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



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

Benefit

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

3.3.3 SINUMERIK Operate Display Manager

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

	SINUMERIK 840D sl				
	Option: P81				

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

Partitioning of the display area into three or four areas.

Example:

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

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

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

Benefit

- **Effective use of large screens with individually configurable contents.**

3.4 Shortcuts

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

	SINUMERIK 840D sl
	Basic configuration

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

CTRL + A	Select all (editor functionality)
CTRL + C	Copy
CTRL + V	Paste
CTRL + X	Cut
CTRL + I	Calculation of the time from/to line/block
CTRL + L	Language selection
CTRL + M	Maximum simulation speed
CTRL + P	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

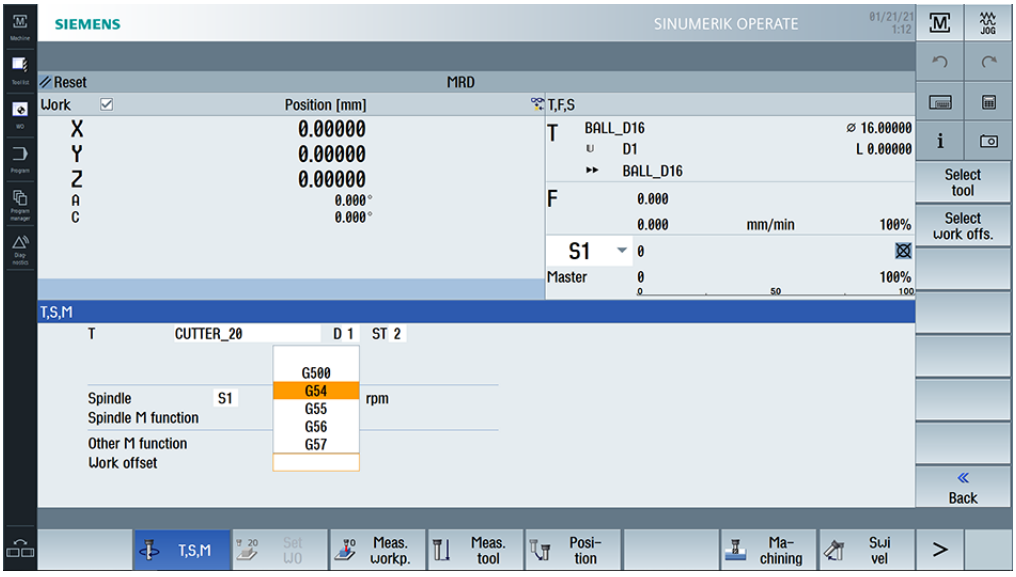
4.1 TSM universal cycle

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

✓	SINUMERIK 840D sl
	Basic configuration

A universal cycle is available in setup mode for the most commonly used machine functions. These include:

- Tool change, also replacement 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
- Selection of the machining plane



Benefit

- **User-friendly manual input function with dialog prompting**

4.2 Work offsets

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

	SINUMERIK 840D sl
	Basic configuration

The following work offsets are possible:

- Settable work offsets:

  It is possible to enter as many as 100 work offsets (G54 to G57, G505 to G599), offset coordinates, angles and

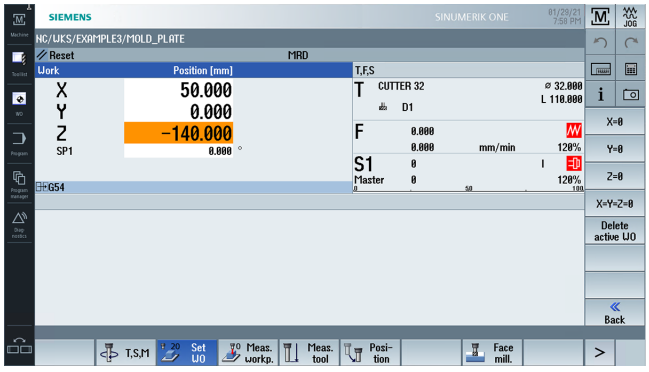
  scaling factors.

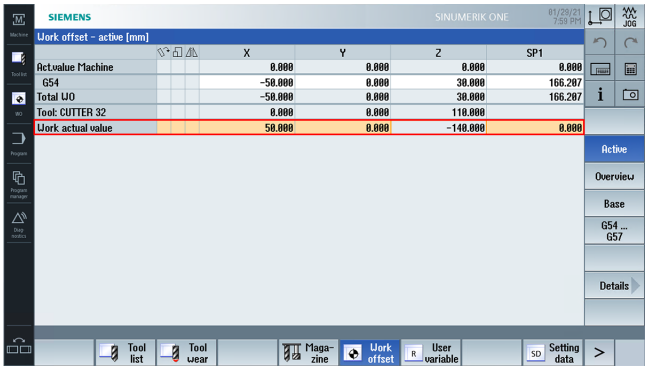
- Programmable work offsets:

  The programmable work offsets allow you, for example, to work with different work offsets for repetitive machining operations at different positions on the workpiece.

- External work offsets:

  Axis-related linear work offsets can also be activated via the PLC user software.





Benefits

- **Flexible machining thanks to a large number of adjustable work offsets**
- **User-conform understandable representation of the number of work offsets**

4.3 Measuring a workpiece

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration/P16		Basic configuration/P16		Basic configuration/P16

	SINUMERIK 840D sl
	Basic configuration

The workpieces can be measured as follows:

- Edge finder, dial gauge, reference tool
- 3D switching probe

The following measuring cycles are available:

- Calibrate probe
- Point measurement for edges
- Orienting the edge (angle)
- Inner/outer corner (3 or 4 points)
- Orienting the edge by means of 2 holes/spigots
- Rectangular or circular pockets, rectangular or circular spigots
- Center point of 3 or 4 holes or spigots
- Orienting the plane with three points

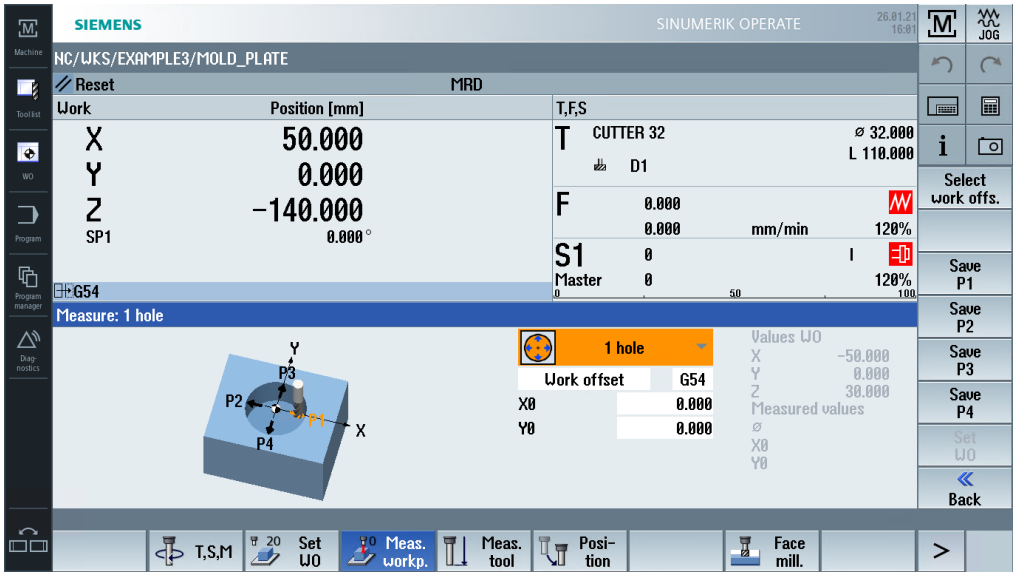
Note

Extended operating functions for SINUMERIK 828D

The basic configuration of SINUMERIK Operate includes the following measurement variants: set edge, align edge, right-angled corner, 1 hole, 1 circular spigot and rectangular spigot.

For further measurement variants, you need the option Extended operating functions, P16.

The measurement results can be output in a measuring log (see Chapter Logging measurement results in JOG (Page 36)).



Benefits

- **Time saving due to user-friendly determination of the workpiece's clamping position instead of orienting the workpiece by hand**
- **The measurement results can be output in a measuring log**

4.4 Measuring a tool

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

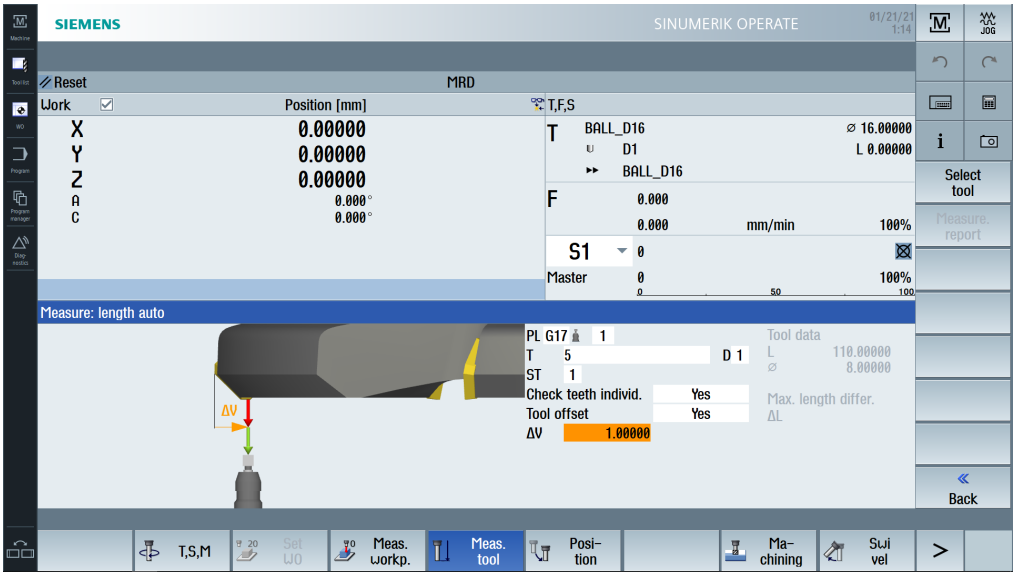
☑	SINUMERIK 840D sl
	Basic configuration

The tool compensation values can be directly determined in setup mode.

The following variants are supported:

- Manual or switching probe
- Scratching with tool at known workpiece geometry

The measurement results can be output in a measuring log (see Chapter Logging measurement results in JOG (Page 36)).



Benefit

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

4.5 Logging measurement results in JOG

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

☑	SINUMERIK 840D sl
	Basic configuration

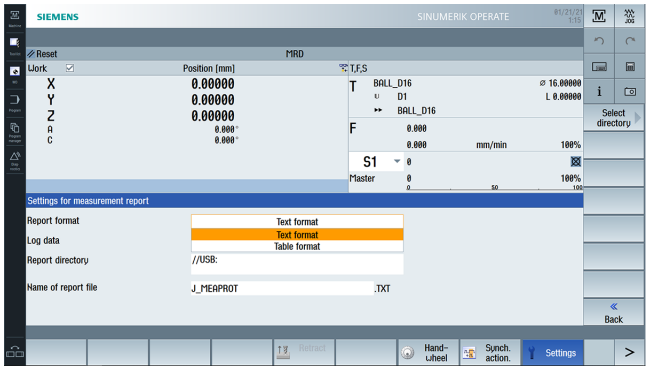
The results for measuring in JOG can be logged. The standard log contains the measurement results of the most recently performed measuring method.

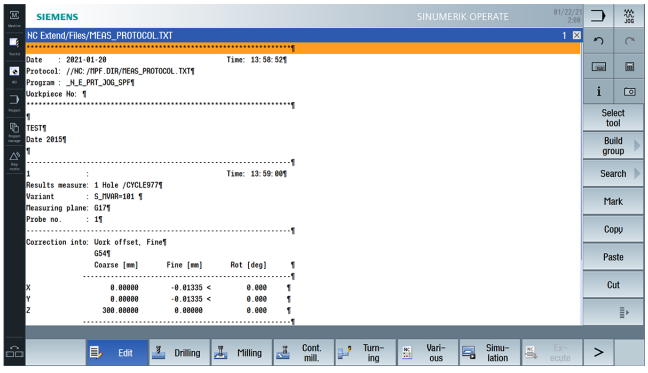
The function is available as milling technology for the workpiece and tool measurement.

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 Face milling cycle

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

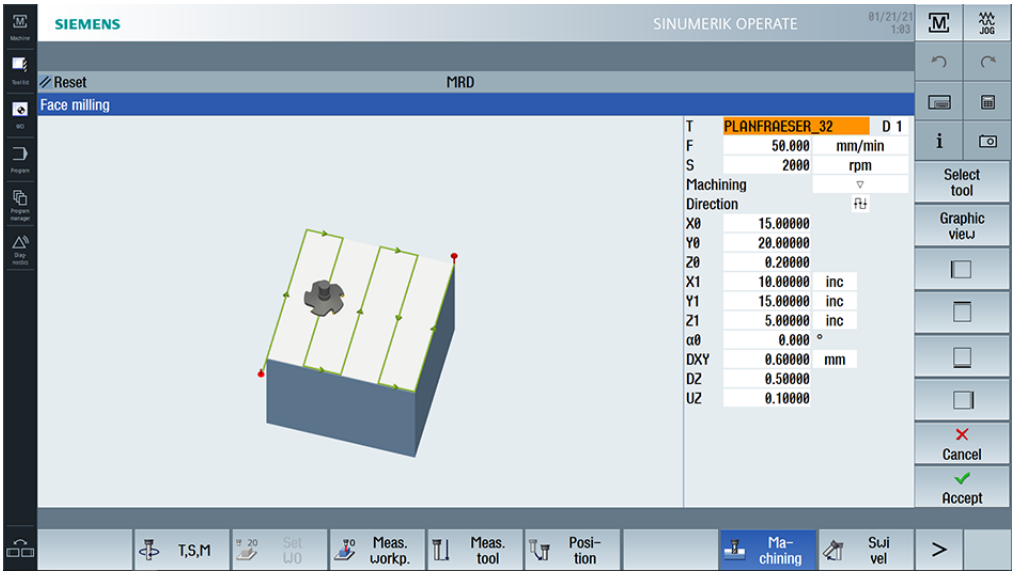
☑	SINUMERIK 840D sl
	Basic configuration

A face milling cycle for preparation of the blank for machining is available directly in setup mode. You can select the tool directly from the list. Input the feedrate and the spindle speed / cutting speed.

You can specify the following parameters:

- Machining strategy and direction
- Machining limitations

The input values are retained even after switching off and on again, so that users can always restart their face milling operation with minimum effort.



Benefit

- **Preparation of workpiece without having to create a part program**

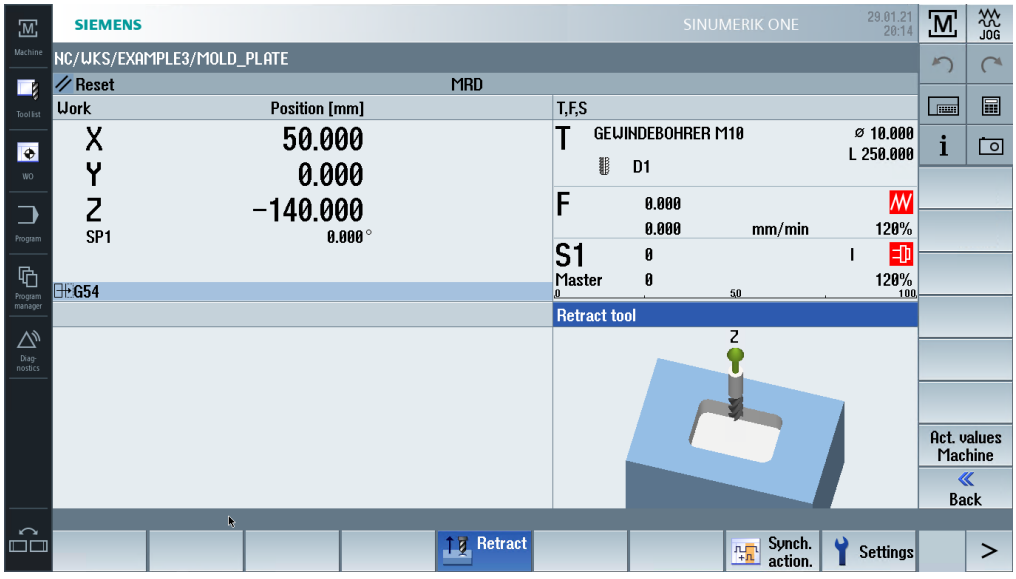
4.7 Retract

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

☑	SINUMERIK 840D sl
	Basic configuration

The Retract function supports the manual retraction of the tool after an interruption. In the JOG mode, after the interruption, the tool can be retracted from the workpiece in the tool direction.

