

SIEMENS

SINUMERIK 840D sl

SINUMERIK Operate - Turning




Control system overview
for machine tools' sales people

Preface	
Introduction	1
System overview	2
CNC operation in manual mode (JOG)	3
Tool management	4
User memory	5
Data transfer	6
Graphical support functions	7
CNC operation in automatic mode (AUTO)	8
CNC performance capability and optimization functions	9
CNC programming methods	10
Simulation	11
CNC technology cycles	12
Complete machining	13
Multi-channel machining	14
PC software	15
Integrated safety functions	16
Option list for the SINUMERIK package	17
Summary of unique features	18

Legal information

Warning notice system

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

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


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

Qualified Personnel

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

Proper use of Siemens products

Note the following:

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

Trademarks

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

Disclaimer of Liability

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

Preface

Scope of validity

This document provides you with an overview of the range of functions included in the **SINUMERIK 840D sl** Version **4.5** for turning machines.

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

Organization of the information

- Of the varied functional features of the SINUMERIK products, only those are listed which are of direct value to the machine user.
- All functions contained in the machine's basic configuration will be identified as follows:
 - Basic configuration
- All functions not contained in the machine's basic configuration will be identified as follows:
 - Option: ...
- You can find a summary of the most important unique selling points of the SINUMERIK 840D sl in the chapter "Summary of highlights".
- 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
Email: xxx@machinemanufacturer.com

Service

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

Homepage:

<http://www.machinemanufacturer.com>

Contact person at Siemens

Country	Name	e-mail	Phone
Argentina	Santiago Fernandez Veron	santiago_fernandez.veron@siemens.com	+54 (11) 4738-3348
Belgium	Pieter Vanderhaeghen	pieter.vanderhaeghen@siemens.com	+32 253-69697
Bosnia and Herzegovina	Helmut Stralz	helmut.stralz@siemens.com	+43 51 707 29115
Brazil	Gustavo Marino	marino.gustavo@siemens.com	+ 55 (11) 3908-1752
Bulgaria	Helmut Stralz	helmut.stralz@siemens.com	+43 51 707 29115
China	Hong Xia Mu	hongxia.mu@siemens.com	+86 (10) 64765419
Denmark	Stefan Karlstrand	stefan.karlstrand@siemens.com	+46 500 774-148
Germany (Headquarters)	Gerhard Micka	gerhard.micka@siemens.com	+49 9131 98 3314
Finland	Juha Meriaho	juha.meriaho@siemens.com	+358 50 59 26181
France	François Chevalier	francois.chevalier@siemens.com	+33 (0)1 49 22 35 19
Great Britain	Tony Bennison	tony.bennison@siemens.com	+44 780 882 2054
India	Narayanan Shankar	narayanan.shankar@siemens.com	+91 99451 88837
Indonesia	Andy Lesmono	andy_lesmono@cncdesign.com.au	+62 21 7918 6001
Italy	Nicodemo Megna	nicodemo.megna@siemens.com	+39 335 6328927
Canada	Sagar Arora	sagar.arora@siemens.com	+1 (416) 270-4964
Korea	Kim Sung Hyun	sunghyun.kim@siemens.com	+ 82 55 268 1906
Croatia	Matjaz Mlinsek	matjaz.mlinsek@siemens.com	+386 1 47 46 152
Lithuania	Juha Meriaho	juha.meriaho@siemens.com	+358 50 59 26181
Malaysia	Ridwan Aziz	ridwan_aziz@cncdesign.com.au	+603 5621 6126
Mexico	Marco Lopez	marco.lopez@siemens.com	+52 (55) 5328-2000
Netherlands	Frank de Korte	frank.de.korte@siemens.com	+31 70 333 1568
Norway	Stefan Karlstrand	stefan.karlstrand@siemens.com	+46 500 774-148
Austria	Helmut Stralz	helmut.stralz@siemens.com	+43 51 707 29115
Portugal	Molaguero Godoy	juan.molaguero.ext@siemens.com	+34 670929001
Romania	Matjaz Mlinsek	matjaz.mlinsek@siemens.com	+386 1 47 46 152
Russia	Alexander Kudinov	alexander.kudinov@siemens.com	+7 495 737 2442
Sweden	Stefan Karlstrand	stefan.karlstrand@siemens.com	+46 500 774-148
Switzerland	Hans-Peter Kueng	hans-peter.kueng@siemens.com	+41 585 581 524
Serbia Montenegro	Helmut Stralz	helmut.stralz@siemens.com	+43 51 707 29115
Slovakia	Matjaz Mlinsek	matjaz.mlinsek@siemens.com	+386 1 47 46 152
Slovenia	Matjaz Mlinsek	matjaz.mlinsek@siemens.com	+386 1 47 46 152
Spain	Molaguero Godoy	juan.molaguero.ext@siemens.com	+34 670929001
Taiwan	Jerry Lin	jerry.lin@siemens.com	+886 4 2261 9225
Thailand	Rajeev Madhyastha	rajeev_madhyastha@cncdesign.com.au	+66(0)2993 7485
Czech	Eva Klocova	eva.klocova@siemens.com	+420 605726829
Turkey	Taner Okayi	taner.okayi@siemens.com.tr	+90 216 459 3906
Hungary	Istvan Joo	istvan.joo@siemens.hu	+36 1 471 1598
USA	Jon Cruthers	jon.cruthers@siemens.com	+1 (847) 952 4124
Vietnam	Pham-Vu Trung	pham-vu.trung@siemens.com	+84 908 346 534

Homepage:

For further information please visit ...

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

<http://www.automation.siemens.de/doconweb>

Table of contents

	Preface	3
1	Introduction.....	9
	1.1 Application.....	9
	1.2 Machine spectrum.....	10
2	System overview.....	11
	2.1 SINUMERIK 840D sl.....	11
	2.2 Operator panel fronts	12
	2.3 Operator panel equipment	16
3	CNC operation in manual mode (JOG).....	19
	3.1 TSM universal cycle.....	19
	3.2 Positioning cycle	20
	3.3 Stock removal cycle	21
	3.4 Measure tool	22
	3.5 Measure workpiece.....	23
	3.6 Work offsets	24
4	Tool management.....	25
	4.1 Tool table	25
	4.2 Monitoring of tool life and workpiece count.....	26
	4.3 Replacement tools	27
	4.4 Setup data.....	27
5	User memory	29
	5.1 Buffered CNC work memory.....	29
	5.2 CompactFlash card.....	29
	5.3 Hard disk.....	30
	5.4 Card reader	30
	5.5 USB Hub 4	31
6	Data transfer.....	33
	6.1 Program manager	33
	6.2 Data transfer using the USB memory stick.....	34
	6.3 RCS Commander.....	34
	6.4 Ethernet networking	35

7	Graphical support functions	37
7.1	Animated Elements	37
7.2	Onboard documentation.....	38
8	CNC operation in automatic mode (AUTO)	39
8.1	Block search.....	39
8.2	Basic block display	40
8.3	Program control.....	41
8.4	Simultaneous recording	42
9	CNC performance capability and optimization functions	43
9.1	80bit NANO ^{FP} accuracy.....	43
9.2	Jerk limitation	44
9.3	Dynamic feedforward control	45
9.4	Look Ahead	46
10	CNC programming methods	47
10.1	Overview of programming methods	47
10.2	programGUIDE and SINUMERIK CNC programming	48
10.2.1	Introduction	48
10.2.2	Program editor	48
10.2.3	Languages	49
10.2.4	programGUIDE input support.....	50
10.3	ShopTurn.....	51
10.3.1	Introduction	51
10.3.2	Sequence editor	51
10.3.3	Interlinking of sequences	52
10.3.4	Broken-line graphics	53
10.4	Online ISO dialect interpreter.....	54
11	Simulation	55
11.1	2D simulation	55
11.2	3D simulation	56
12	CNC technology cycles	57
12.1	Highlights of machining cycles.....	58
12.1.1	Stock removal along contour with blank contour	58
12.1.2	Engraving cycle.....	59
12.1.3	Counterspindle cycle.....	60
12.2	Residual material detection.....	61
12.2.1	Residual material detection during turning.....	61
12.2.2	Detection of residual material when milling.....	62
12.3	In-process measuring for workpieces and tools.....	63

13	Complete machining	65
13.1	End face machining (TRANSMIT).....	65
13.2	Peripheral surface machining (TRACYL).....	66
14	Multi-channel machining	67
14.1	Overview	67
14.2	programSYNC job list	68
14.3	Double editor.....	69
14.4	Simulation	70
15	PC software	71
15.1	CAD reader for PC	71
15.2	SinuTrain.....	72
15.3	Computer-based training	72
16	Integrated safety functions	73
17	Option list for the SINUMERIK package	75
18	Summary of unique features	77
	Index	79

Introduction

1.1 Application

With the SINUMERIK 840D sl and the operating software SINUMERIK Operate you have a tailored solution for all CNC turning machines used worldwide.

When operating the machine tool, you are supported by graphic help screens and Animated Elements.

The functions in the manual mode enable a quick, practical machine set-up. More specifically, this consists of calculating the workpiece position in the machine, as well as the maintenance and dimensioning of the tools in use.

You have three different options when programming:

- programmGUIDE and SINUMERIK CNC programming, with a wide selection of technology cycles for medium series and large series production.
- ShopTurn, with automatic cascading (linking) of associated machining steps for small series production.
- ISO dialect and SINUMERIK CNC programming language with IOSO dialect interpreter.