Typical applications include machining while deploying the CYCLE800 swivel cycle, 5-axis machining with TRAORI as well as tapping without compensating chuck.



Benefit

- **Machining can be continued at the point of interruption**

4.8 Stop and retract (ESR)

4.8.1 Stop and retract (ESR) - Drive-autonomous

☑	SINUMERIK 828D SW24x	☑	SINUMERIK 828D SW26x	☑	SINUMERIK 828D SW28x
	Option: M60		Option: M60		Option: M60

☑	SINUMERIK 840D sl
	Option: M60

The drive-autonomous stop and retract (ESR) function offers the possibility of flexibly responding when a fault situation occurs, irrespective of the higher-level control (NC):

For this purpose, the following axial functions can be configured in the drive:

- Retract
- Extended stop
- Generator operation

The drive-autonomous responses are automatically initiated in fault situations. The triggering of the drive-autonomous responses can also be realized user-specific via the part programs or synchronized actions from the higher-level control. As the stopping and retraction motion of the drive-autonomous ESR are purely axial, in contrast to the control-controlled ESR, couplings are not taken into account.

Benefits

- **Faster, situation-conform stop and retraction of axes after a power failure**
- **Stopping and retraction motions in the drives even when they can no longer be specified from the control, e.g. as a result of a communication failure**
- **Fast resumption of the part program thanks to the block search at the point of interruption**

4.8.2 Extended Stop and Retract ESR - CNC-controlled and drive-autonomous

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

	SINUMERIK 840D sl
	Option: M61

As well as the drive-autonomous stop and retract function, the CNC-controlled stop and retract function is also available. To permit smooth interpolated retraction on the path or contour, the path interpolation can be processed further for a definable period following the triggering event.

The retraction axes are subsequently traversed in synchronism to an absolute or incremental position as programmed. These functions are primarily used for gearing and grinding technologies.

Benefits

- **Faster, situation-conform stop and retraction of axes after a power failure**
- **Safe stopping, also of the safety axes**
- **Fast resumption of the part program thanks to the block search at the point of interruption**

4.9 Swiveling in setup mode

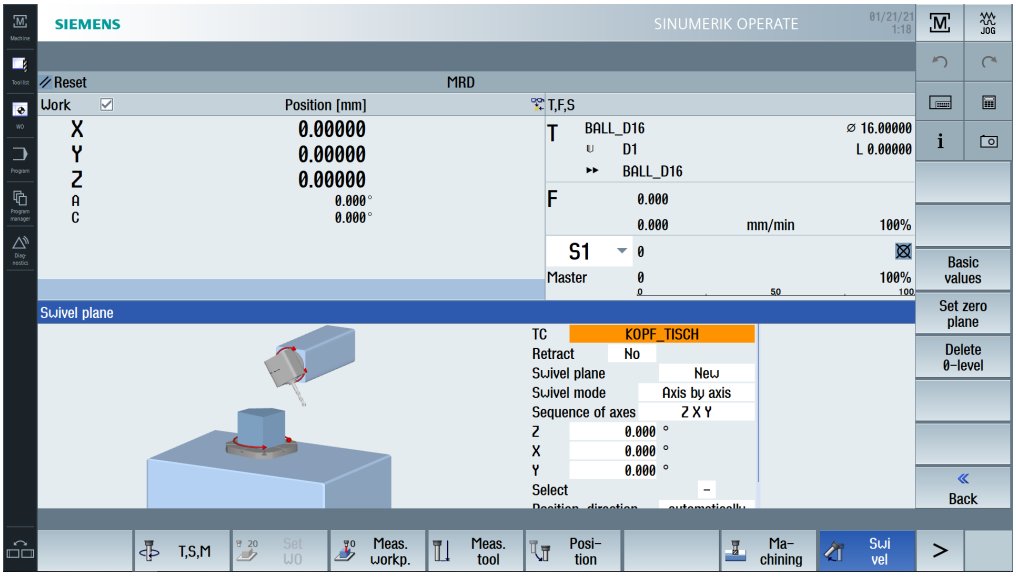
✓	SINUMERIK 828D SW24x	✓	SINUMERIK 828D SW26x	✓	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

✓	SINUMERIK 840D sl
	Basic configuration

You can swivel the machining plane to any angle in setup mode:

- Machining inclined surfaces
- Measure with inclined tool or table

The plane can be swiveled directly including rotation of coordinates or axial swiveling. Using the initial setting softkey, you can traverse the rotary axes of the swivel data set to the initial position. Here, you can select between with and without retraction.



Benefits

- **Swivel the machining plane in setup mode by dialog**
- **Simple setup of the workpiece for machining with swivel axes**

4.10 Manual machine

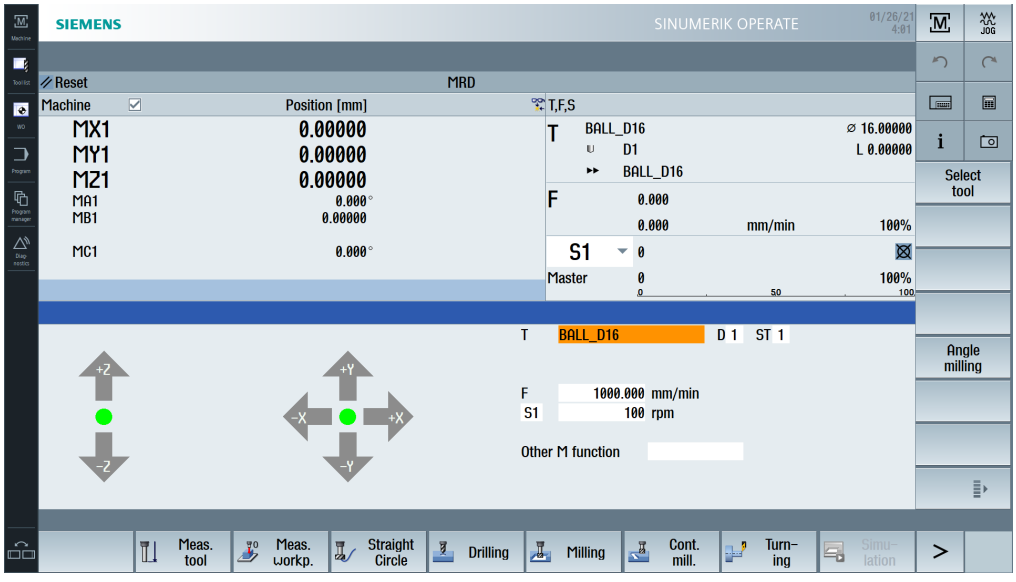
	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: P17		Option: P17		Option: P17

	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:

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



Benefit

- **Simple and intuitive operation of cycle-controlled milling machines**

Tool management

5

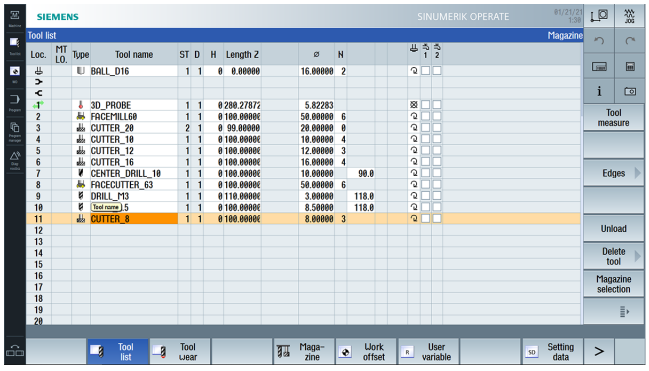
5.1 Tool table

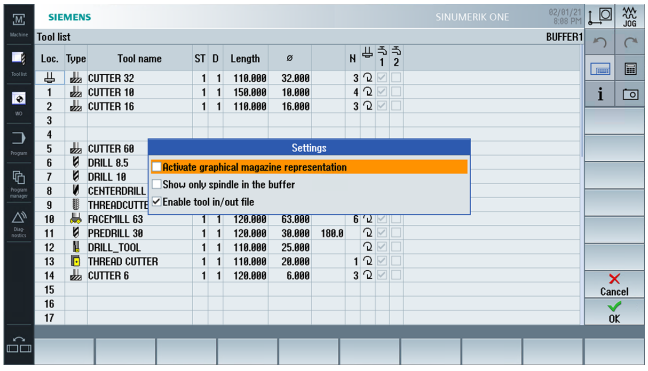
☑	SINUMERIK 828D SW24x	☑	SINUMERIK 828D SW26x	☑	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

☑	SINUMERIK 840D sl
	Basic configuration

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

- Tools are assigned to the desired magazine locations with the load function.
- For each tool, you can store the following data:
  - Tool type: e.g. face milling cutter, taps and 3D probes
  - Clear tool name in plain text, example: CUTTER_HEAD_63MM
  - Max. of 9 cutting edges per tool
  - Tool length and diameter
  - Nose angle for drills or number of teeth for milling tools
  - Spindle direction and coolant (level 1 and 2) and up to four additional functions
- Direct transfer of the tool from the list in the program or for measurement
- You can select multiple tools and load, unload or delete them
- 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**

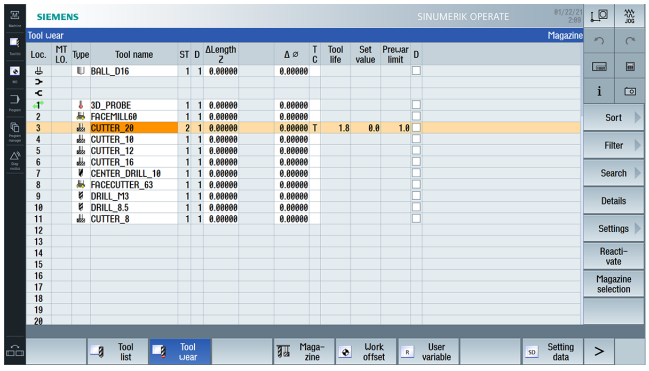
5.2 Monitoring of tool life and workpiece count

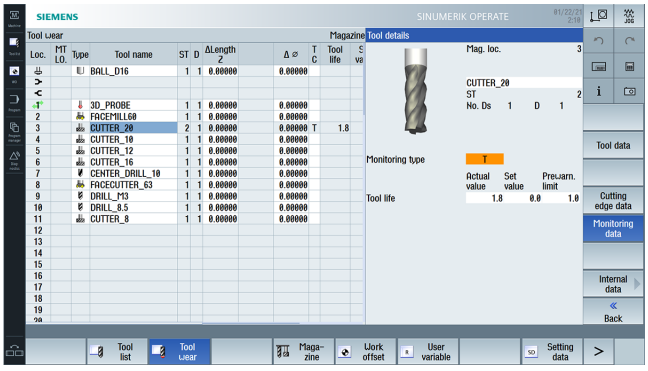
	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	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
- Provided the desired tool is not in the magazine, SINUMERIK Operate will request a manual tool change.





Benefits

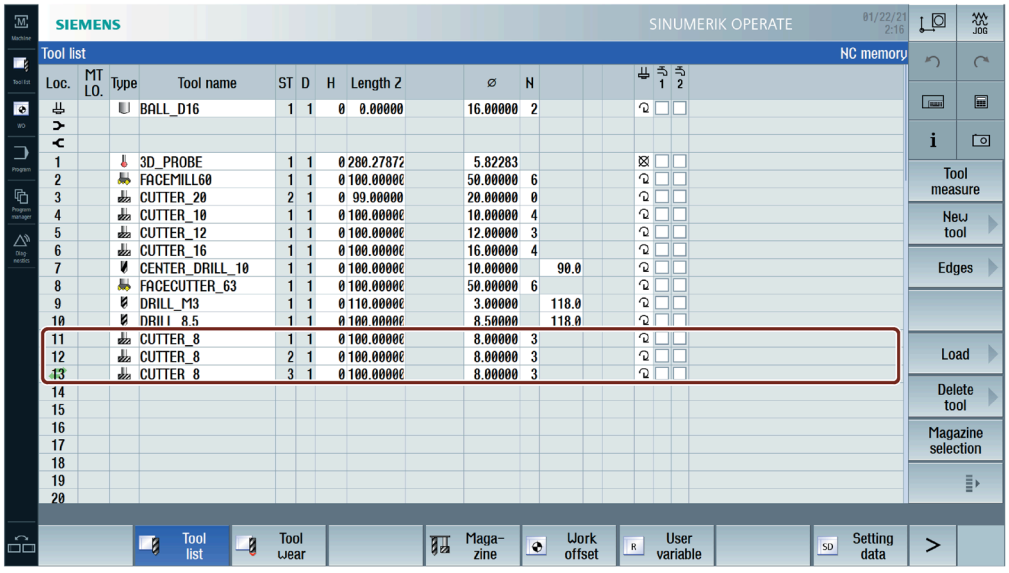
- **Reduction of machine standstill times via tool monitoring**
- **Support of tool life monitoring or job time monitoring as standard**

5.3 Replacement tools

☑	SINUMERIK 828D SW24x	☑	SINUMERIK 828D SW26x	☑	SINUMERIK 828D SW28x
	Option: M78		Option: M78		Option: M78

☑	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. Replacement tools are identified with an increasing number in the ST column.



Benefit

- **Automatic exchange of identical tools for unmanned operation**

5.4 Identifying tool demand

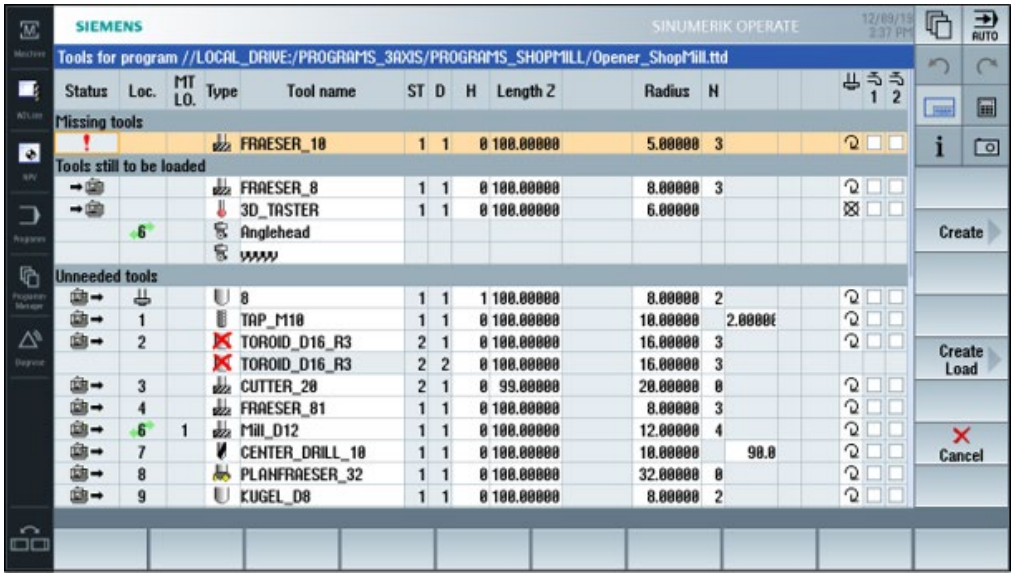
☑	SINUMERIK 828D SW24x	☑	SINUMERIK 828D SW26x	☑	SINUMERIK 828D SW28x
	Option: M77		Option: M77		Option: M77

☑	SINUMERIK 840D sl
	Option: M77

When executing and simulating part programs, all required tools are optionally written as well. If you then use the part program again, SINUMERIK Operate can check whether all of the required tools are available. This creates a list of all tools with the following identifiers:

- Tool unknown
- Tool known but not loaded
- Tool known and loaded
- Tool is not used and can be unloaded.

You can load and unload tools directly in the list. In addition, you can create new tools based on the recorded data.



Benefit

- **Quick and simple check whether all machine tools are loaded before starting the program.**

5.5 Angle head adapter

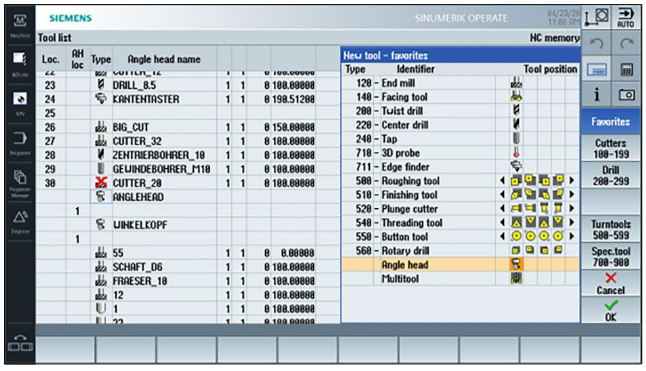
✓	SINUMERIK 828D SW24x	✓	SINUMERIK 828D SW26x	✓	SINUMERIK 828D SW28x
	Option: M56		Option: M56		Option: M56

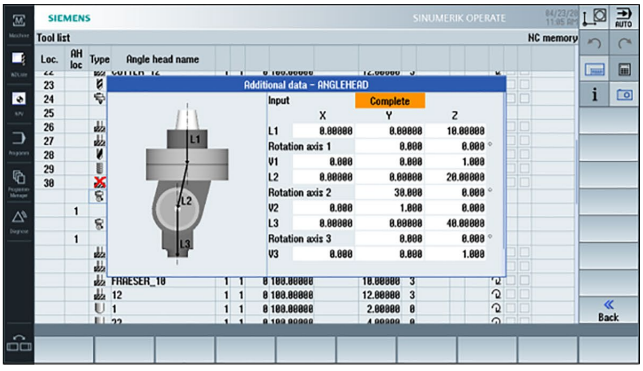
✓	SINUMERIK 840D sl
	Option: M56

With the "Angle head adapter" function, you can describe the angle head and the tool separately and then "marry" them. This also allows any tool type to be used in an angle head.

You can specify the necessary geometric sizes of the angle head in the SINUMERIK Operate user interface and assemble the tool and adapter.

The angle head adapter can be used for all technological functions of SINUMERIK Operate and for the cycles.





Benefit

- **Simple, intuitive input of data and assembly of tool and adapter.**

Data management

6

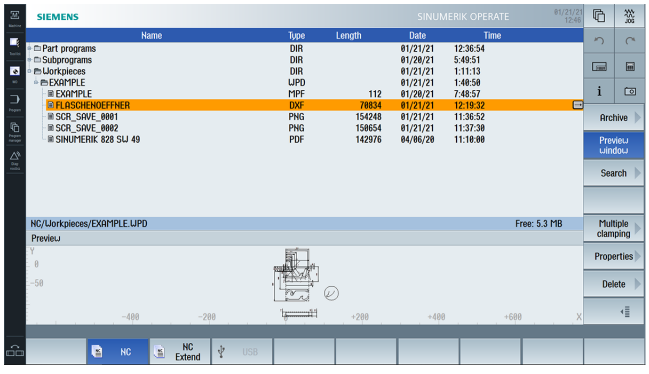
6.1 Program Manager

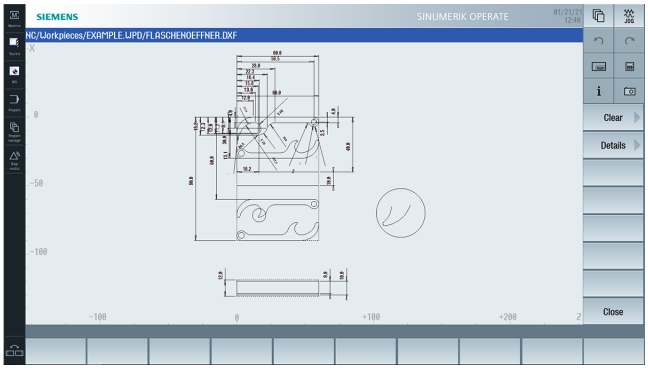
	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

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

- Plain names with as many as 24 characters for directories and files
- Management of 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
- Edit part programs 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 allows quick identification of programs without having to open them**

6.2 Ethernet networking

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

	SINUMERIK 840D sl
	Basic configuration

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

- The data transfer rate is 10/100 Mbps.
- 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

- **Easy and economical connection via Ethernet (TCP/IP) to Windows PCs**
- **No software needs to be installed on the servers**

CNC operation in automatic mode (AUTO) 7

7.1 Block search

✓	SINUMERIK 828D SW24x	✓	SINUMERIK 828D SW26x	✓	SINUMERIK 828D SW28x
	Basic configuration/P16		Basic configuration/P16		Basic configuration/P16

✓	SINUMERIK 840D sl
	Basic configuration

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

The following search variants are available:

- specifically to the point of interruption, also possible after power off
- to any CNC block in DIN/ISO programs
- to any subprogram levels in DIN/ISO programs
- in ShopMill machining step programs
- in position patterns for machining step programming
- accelerated block search in large mold making programs

Note

Extended operating functions for SINUMERIK 828D

For the extended block search (program/block search pointer, levels up/down, interruption point), you need the Extended operating functions option, P16.

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.
- DRF – handwheel offset

  This selection allows you to enter an additional incremental work offset while processing in automatic mode with an electronic handwheel.
- SKP

  Skip blocks are skipped during machining.
- MRD

  The display of the measurement result can be enabled or disabled during program execution.
- CST - Configured stop

  Option: see the following description

Configured stop (CST)

☑	SINUMERIK 828D SW24x	☑	SINUMERIK 828D SW26x	☑	SINUMERIK 828D SW28x
	Option: S24		Option: S24		Option: S24

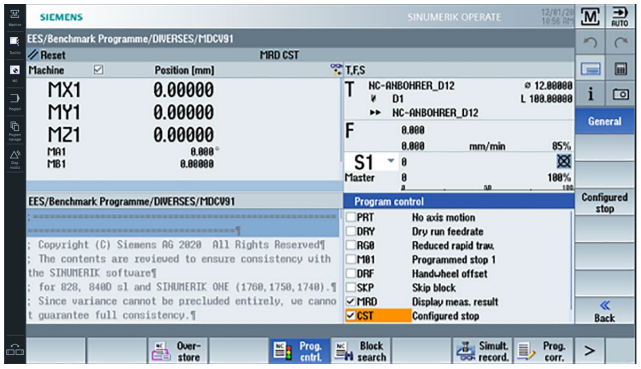
☑	SINUMERIK 840D sl
	Option: S24

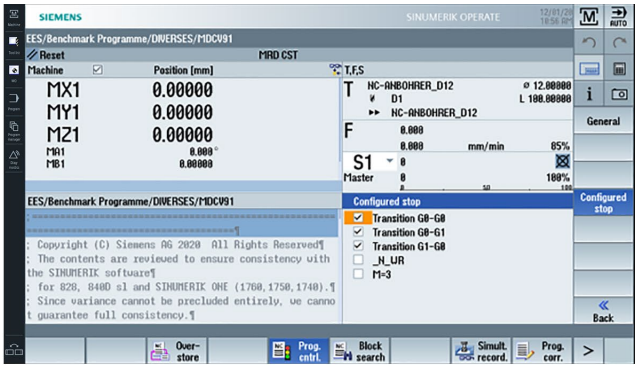
The Configured stop option offers, beyond the basic scope of program control, the following possibilities:

- Additional single block type with stop and NC start only at specific and/or definable "types" of block ends.

  Examples: G0- Non-G0, auxiliary function M.*, subprograms

- The message can be configured language-independently for the type of "Configured stop".





Benefits

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

7.3 Execution from external storage devices

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

☑	SINUMERIK 840D sl
	Basic configuration

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

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

☑	SINUMERIK 840D sl
	Option: P75

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

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

Benefit

- **Quick and easy access to part programs on external storage media**

7.4 Basic block display

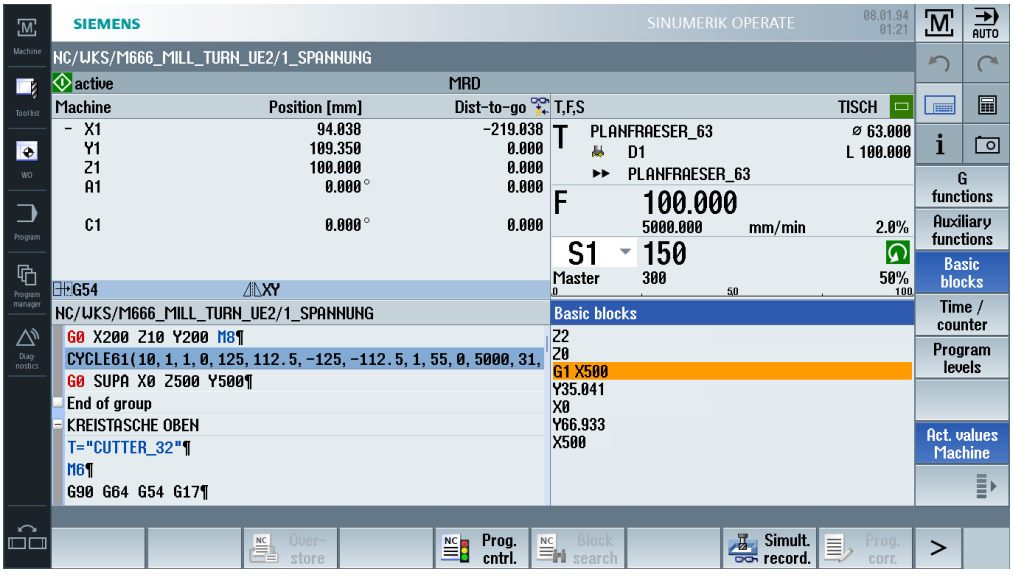
☑	SINUMERIK 828D SW24x	☑	SINUMERIK 828D SW26x	☑	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

☑	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 to you for ShopMill as well as for programGUIDE (figure below).



Benefit

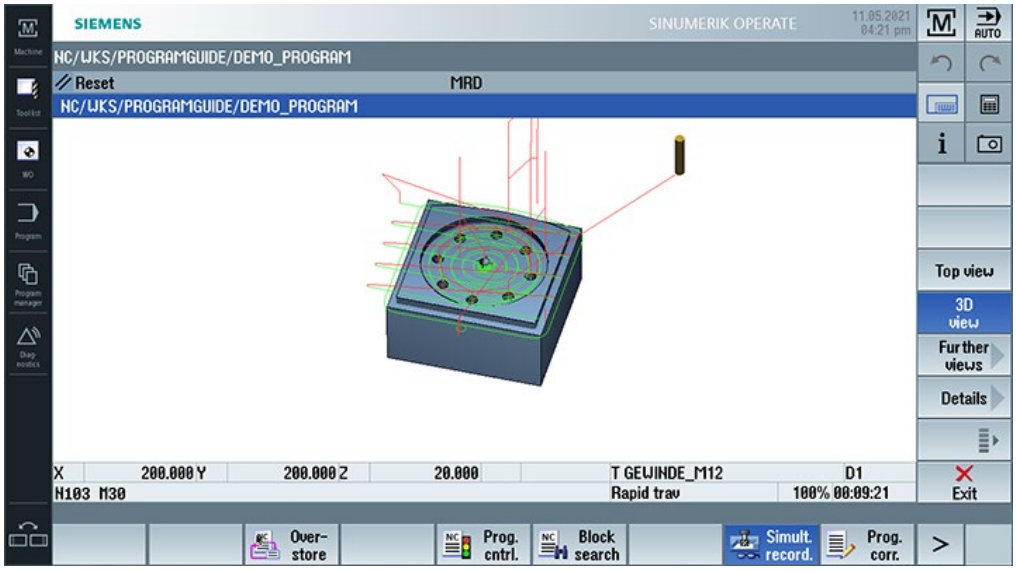
- **Optimal control of the program execution, also in complex sequences or machining cycles, especially in single block mode**

7.5 Simultaneous recording

☑	SINUMERIK 828D SW24x	☑	SINUMERIK 828D SW26x	☑	SINUMERIK 828D SW28x
	Option: P22		Option: P22		Option: P22

☑	SINUMERIK 840D sl
	Option: P22

While machining the workpiece the tool paths can be recorded on the screen of the control in the plan view, 3-side 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

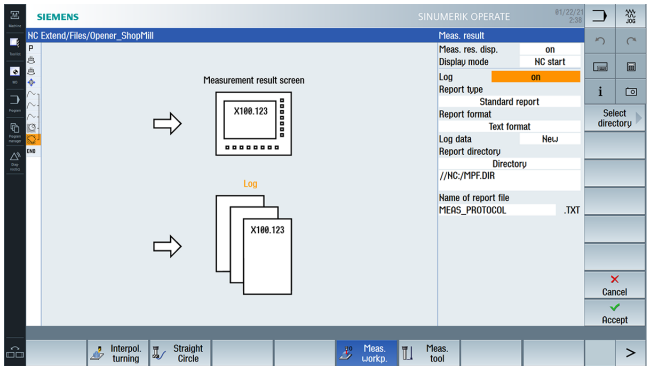
In automatic operation, you can output the measurement results as measuring log. You can configure the output. The following settings are some of those possible:

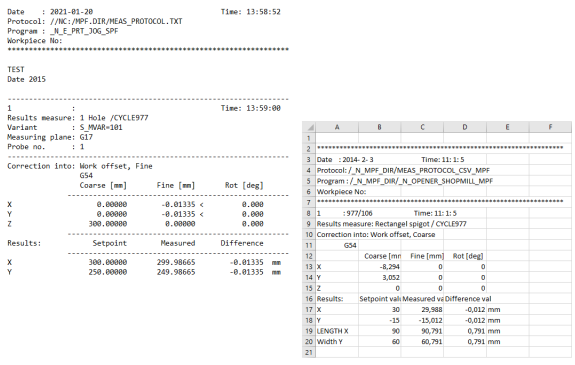
- Display mode: autom. 8 s, NC start, for alarm
- Log type: standard log, user log
- Log format: text format (*.txt), table format (*.csv)
- Log data: new (discard old log data), append (append to old log data)
- Log storage: storage directory (complete path)

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

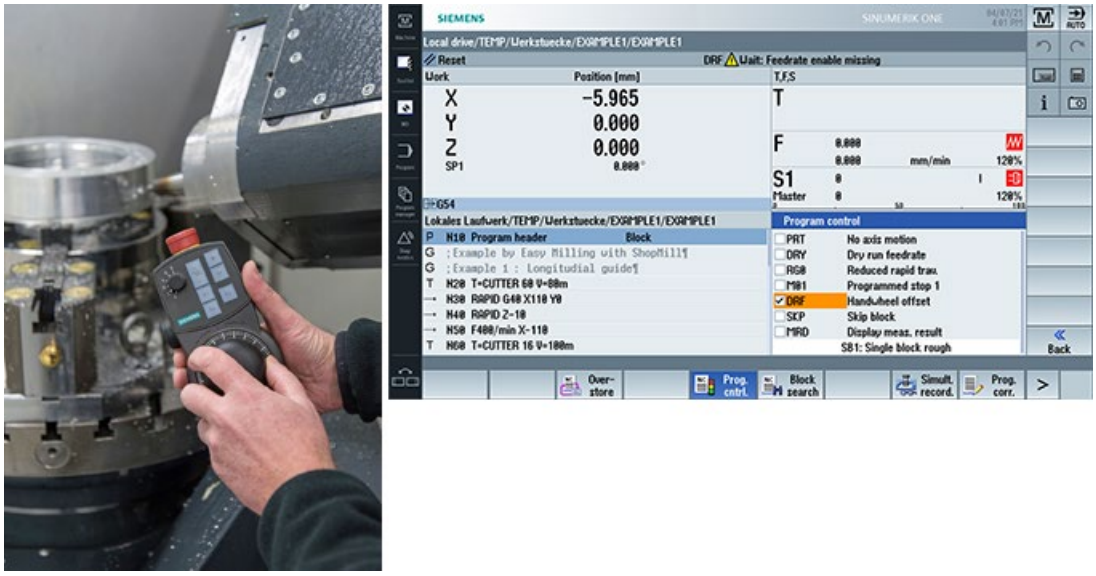
- **Simple logging of measured values in log files**

7.7 Handwheel override

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

	SINUMERIK 840D sl
	Basic configuration

In the AUTOMATIC mode, while executing a program, small corrections and override feed of the tool in the tool direction are possible using a handwheel. When the orientation of the tool changes, the handwheel override that has been accumulated is also rotated. The manual correction acts in the form of override to the traversing motion from the NC program.



Benefit

- **Small corrections and feeds of the tool in the tool direction are possible using a handwheel.**

CNC functions

8

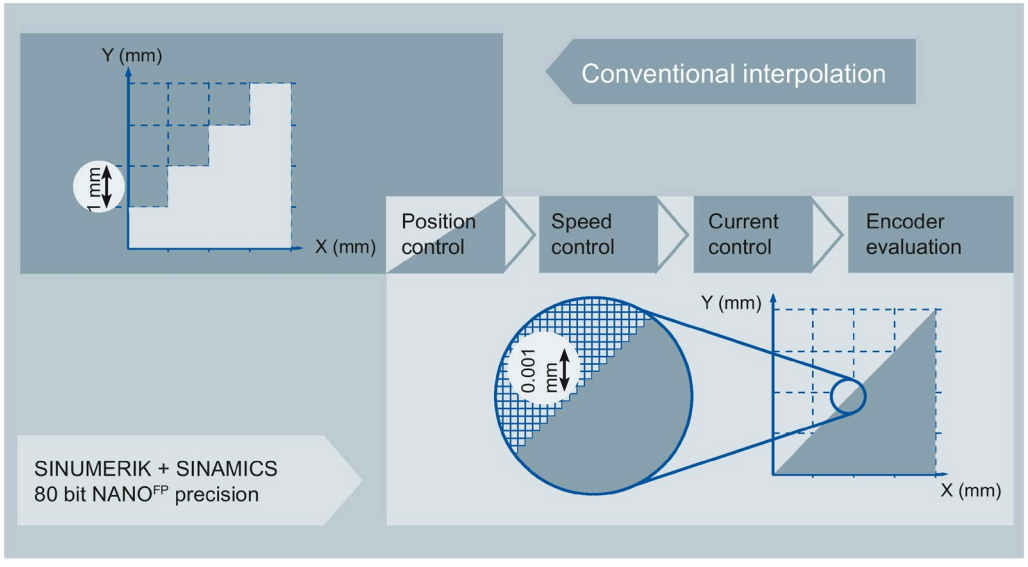
8.1 80-bit NANO floating-point accuracy

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

✓	SINUMERIK 840D sl
	Basic configuration

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

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



Benefit

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

8.2 Block change times

8.2.1 SINUMERIK 828D

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

The following table shows the minimum block change times with compressor depending on the deployed PPU:

PPU 270.4/PPU 271.4/PPU 290.4		
SW24x	**SW26x**	**SW28x**
~3 ms	~2 ms	~1 ms

Benefit

- **Minimum block change times for the associated performance versions**

8.2.2 SINUMERIK 840D sl

☑	SINUMERIK 840D sl
	Basic configuration

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

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

Benefit

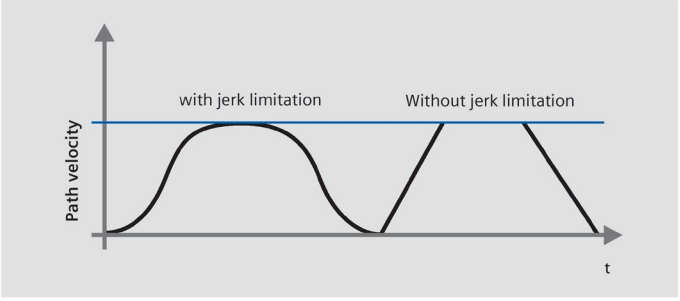
- **Minimum block change times for the associated performance versions**

8.3 Jerk limiting

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

	SINUMERIK 840D sl
	Basic configuration

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



Benefits

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

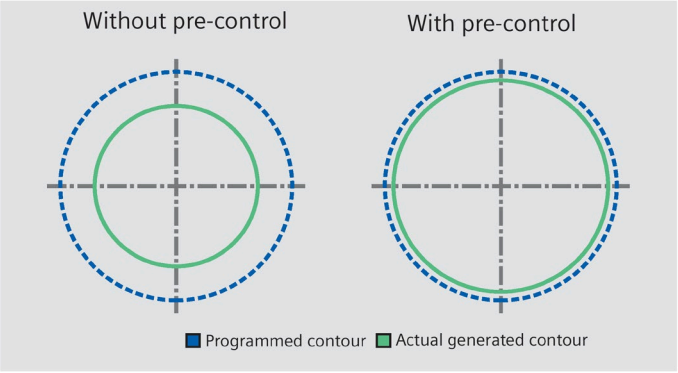
8.4 Dynamic feedforward control

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

☑	SINUMERIK 840D sl
	Basic configuration

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

Example:



Benefit

- **Higher path accuracy through compensation of contouring errors**

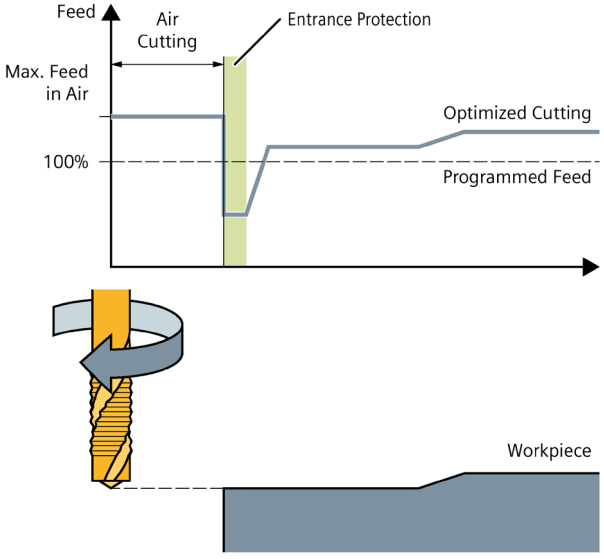
8.5 Adaptive Control & Monitoring (ACM)

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

	SINUMERIK 840D sl
	Option: via SISW

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

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



SINUMERIK 828D - Hardware solution

The solution consists of two components:

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

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

Adaptations to the PLC user program are required for ACM.

SINUMERIK 840D sl - Software solution

The solution consists of two main components:

- Real-time component:

  Compile Cycle Run MyCC /IMD to access the necessary data

- HMI component:

  SINUMERIK Operate, based on Run MyHMI /3GL

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

Note

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

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

Benefit

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

8.6 Intelligent Load Control (ILC)

✓	SINUMERIK 828D SW24x	✓	SINUMERIK 828D SW26x	✓	SINUMERIK 828D SW28x
	Option: S11		Option: S11		Option: S11

✓	SINUMERIK 840D sl
	Option: S11

The "Intelligent load adjustment" function is used to optimize the following characteristics of a machine tool by adapting dynamic response and control parameters:

- Shorter machining times
- Increased dynamic response
- Greater accuracy
- Higher accuracy

The clamping and the weight of the workpiece influence the dynamic response of the machine due to their moment of inertia. During axis movements, inaccuracies in workpiece machining can arise. Using cycle CYCLE782, you can automatically adapt the controller settings of the drive or the dynamic response parameters of the axes to a specific situation. The following axes are supported:

- Rotary table for holding a workpiece
- Linear axes
- Spindles
- Other rotary axis (e.g. A axis for the rocker)

Benefits

- **You achieve faster and more accurate machining on the workpiece.**

Tool and mold making

9

9.1 High speed settings

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

✓	SINUMERIK 840D sl
	Basic configuration

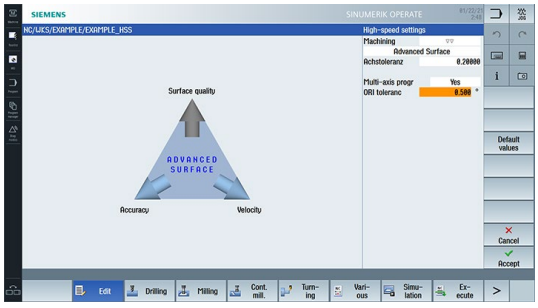
The High Speed settings cycle enables easy parameterization of the optimum motion control in relation to the machining type and the part program contour tolerance band.

The high-speed setting cycle sets automatically the associated optimum combination of accuracy, speed and surface quality – for 3-axis and 5-axis machining of free form surfaces.

The cycle is called within the DIN/ISO editor or in ShopMill. Calling this function activates **Advanced Surface** and/or **Top Surface** depending on the options and the configuration. The best available mold making function is automatically used.

The following settings are possible:

- Machining type
  - Roughing
  - Rough-finishing
  - Finishing
- Tolerance
- Multiple axis program yes/no
- Orientation tolerance and rotary axis tolerance



Benefit

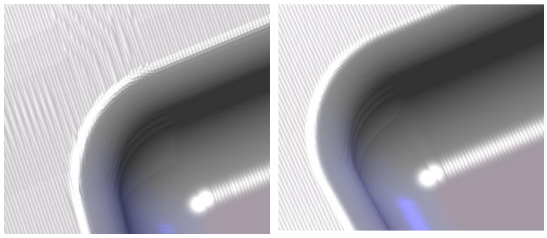
- **Simple and easily understandable parameterization of the required machining type (roughing, pre-finishing or finishing) with an interactive screen**

9.2 Advanced Surface and Top Surface

Machining of free-form surfaces involves high requirements regarding speed, precision and surface quality. The "High Speed Settings" cycle simplifies the parameterization of mold making applications.

The "Advanced Surface" and "Top Surface" options allow the manufacturing of high-quality mold making workpieces.

Conventional CNC **with Advanced Surface and Top Surface**

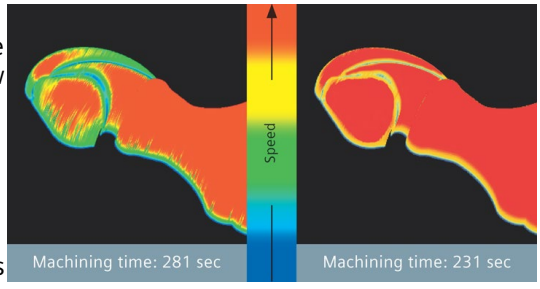


Perfect surface

SINUMERIK Operate can even cope with inadequate CNC block sequences in mold making programs: New forward-thinking, mathematical algorithms perform fully identical calculations for the movement paths in forward and reverse directions. This means that reverse paths on molds yield mirror-finish workpiece surfaces.

Minimum machining time

In addition, Advanced Surface and Top Surface ensure shortest machining times. A brand new type of motion control calculates an ideally smooth surface, for which it keeps the tool within the optimum speed range at all times.



One-off optimizing

The Advanced Surface and Top Surface algorithms guarantee optimum workpiece surfaces and shortest machining times after just a single optimization of the system.

Benefit

- **Advanced Surface and Top Surface are synonyms for milling at physical machine limits; coupled with maximum speed, accuracy and best surface quality, not only for mold making**

9.2.1 Advanced Surface

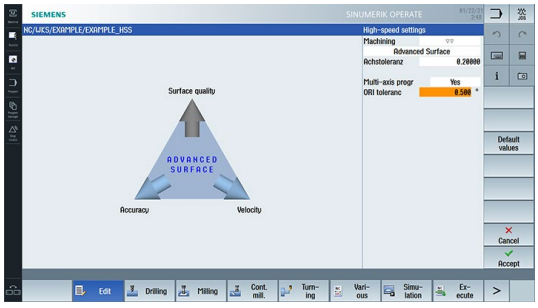
	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Option: S07

With Advanced Surface you can easily parameterize optimum speed control depending on the machining type (roughing, rough-finishing, finishing).

Enter the following settings for Advanced Surface:

- Tolerance of the machining axes
- Machining type
  - Finishing
  - Rough-finishing
  - Roughing
  - Deselection
- Multiple axis program yes/no



Benefit

- **Advanced Surface permits maximum productivity coupled with simple process parameterization – from 3-axis multipass milling through to dynamic 5-axis machining**

9.2.2 Top Surface

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: S17		Option: S17		Option: S17

	SINUMERIK 840D sl
	Option: S17

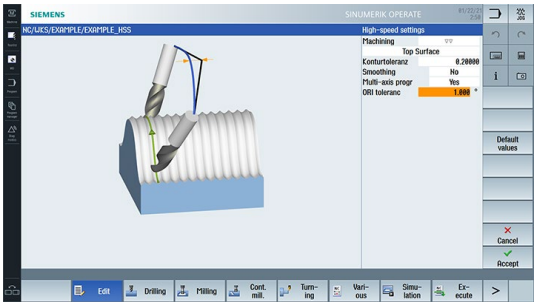
The High Speed Settings cycle, Top Surface option ensures a significantly improved workpiece surface for inclined multipass finishing programs, even for poor data quality and/or irregular point distribution in NC programs from the CAD/CAM system.

The dynamic response is also optimized:

- Improved observance of the acceleration and jerk limits
- Lower vibration excitation of the machine

In addition to selecting the machining types (finishing, rough-finishing, roughing), the following settings are possible:

- Smoothing yes/no
- Multiple axis program yes/no
- Contour and orientation tolerance

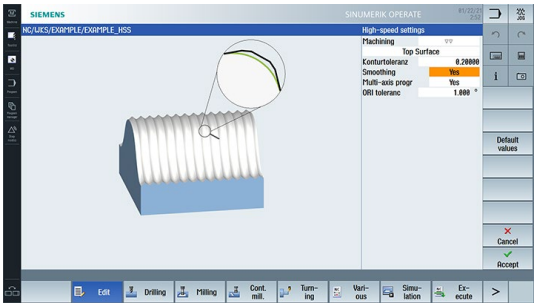


The contour tolerance is shown as magnifying glass.

Standard values:

- Roughing 0.1
- Rough-finishing 0.05
- Finishing 0.01

The smoothing is also shown in the magnifying glass:



- Smoothing adds shine to the surface.
- Without smoothing, high-precision contours appear perfectly.

Benefits

- **Perfect surface quality - Correction of irregularities from the CAD/CAM data, direction-independent identical smoothing of the milling paths**
- **High accuracy**
- **Stable milling machine - significantly smoother machine running, less wear, long-term availability**
- **Perfect usability - simple and graphical operator screens, optimum surface quality, even with the default setting, for most programs**

9.2.3 Top Speed

☑	**SINUMERIK 840D sl**
	Option: S13

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

Top Speed permits the increase of the axial jerk value during 3-axis and 5-axis simultaneous machining while maintaining high contour precision. An increase in traversing velocity and thus a reduction in machining time are the direct result. A combination of Top Speed and Top Surface for maximum performance during 3-axis and 5-axis simultaneous machining is obligatory.

Benefits

- **Significant increase in machining speed possible, for example in tool and mold making, by increasing the jerk values - Top Speed**

9.2.4 Top Speed Plus

☐	**SINUMERIK 840D sl**
	Not available

☑	**SINUMERIK 828D SW24x**	☑	**SINUMERIK 828D SW26x**	☑	**SINUMERIK 828D SW28x**
	Option: S62		Option: S62		Option: S62

Top Speed Plus and Top Surface are used together in the processing of CAM-generated 3/5-axis simultaneous machining operations, for example in tool and mold making. The innovative filter technology ensures that the highest motion dynamics of the individual machine axes can be set, while at the same time providing improved surface quality and high contour accuracy.

Benefits

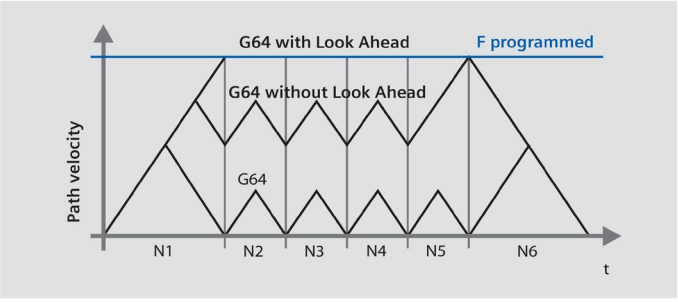
- **When using Top Speed Plus, the restrictions that all axes must be operated with the same filter and jerk limitation values no longer apply**

9.3 Look Ahead

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

☑	SINUMERIK 840D sl
	Basic configuration

The Look Ahead function (the function is part of Advanced Surface) achieves an optimum machining speed by looking ahead over a parameterized number of traversing blocks. With tangential block transitions, the axis is accelerated and decelerated beyond block boundaries, so that no drops in speed occur.



Benefit

- **Shorter machining times through optimum speed control**

CNC programming methods

10

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.

ShopMill machining step programming

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

Machining operations such as traversing movements, drilling or pocket milling are shown in ShopMill in the form of machining steps. This means that CNC programs are very compact and are easy to generate and read – even for complex machining operations. Associated sequences are automatically interlinked and can be assigned any position patterns.

Benefit

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

10.1 programGUIDE DIN/ISO and SINUMERIK high-level language

10.1.1 Introduction

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

☑	**SINUMERIK 840D sl**
	Basic configuration

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

- DIN/ISO editor
- Languages
- programGUIDE input support

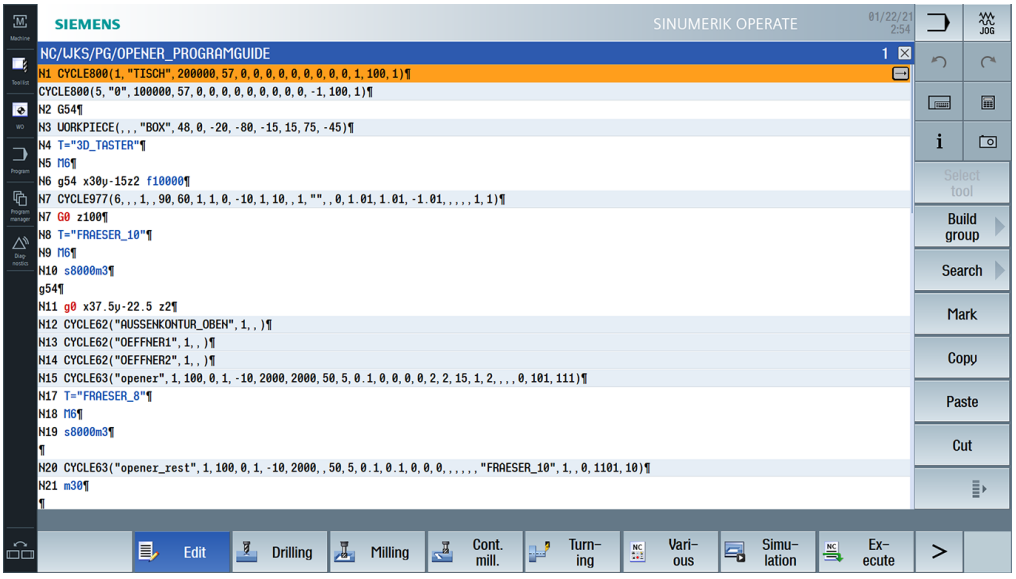
These functions belong to the basic configuration of SINUMERIK Operate.

10.1.2 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 is 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", "Paste" and "Cut" block
- "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 (many MB large) can be edited extremely fast**

10.1.3 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 ...

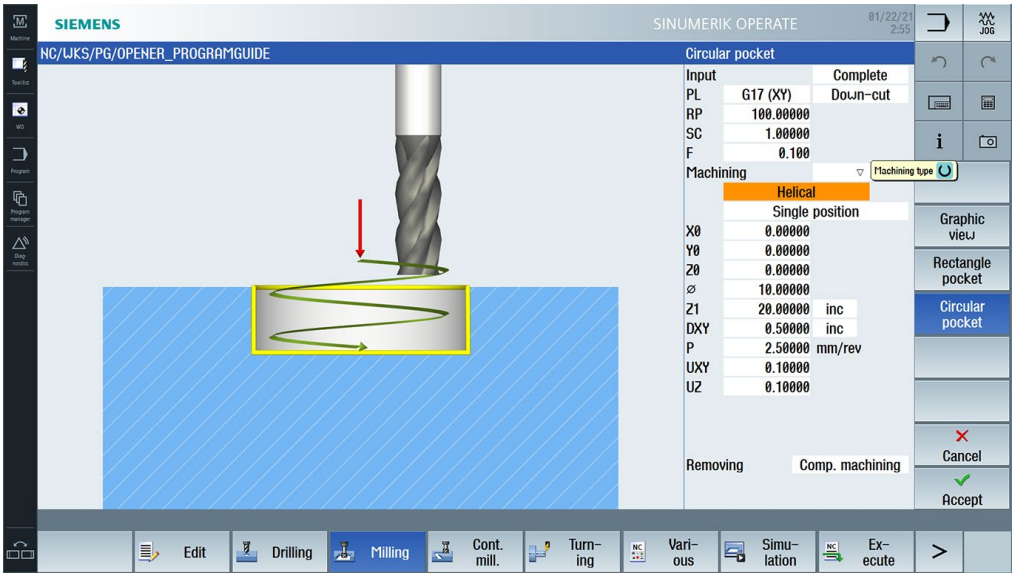
Benefits

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

10.1.4 programGUIDE input support

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

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 consistency of the input support**

10.2 ShopMill - machining step programming

10.2.1 Introduction

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

☑	**SINUMERIK 840D sl**
	Option: P17

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

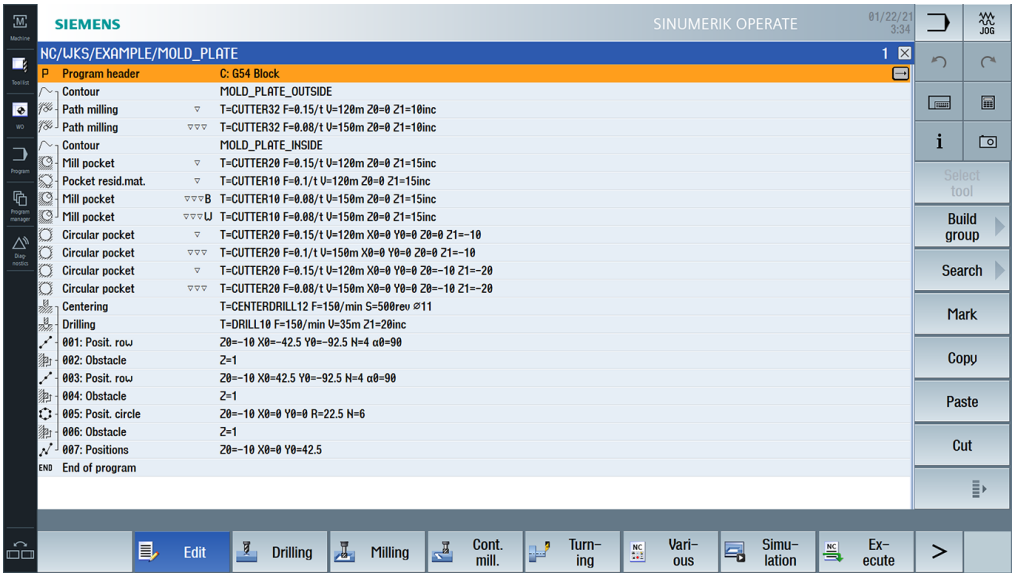
- Sequence editor
- Interlinking of sequences
- Broken-line graphics

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

10.2.2 Sequence editor

The graphical programming is performed via a graphic interactive sequence editor. Each program line represents a technological sequence (such as: face milling, 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 technological 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.



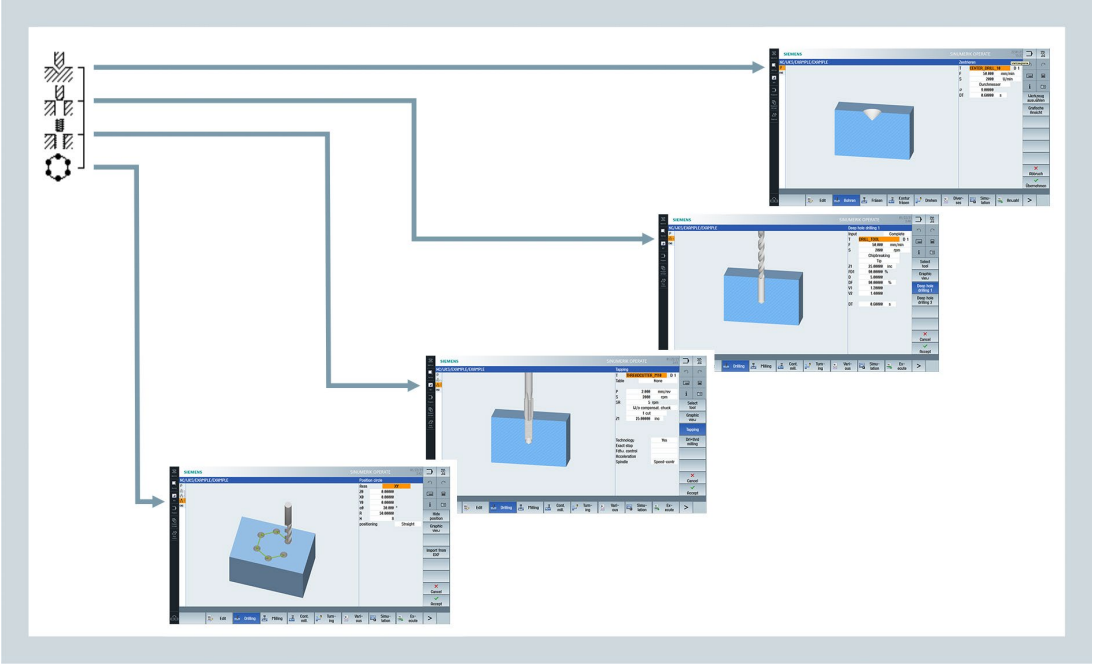
Benefits

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

10.2.3 Interlinking of sequences

In ShopMill, 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, deep-hole drilling and tapping are applied to 6 holes on the pitch circle pattern position.



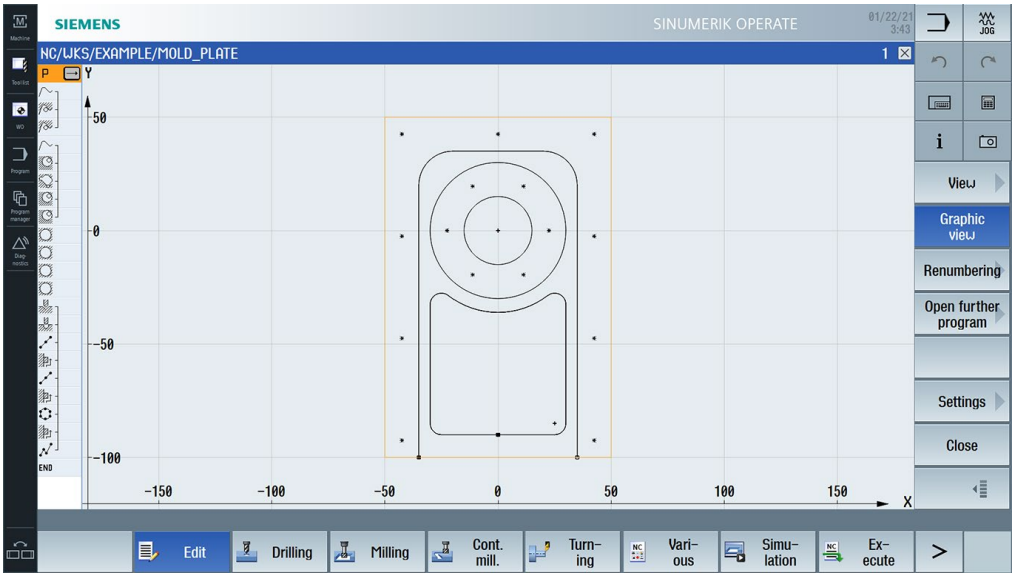
Benefit

- **Reduced programming time due to linking of machining steps**

10.2.4 Graphic 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.

- Plan view of workpiece
- Front view of drilling operations



Benefit

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

Workpiece visualization

11

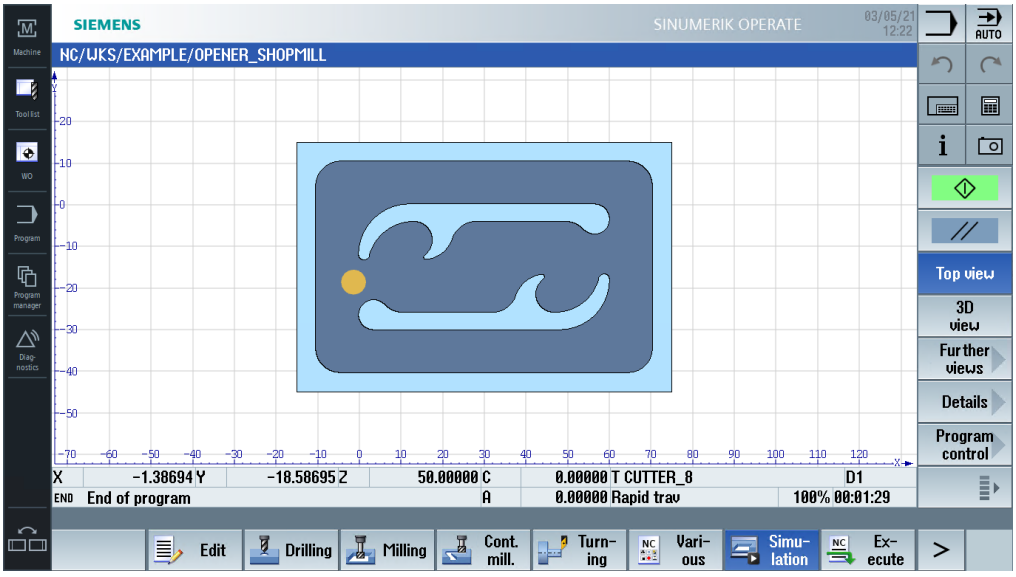
11.1 2D simulation

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

	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 plan view and side 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.**

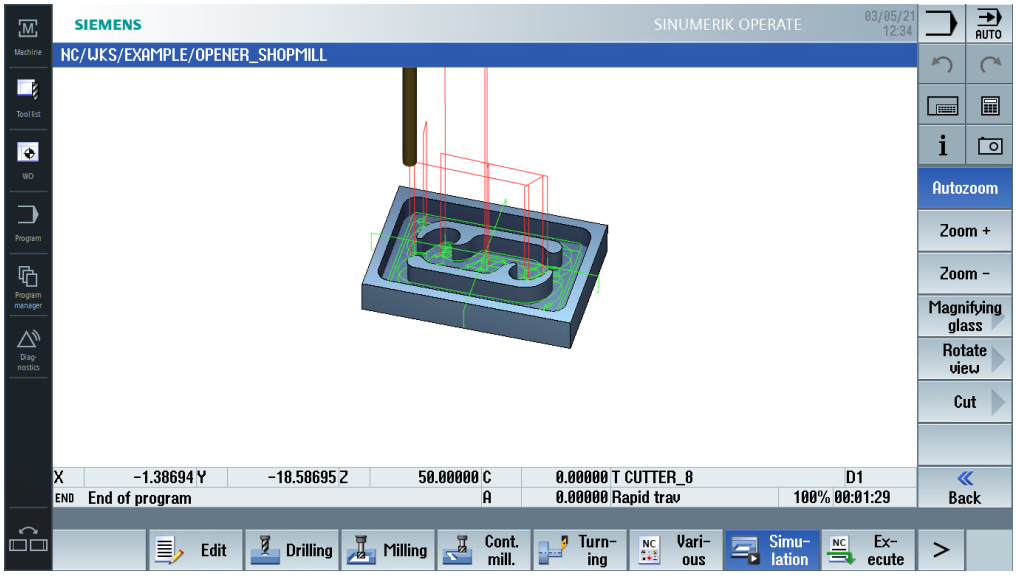
11.2 3D simulation

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: P25		Option: P25		Option: P25

	SINUMERIK 840D sl
	Option: P25

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

- Reliability:
  3 viewing planes and solid model of the finished part, with zoom to details and free rotation of the viewing angle
- Support:
  - Simulation speed controllable by override
  - 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.**

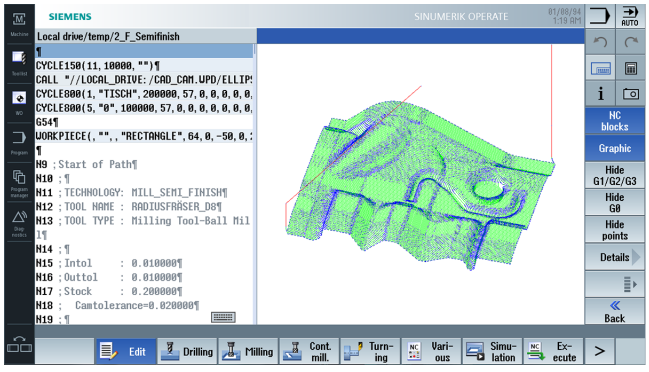
11.3 Mold making fast view

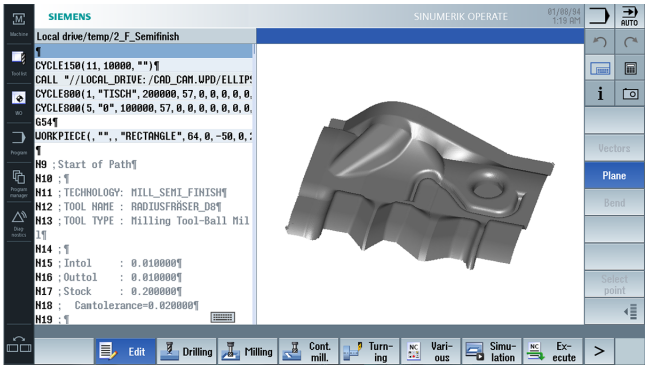
✓	SINUMERIK 828D SW24x	✓	SINUMERIK 828D SW26x	✓	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

✓	SINUMERIK 840D sl
	Basic configuration

The mold making fast view is available, in particular for large part programs.

- Fast view of G0, G1, G2, G3 blocks, VECTORS using the 3D mold building model
- Fast identification of part programs where simulation would take a long time
- Displaying/hiding G0, G1, G2, G3 lines and points
- In addition to the classic view, for mold making programs, you can also display the rotary axis vectors and grid mesh (surface, mesh), for example.





Benefit

- **More safety when handling mold making programs**

CNC technology cycles

12

12.1 CNC technology cycles for programGUIDE and ShopMill

Irrespective of whether you use programGUIDE or ShopMill – 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**
- **Consistency of cycles for programGUIDE and ShopMill**

12.2 Highlights of machining cycles

12.2.1 Overview

For frequently repeated machining tasks, machining cycles are available for the drilling, milling and turning technologies.

- Drilling technology:

  Drilling/centering, drilling/counterboring, deep-hole drilling, tapping with and without compensating chuck, boring 1 ... 5, row of holes, circle of holes, grid of holes, machining on inclined surfaces

- Milling technology:

  Thread milling, elongated holes in a circle, grooves in a circle, circumferential groove, rectangular/circular pocket, face milling, path milling, rectangular/circular spigot, machining on inclined surfaces, high-speed settings for optimized HSC machining, engraving cycle

- Turning technology:

  Groove, undercut, cutting with relief cut, thread undercut, thread cutting, chaining of threads, thread recutting

A selection of machining cycles is explained in more detail below.

12.2.2 Engraving cycle

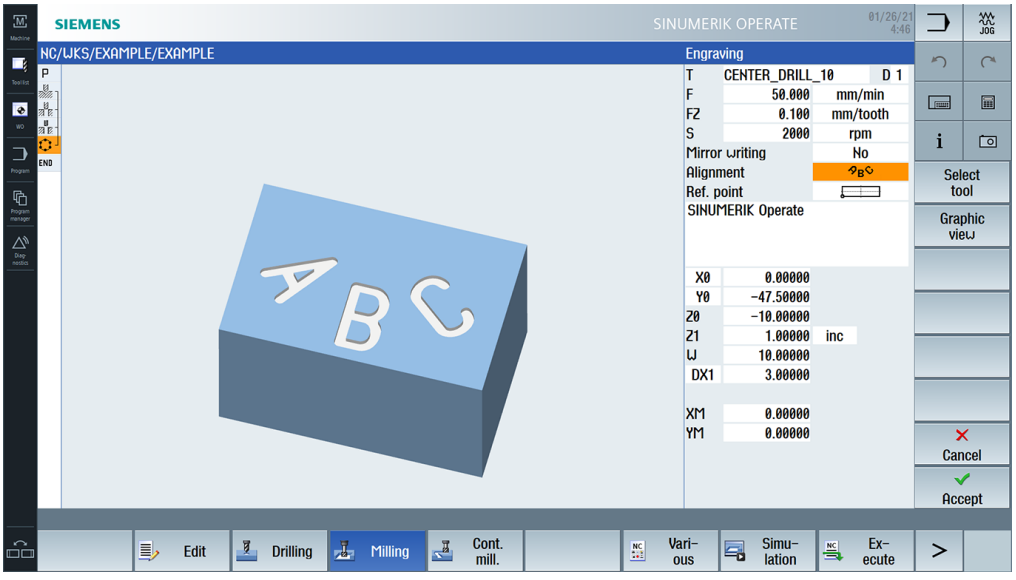
☑	SINUMERIK 828D SW24x	☑	SINUMERIK 828D SW26x	☑	SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

☑	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**

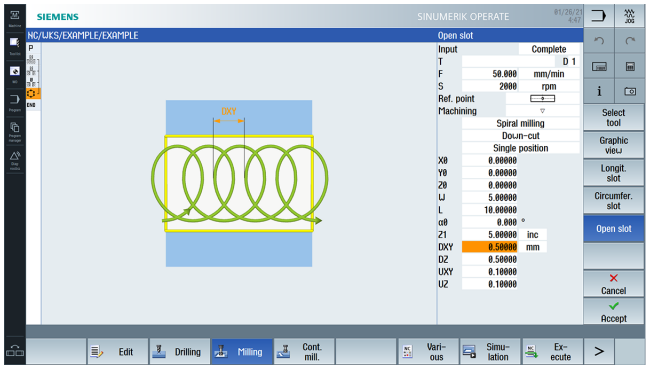
12.2.3 Trochoidal milling

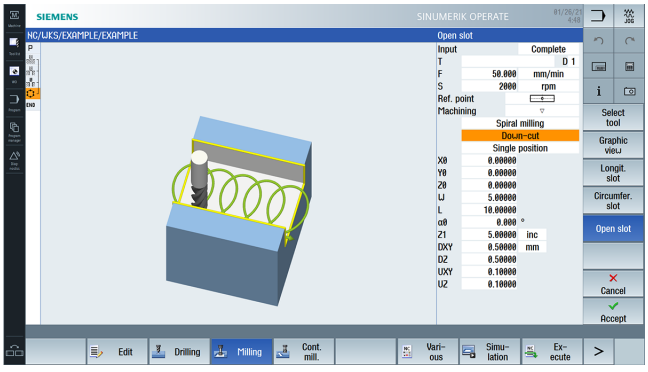
	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

Vortex milling (trochoidal milling) of open slots is available as a milling strategy directly on the controller, i.e. NC programs for path motions do not have to be generated by CAM systems as previously.

- It is the preferred strategy for HSC roughing, the tool is never fully inserted and tool paths are smooth and round
- Simple parameterizing per dialog: Roughing, pre-finishing, finish milling, finishing floor and edge
- You can select as milling direction synchronous operation, reverse rotation, and for maximum cutting volume during roughing the combination reverse rotation and synchronous operation





Benefits

- **Innovative CAM function now available directly on the controller**
- **Reduction in the machining time for slot milling by up to 50%**

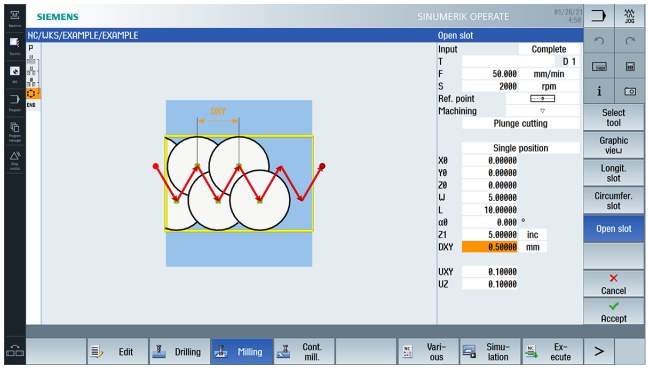
12.2.4 Plunge milling

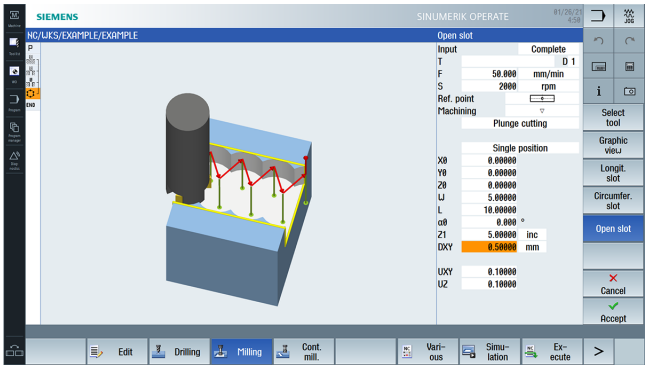
	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Basic configuration		Basic configuration		Basic configuration

	SINUMERIK 840D sl
	Basic configuration

For machining deep pockets and slots in thin-walled workpieces, the plunge milling cycle is available for open slots.

- As types of machining you can select roughing, pre-finishing and finishing of the edge and/or floor
- Essentially, forces apply only along the main spindle axis, therefore, hardly any distortion of the tool occurs.





Benefits

- **Less vibrations and deeper cutting depth thanks to the new machining strategy plunge milling**
- **Reduced cutting pressure and distortion enable higher productivity when machining thin-walled workpieces**

12.2.5 Deep-hole drilling

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

☑	SINUMERIK 840D sl
	Basic configuration

Easy-to-use cycles for deep-hole drilling are available in SINUMERIK Operate.

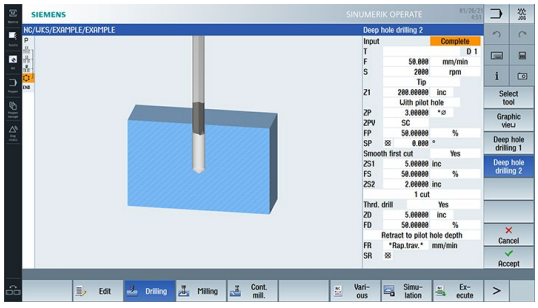
The tool drills at the programmed spindle speed and feedrate to the entered final drilling depth.

Deep hole drilling is performed with a depth infeed of a maximum definable depth executed several times, increasing gradually until the final drilling depth is reached.

For example, the drilling machine can be retracted after each infeed depth either to the piloting depth + safety clearance for chip removal or by the length of the programmed retraction path for chip breakage.

You can also choose between the following drilling strategies:

- None / with spot drilling
- With or without pilot hole
- Soft first cut yes/no
- Chip breaking/removal
- Chip breaking and swarf removal
- 1 cut - drill in one step to the end depth
- Swarf removal to the piloting depth / safety clearance
- Retraction to the piloting depth / retraction plane
- Position pattern



Benefit

- **Generate drill holes with more than one feed to any positions**

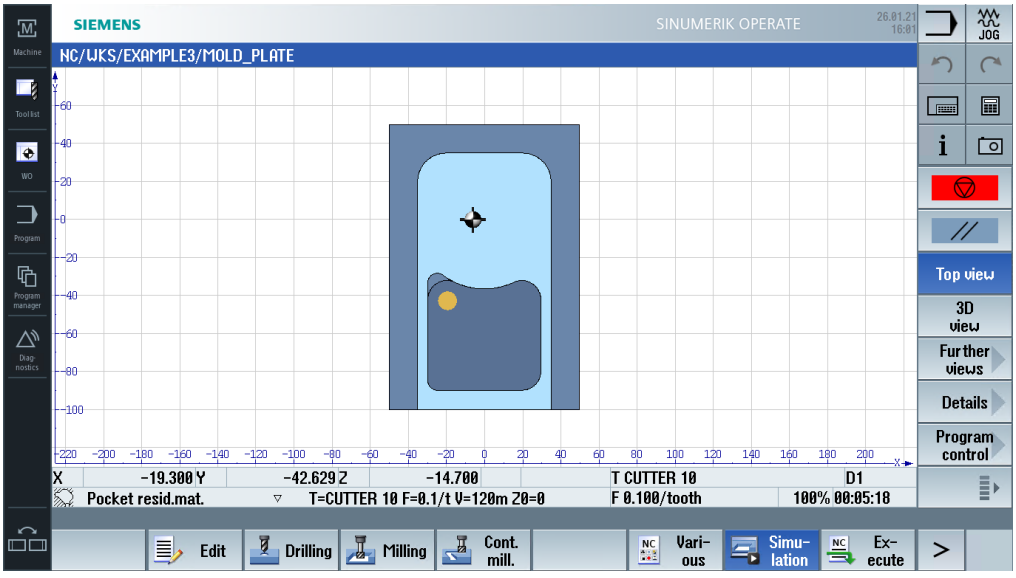
12.3 Residual material detection for contour cycles

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: P13		Option: P13		Option: P13

	SINUMERIK 840D sl
	Option: P13

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

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

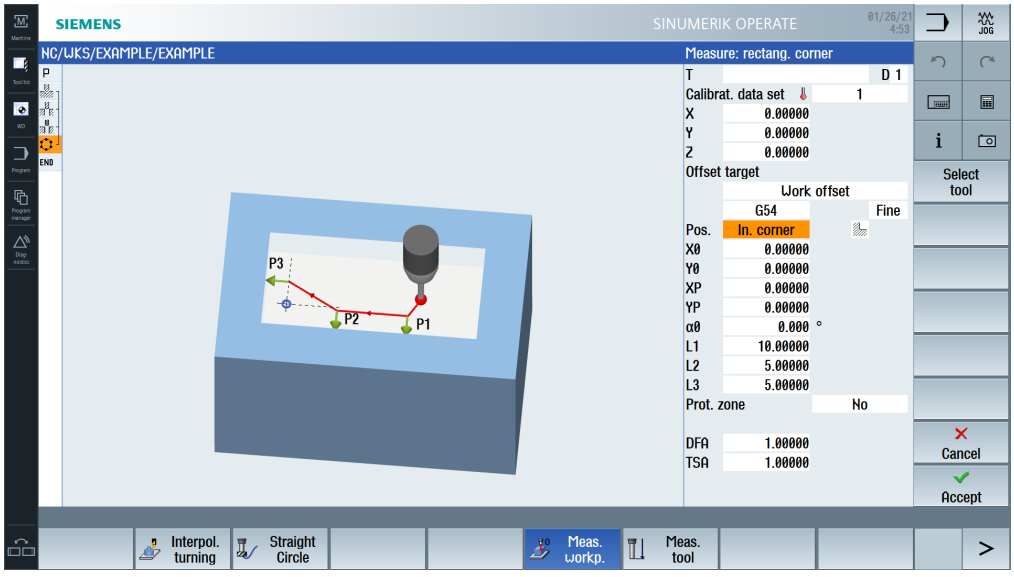
- **Shorter machining times through the use of a large tool for the substantial part of the stock removal and a smaller tool for the remaining residual material**
- **Avoidance of non-cutting movements while achieving extremely simple programming**

12.4 In-process measuring for workpiece and tool

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

	SINUMERIK 840D sl
	Option: P28

For measuring tasks in the automatic mode, powerful measuring cycles are available both under ShopMill as well as under programGUIDE. Input screens with dynamic help displays are used for convenient entry of the measuring parameters.



You can perform the following measuring tasks:

- Workpiece measurement: Correction of work offsets, correction of tool geometry or only measuring
- Tool measurement: Correction of tool geometries
- Display of measurement results
- Logging of measurement results

The following workpiece measuring versions are available:

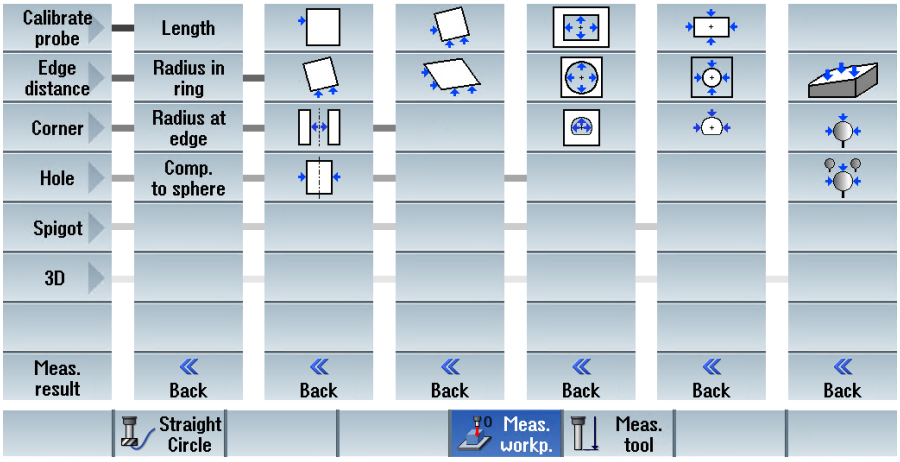


Figure 12-1 In-process measurement

- Calibrating - length, radius in ring, radius at edge, calibration on a sphere, calibration in a slot
- Measuring edge - point/ surface, align edge, distance groove/web,
- Measuring corner - right-angled corner with 3 points or any corner with 4 points, internal/external
- Measuring holes - over 4 or 3 points on a segment of a circle - rectangular pocket
- Measuring spigots - over 4 or 3 points on a segment of a circle - rectangular pocket
- 3D measuring - align plane - sphere

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**
- **Measuring cycles are now also available for ShopMill sequence programs**

12.5 Measure multiple axis kinematics

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: P18		Option: P18		Option: P18

	SINUMERIK 840D sl
	Option: P18

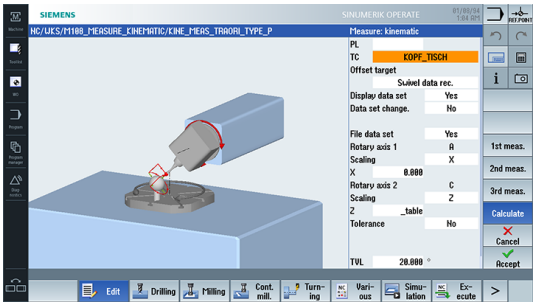
CYCLE9960 corrects or checks the geometric vectors for defining the kinematic 5-axis transformation. With only one call of the cycle, the kinematics are measured, and the determined values and deviations are represented in the measurement result screen.

For the measurement, up to twelve positions of a measuring sphere on each rotary axis are sensed using workpiece probes. The ball positions are defined in a specified rotary axis area in accordance with the geometric ratios on the machine. The ball position is set via the automatic repositioning of each rotary axis to be measured.

With CYCLE9960, it is also possible to measure the deviation on the tool tip (TCP) with active transformation for various rotary axis positions, after the measurement and compensation of the kinematics. These deviations can be compensated with the VCS (Volumetric Compensation System).

Possible fields of application:

- Check and correct the machine kinematics, e.g. during the machining process or for collision.
- Measure and adapt the reference, e.g. for kinematics with changeable heads.
- Measure and correct interpolation points, e.g. for compensation of the TCP via VCS Rotary.



Benefits

- **Complete measurement with one cycle call possible**
- **Improved machine accuracy through automatic probe calibration**

Complete machining

13

13.1 Cylinder surface transformation (TRACYL)

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

☑	SINUMERIK 840D sl
	Option: M27

Peripheral surface machining can be executed on machines with an additional part apparatus. It is typically handled with an A axis.
Peripheral surface machining offers a series of additional functions in comparison to simple positioning along the A axis.

Programming in the run-off

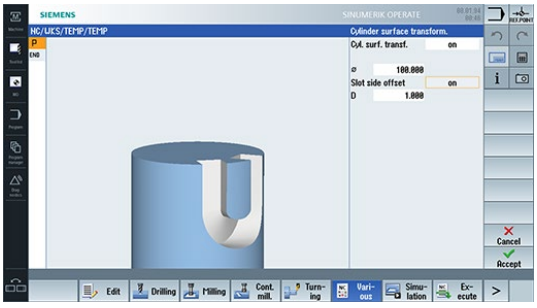
The axis behaves like a Y axis while programming in the run-off. All plane machining can also be executed in the run-off.

- Drilling operations at any position patterns
- Milling (pockets, contour pockets)

The Y values are converted while machining along the A axis rotation. The Y axis of the machine does not move.

Milling grooves with parallel walls

Peripheral surface machining offers the possibility of milling grooves on parallel walls with and without groove side offset. This is also possible when the diameter of the milling cutter is smaller than the groove width. In this case, the cutter radius compensation may be used. The required Y axis compensating movements are automatically calculated by the controller.



Benefits

- **Additional business through expansion of workpiece spectrum**
- **Reduction of set-up times by complete machining on one machine**

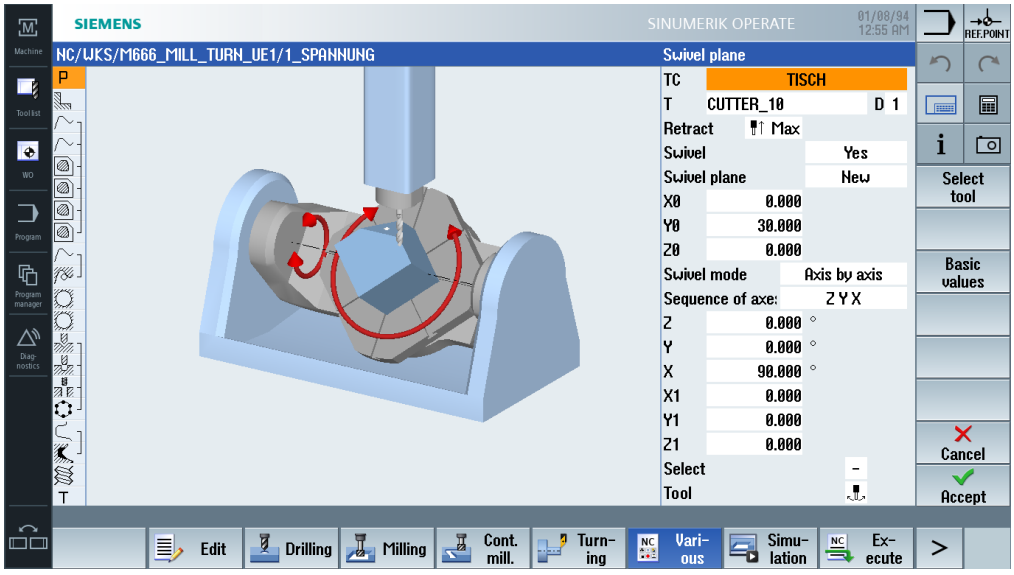
13.2 Swivel plane (CYCLE800)

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

☑	SINUMERIK 840D sl
	Basic configuration

Multi-face machining saves setup times and increases the precision of finished adjoining sides because the part must not be reclamped. The swivel cycle is used for easy input of parameters for automatic machining and measuring on the various planes.

- A prerequisite is that the machine is equipped with additional rotary axes (swivel head and/or swivel table).
- The swivel cycle is available in the ShopMill machining step - as well as in programGUIDE DIN/ISO programming.
- The planes can be swiveled not only by direct swiveling with rotation of coordinates and swiveling about the axes, but also by specifying a projection or spatial angle for swiveling.
- Flexible combination of shift - swivel - shift.
- Turning or moving are not machine-specific, as they are based on the workpiece coordinate system X, Y and Z.
- Fixed relief positions available



Benefit

- **Programming with standard cycles and easy transformation on the inclined plane through the swivel cycle**

13.3 5-axis machining package (TRAORI)

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

	SINUMERIK 840D sl
	Option: S33 (Package)

In contrast to static transformations (swiveling) in which the tool is perpendicular to the machining plane, the 5-axis machining package TRAORI allows the dynamic coupled motion of a tool along the workpiece surface. It is used for 5-axis mold making applications and in the aviation industry, for example.

- Any tool orientation
- Remote Tool Center Point function (RTCP)
- Part programs not dependant on kinematics (vector programming)

Benefits

- **Programming the tooltip in workpiece coordinates**
- **Programmed speed with reference to the tooltip**
- **Programming the tool orientation independent of the machine kinematics**

13.4 Milling-turning

13.4.1 Introduction

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

	SINUMERIK 840D sl
	Basic configuration

The milling technology in ShopMill and programGUIDE provides comprehensive technology cycles for turning and contour turning.

Among others, the following functions are available for milling-turning:

- TSM mode
- Tool measurement
- Face milling / stock removal
- Turning cycles for stock removal, grooving, undercutting, threading and tapping
- Contour turning cycles for stock removal / residual stock removal, plunge cutting / residual plunge cutting, plunge turning / residual plunge turning
- Swivel tool

You can check the programming result, even for milling-turning, with the Simulation function.

Benefits

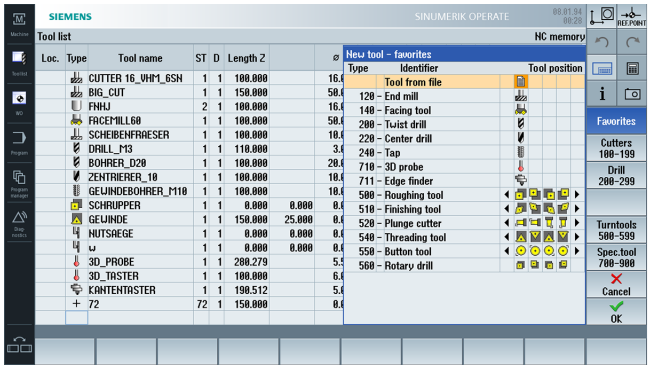
- **Consistent look-and-feel for turning and milling permits a high degree of consistency in the operation and programming for milling-turning**
- **Consistent technology cycles for milling, turning and measuring tasks**
- **Powerful tool management for milling-turning, including multitools**
- **Simulation permits a high degree of process reliability**

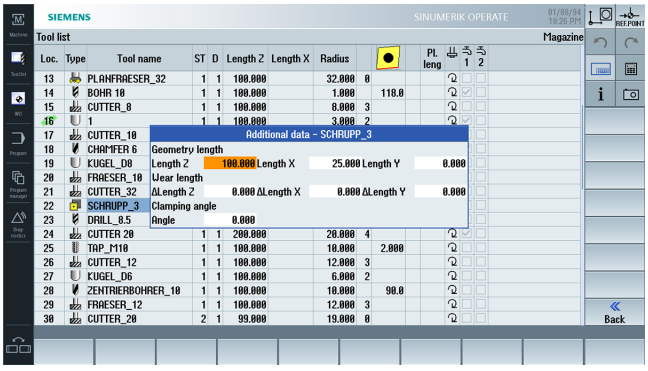
13.4.2 Tool management

For multitasking machines – for milling-turning or turning-milling – you are provided with an extended tool management for turning and milling tools.

The turning tools are displayed automatically in the milling-turning technology. In the "Extended data" dialog, you can enter the tool-specific basic rotation for the turning tools.

In addition to turning and milling tools, you can also deploy complex tools, such as multitools. There are additional parameters for multitools, e.g. distance definition using the location number or angle – and different tool types for each location. All tools are shown as icons



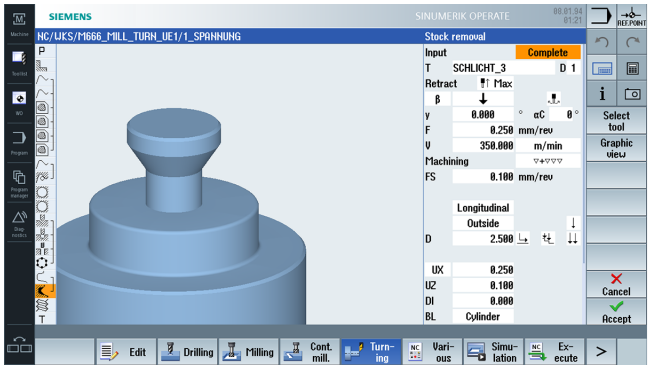


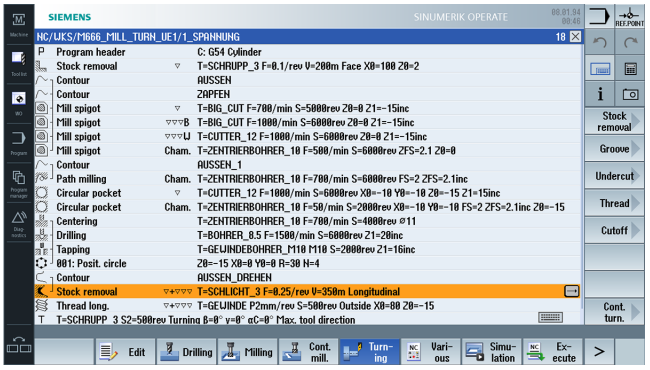
Benefits

- **One tool management for turning and milling tools – including support for multitools**
- **All tools are displayed as symbols**
- **Tool name in plain text**

13.4.3 Programming

For milling-turning machining, programGUIDE and ShopMill provide not only standard cycles, but also turning cycles and contour cycles. You are supported with the appropriate cycles for turning machining as well as turning contour machining and aligning the turning tool.



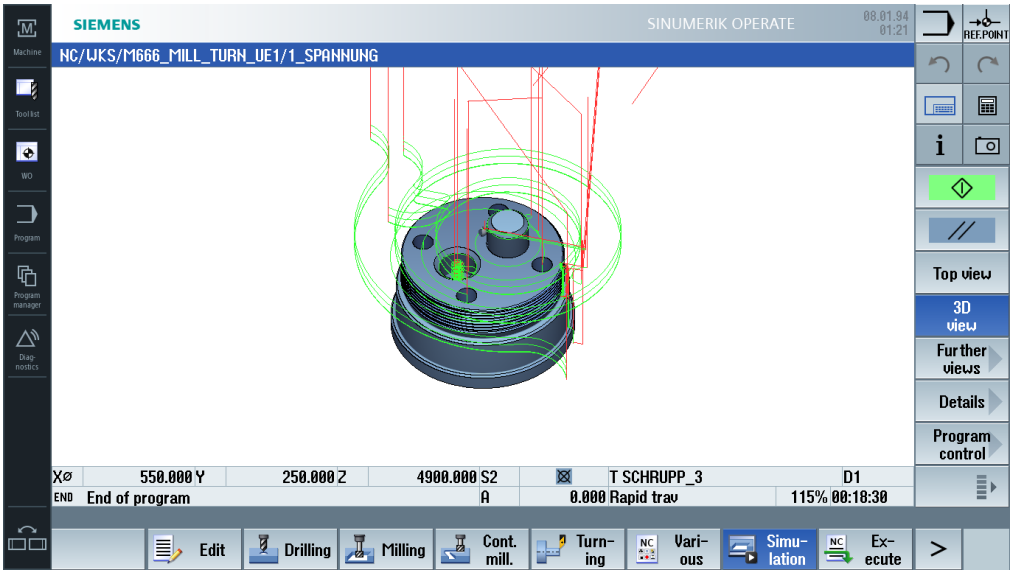


Benefit

- **Turning cycles for programGUIDE and ShopMill as for SINUMERIK Operate turning technology**

13.4.4 Simulation

Also for milling-turning, the usual views are available to simulate the workpiece.



Benefit

- **Maximum process reliability through simulation using real geometry values**

13.5 Interpolation turning

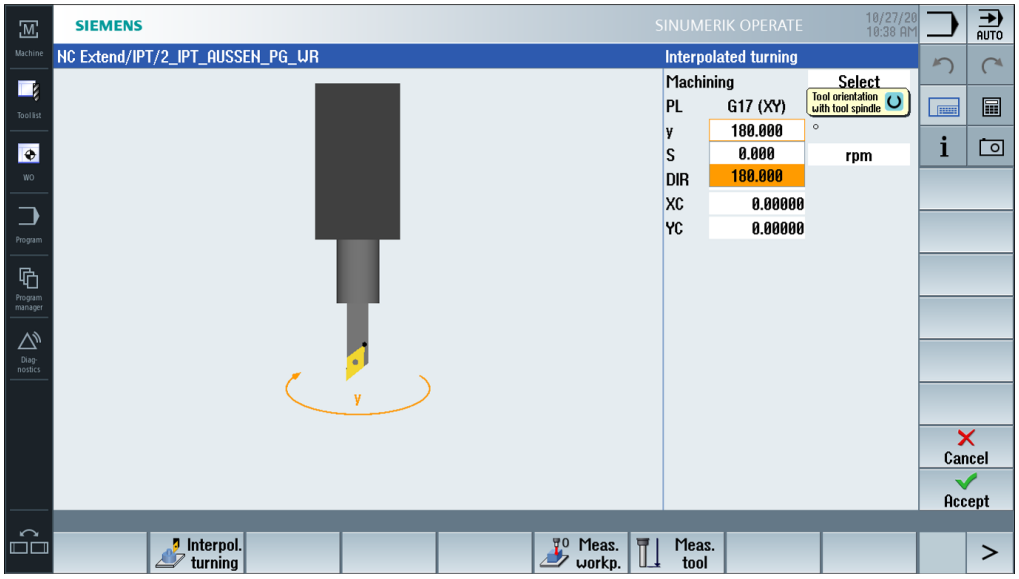
	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: P57		Option: P57		Option: P57

	SINUMERIK 840D sl
	Option: P57

For interpolation turning, the CNC guides the turning tool in a position-controlled way around a fixed workpiece. For this purpose, the main spindle of the machining center is switched to position-controlled mode (also called axis mode). The feed axes interpolate in a spiral/circular arrangement in the x-y/x-y-z plane, while at the same time the main spindle with the turning tool tracks the feed axes.

Application examples: Stock removal (longitudinal/face turning and contour turning), recessing, grooving on cubic workpieces produced on milling machines;

Examples for workpieces: Sealing surfaces for master brake cylinder, brake caliper, hydraulic valve housing, pivot bearing, or gearbox housing.



Benefits

- **Off-center turning on machining centers (milling machines) and turning machines with a B axis with reduced machining time because machining operations are completed in one clamping operation**
- **Reduced investment costs**

Automation 14

14.1 Robot connection

14.1.1 SINUMERIK Run MyRobot /EasyConnect

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

☑	SINUMERIK 840D sl
	Basic configuration

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

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

Note:

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

Benefit

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

14.1.2 SINUMERIK Run MyRobot /Handling

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

*) ab SINUMERIK RUN MyRobot /Handling V4.0

☑	SINUMERIK 840D sl
	Option: via SISW

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

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

Note

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

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

Benefit

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

14.1.3 SINUMERIK Run MyRobot /Direct Handling

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

	SINUMERIK 840D sl
	Option: via SISW

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

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

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

Benefit

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

14.2 Multiple clamping

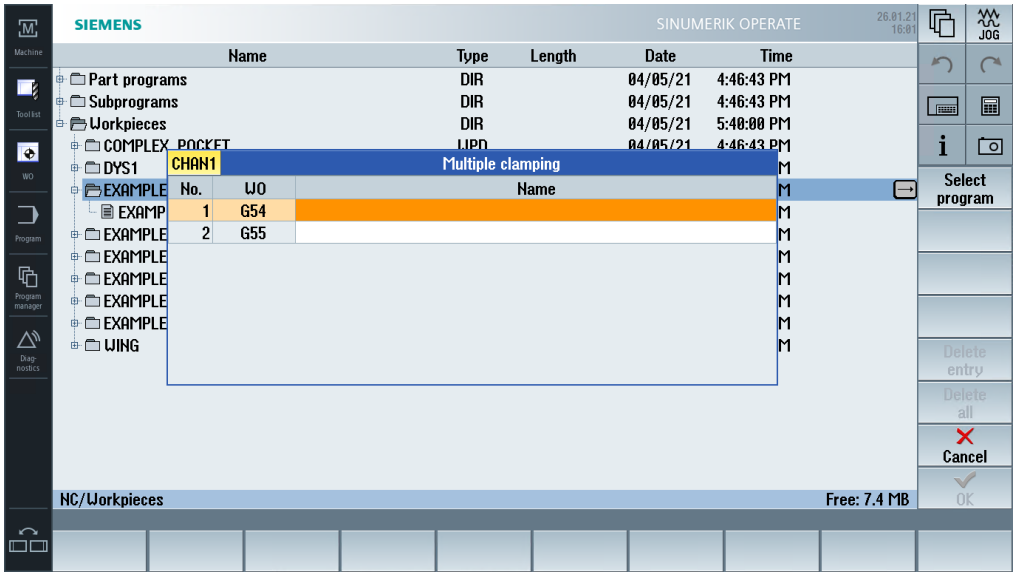
☑	SINUMERIK 828D SW24x	☑	SINUMERIK 828D SW26x	☑	SINUMERIK 828D SW28x
	Option: P17 (ShopMill/ShopTurn)		Option: P17 (ShopMill/ShopTurn)		Option: P17 (ShopMill/ShopTurn)

☑	SINUMERIK 840D sl
	Option: P17 (ShopMill/ShopTurn)

With the Multiple clamping function, you can optimize identical or different workpiece programs for multiple clamping at the push of a button.

The necessary programs for each individual workpiece are created with ShopMill. The Multiple clamping function automatically generates a new "multiple clamping program" from these programs. In this program, the order of all tools used is rearranged for all workpieces, i.e. the number of tool changes will be reduced significantly, thus increasing the productivity. The flow pattern continues for all used tools of all workpieces.

Without the use of the multiple clamping function, the control system would process the workpiece programs sequentially, i.e. the same tools would be used and substituted several times, thus leading to loss of time.



Benefits

- **When machining different workpieces, the Multiple clamping function minimizes the number of tool changes to a minimum and thus ensures a decisive increase in productivity.**

Digitalization 15

15.1 Digitalization - Overview

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

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

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

- Order preparation and execution
  - Manage MyResources /Tools (Page 110)
  - Manage MyResources /Programs (Page 111)
- Efficiency and flexibility in production
  - Analyze MyPerformance (in line) (Page 112)
  - Analyze MyPerformance /OEE Monitor (MindSphere) (Page 113)
  - Analyze MyPerformance /OEE Tuning (MindSphere) (Page 114)
  - Manage MyMachines (Page 115)
  - Manage MyMachines /Remote (Page 116)
- Machine availability
  - Analyze MyMachine /Condition (SINUMERIK Edge) (Page 122)
  - Optimize MyMachining /Trochoidal (SINUMERIK Edge) (Page 119)
  - Optimize MyMachining /AC AUTO (Page 117)
- Improved machining processes
  - SINUMERIK Edge (Page 118)
  - Analyze My Workpiece /Capture (SINUMERIK Edge) (Page 120)
  - Analyze My Workpiece /Monitor (SINUMERIK Edge) (Page 121)
  - Analyze My Workpiece /Toolpath (SINUMERIK Edge) (Page 123)

Note

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

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

15.2 Manage MyResources /Tools

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

	SINUMERIK 840D sl
	Option: via SISW

Manage MyResources /Tools permits centralized management of tools.

- Factory-wide management of tools:

  Tool planning and stock management for tools and components
- Tool stock during magazine assignment:

  Overview of the current tool data, setup dialog for loading and unloading the physical tool
- Data exchange and evaluation:

  Standard interface to measuring devices for tool presetting, provision of tool history

Benefit

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

15.3 Manage MyResources /Programs

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

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

Manage MyResources /Programs supports you in managing NC packages.

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

Benefit

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

15.4 Analyze MyPerformance (in line)

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

☑	SINUMERIK 840D sl
	Option: via SISW

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

- Acquisition of machine data to calculate key parameters:

  Key parameters provide information about the state of the plant and allow optimization potential to be identified.

- Acquisition of alarms/messages:

  Avoiding disruptions to production by supporting maintenance and deriving preventive maintenance measures.

- Provision of the widest range of evaluations and analyses:
  - Display of the average duration of disruptions and their percentage of the planned machine usage time.
  - Weak point analysis by showing the effects on upstream and downstream stations.

Benefits

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

15.5 Analyze MyPerformance /OEE Monitor (MindSphere)

	Control-independent
	Option: via SISW

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

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

Benefits

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

15.6 Analyze MyPerformance /OEE Tuning (MindSphere)

	Control-independent
	Option: via SISW

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

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

Benefits

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

15.7 Manage MyMachines

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

☑	SINUMERIK 840D sl
	Option: via SISW

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

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

Benefit

- **Increased availability, utilization and efficiency of machine tools.**

15.8 Manage MyMachines /Remote

☑	**SINUMERIK 840D sl**
	Option: via SISW

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

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

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

Benefits

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

15.9 Optimize MyMachining /AC AUTO

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

	SINUMERIK 840D sl
	Option: via SISW

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

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

Precondition:

- Run MyCC /IMD
- Run MyHMI /3GL

Optional:

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

Benefit

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

15.10 SINUMERIK Edge & Applications

15.10.1 SINUMERIK Edge

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

	SINUMERIK 840D sl
	Option: via SISW

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

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

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

Benefits

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

15.10.2 Optimize MyMachining /Trochoidal

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

	SINUMERIK 840D sl
	Option: via SISW

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

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

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

Benefits

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

15.10.3 Analyze MyWorkpiece /Capture

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

	SINUMERIK 840D sl
	Option: via SISW

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

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

Benefit

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

15.10.4 Analyze MyWorkpiece /Monitor

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

	SINUMERIK 840D sl
	Option: via SISW

Analyze MyWorkpiece /Monitor provides you with the following functions:

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

Benefit

- **Documentation of workpiece and process quality**

15.10.5 Analyze MyMachine /Condition

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

	SINUMERIK 840D sl
	Option: via SISW

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

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

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

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

Benefits

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

15.10.6 Analyze MyWorkpiece /Toolpath

	Control-independent
	Option: via SISW

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

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

Analyze MyWorkpiece /Toolpath provides various analysis functions:

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

Benefits

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

Tools and information

16

16.1 DXF reader

	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	Option: P56		Option: P56		Option: P56

	SINUMERIK 840D sl
	Option: P56

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

- **DXF Reader in the Program Manager**

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

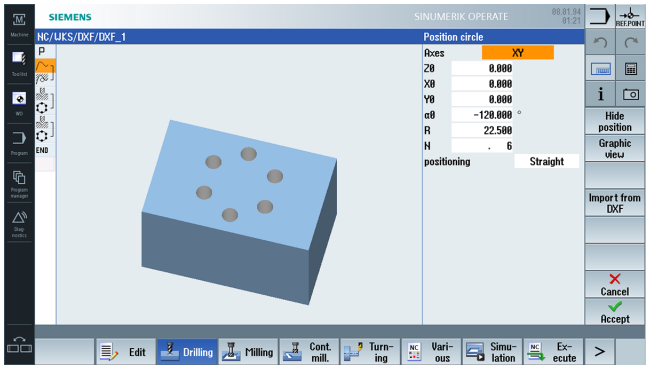
- **Import DXF data in the contour calculator**

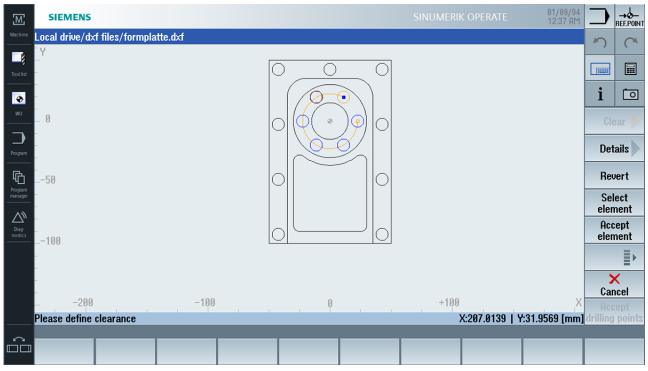
  You can either clean the imported DXF data automatically or select the desired layer yourself.

  Cleaned DXF data can be buffered as new DXF file.

- **Import DXF data in position patterns**

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





Benefits

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

16.2 SinuTrain for SINUMERIK Operate

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

☑	**SINUMERIK 840D sl**
	Option: C43, via SISW

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

SinuTrain for SINUMERIK Operate taps into the following applications:

In work preparation:

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

In training:

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

For presentation:

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

Note

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

Note

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

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

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 (http://www.siemens.com/cnc4you)

Safety functions

17

17.1 SINUMERIK Safety Integrated

✓	SINUMERIK 828D SW24x	✓	SINUMERIK 828D SW26x	✓	SINUMERIK 828D SW28x
	Option: refer to the catalog		Option: refer to the catalog		Option: refer to the catalog

✓	SINUMERIK 840D sl
	Option: refer to the catalog

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

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

The functional safety for machine tools covers:

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

Benefits

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

17.2 Collision avoidance

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

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

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

Note

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

17.2.1 Protect MyMachine /Axis Lock

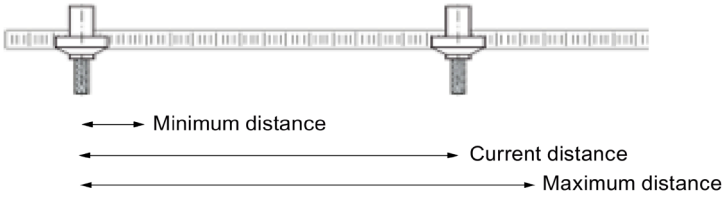
	SINUMERIK 828D SW24x		SINUMERIK 828D SW26x		SINUMERIK 828D SW28x
	not available		not available		not available

	SINUMERIK 840D sl
	Option: 6FC5800-0AN06-0YB0

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

Braking takes place automatically with a predefined delay.

- Up to 20 axis pairs
- Multi-channel



Benefits

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

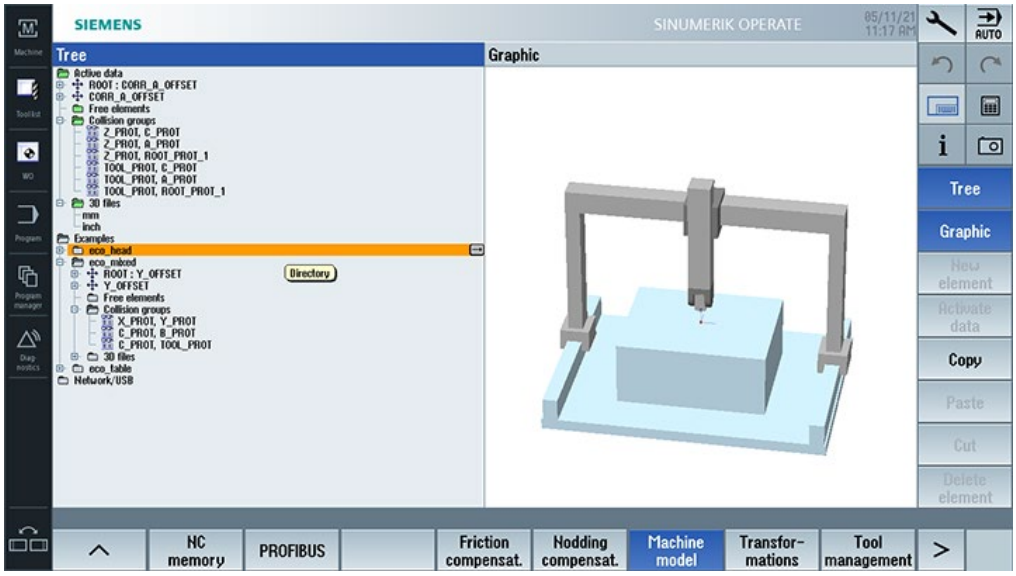
17.2.2 ProtectMyMachine /3D Primitives

☑	SINUMERIK 828D SW24x	☑	SINUMERIK 828D SW26x	☑	SINUMERIK 828D SW28x
	Option: S03		Option: S03		Option: S03

☑	SINUMERIK 840D sl
	Option: S03

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

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



Benefits

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

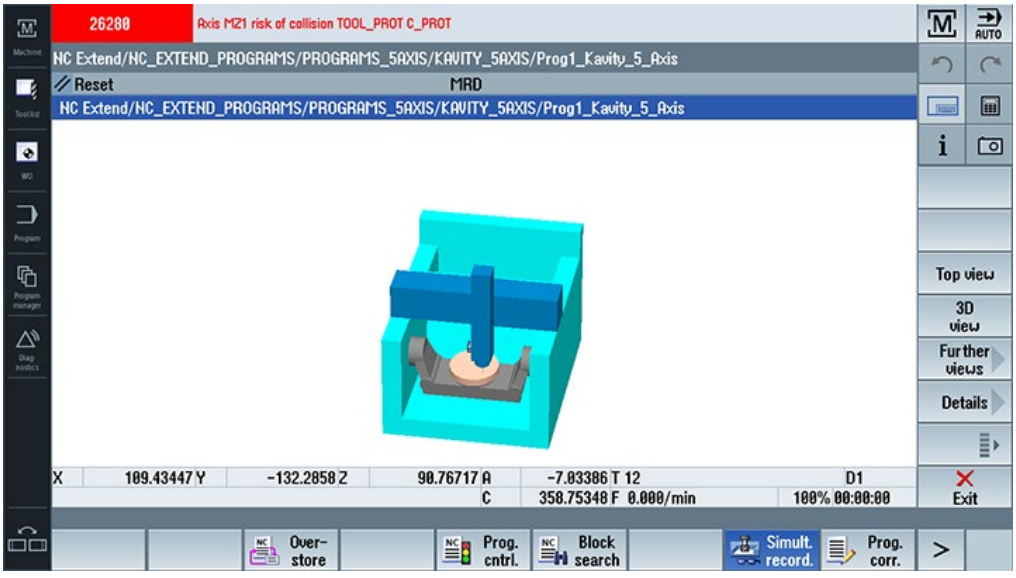
17.2.3 Protect MyMachine /3D STL

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

	SINUMERIK 840D sl
	Option: S02

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

- Such as Protect MyMachine /3D Primitives
- Up to 500 protection area elements (based on CAD – STL format)
- In the modes JOG, MDI, Automatic



Benefit

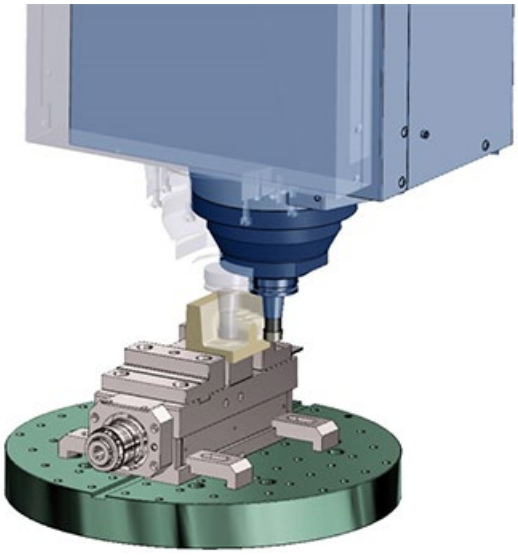
- **Machine-oriented mapping of complex protection areas possible.**

17.2.4 Protect MyMachine /Open

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

	SINUMERIK 840D sl
	Option: S04

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



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

Benefit

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

Index

T