The SINUMERIK 840D sl with the operating software SINUMERIK Operate offers a control configuration that covers all of the required application areas without subsequent commissioning and training costs:

- Easy-to-use interface for all machine functions
- DIN/ISO programming on the machine or offline via the CAD/CAM system
- Graphic programming
- Complete machining
- In-process measuring for workpieces and tools

1.2 Machine spectrum

The SINUMERIK 840D sl with the operating software SINUMERIK Operate is perfectly suited for the following types of machines:

1. Single-slide turning machines with X and Z axes
 - Turning operations
 - Centric drilling on the end face
2. Like 1.) with rotating tools (C axis mode)
 - Any drilling or milling on the end face and outer surface
3. Like 1.) or 2.) with additional Y axis
 - Any drilling or milling on the end face and outer surface
4. Like 1.), 2.) or 3.) with additional counterspindle for front and rear machining

System overview

2.1 SINUMERIK 840D sl

SINUMERIK 840D sl is open, flexible and powerful and offers a standard and integrated structure for operator control, programming and visualization. Integrated into the new, especially compact and reliable SINAMICS S120 drive system, it constitutes a completely digital system. It is best-suited for medium and high performance ranges.



- Digital drive controller
- Modular design for drive controller and power unit
- Up to 31 axes/spindles in up to 10 processing channels
- Intelligent control functions meeting the highest standards of machining technology



You can find further information in catalog NC 61

Highlights



- **Flexible application and guaranteed future through modular design and comprehensive CNC functions**
- **Best dynamics, precision and surface quality through use of high performance processors**
- **Integrated, certified safety functions for man and machine are possible with Safety Integrated**

2.2 Operator panel fronts

These operator panel fronts have an integrated CNC keyboard. We offer matched machine control panels for the version with mechanical keys (protection class IP54) and the version with membrane keys (protection class IP65). You have two choices:

OP 010C

Operator panel 483mm wide, 10.4" display, integrated CNC keyboard, with frontal USB port for memory stick, version with mechanical keys, separate machine control panel



Highlights



- All relevant functions at a glance, thanks to horizontal and vertical softkeys
- Brilliant color display, balanced and high-quality design of operator components
- Simple data handling with easily accessible USB memory stick

OP 010

Operator panel 483mm wide, 10.4" display, integrated CNC keyboard, with frontal USB port for memory stick, version with **membrane-type keys**, separate machine control panel



Highlights



- All relevant functions at a glance, thanks to horizontal and vertical softkeys
- Brilliant color display, balanced and high-quality design of operator components
- Simple data handling with easily accessible USB memory stick

OP 010S

Operator panel 310mm wide, 10.4" display, mechanical keyboard, with frontal USB port for memory stick, separate CNC keyboard and machine control panel



Highlights



- Option of especially slim design
- All relevant functions at a glance, thanks to horizontal and vertical softkeys
- Brilliant color display, balanced and high-quality design of operator components
- Simple data handling with easily accessible USB memory stick

OP 015

Operator panel 483mm wide, 15" display, membrane keys, with frontal USB port for memory stick, separate CNC keyboard and machine control panel



Highlight



- Option of large display for better readability and graphic display, especially for high-performance machines
- All relevant functions at a glance, thanks to horizontal and vertical softkeys
- Brilliant color display, harmonized and high-quality design of operator components
- Simple data handling with easily accessible USB memory stick

OP 019

Operator panel 483 mm wide, 19" display with resolution 1280*1024, membrane keys, can also be operated with gloves, capacitive sensors for fast key initiation, integrated key lock to protect against operator errors, USB 2.0 connector socket for mounting in control desks, separate CNC keyboard and machine control panel



Highlight



- **New operator panel front with especially large display for better readability and graphic display, especially for high-performance machines**
- **All relevant functions at a glance, thanks to horizontal and vertical softkeys**
- **Brilliant color display, harmonized and high-quality design of operator components**

2.3 Operator panel equipment

TCU

The Thin Client Unit (TCU) for a distributed configuration allows the SINUMERIK operator panel front to be spatially separated and/or permits several operator panels to be connected to one SINUMERIK NCU.

- Fewer parts for specifically flat design
- Competitive purchase price and inexpensive replacement parts



Highlights



- Rugged and cost-optimized solution, as there is no hard disk
- Signal transmission between PCU/NCU and operator panel front via Industrial Ethernet

PCU 50

We offer the PCU 50 for customers who specify a hard disk and/or supplementary Windows-based software.

- Windows XP operating system
- Additional CNC memory through hard disk
- Additional PCI slots
- Additional CF card slot
- DVI interface



Highlights



- Flexible software expansion on Windows platform
- Flexible hardware expansion via PCI slots

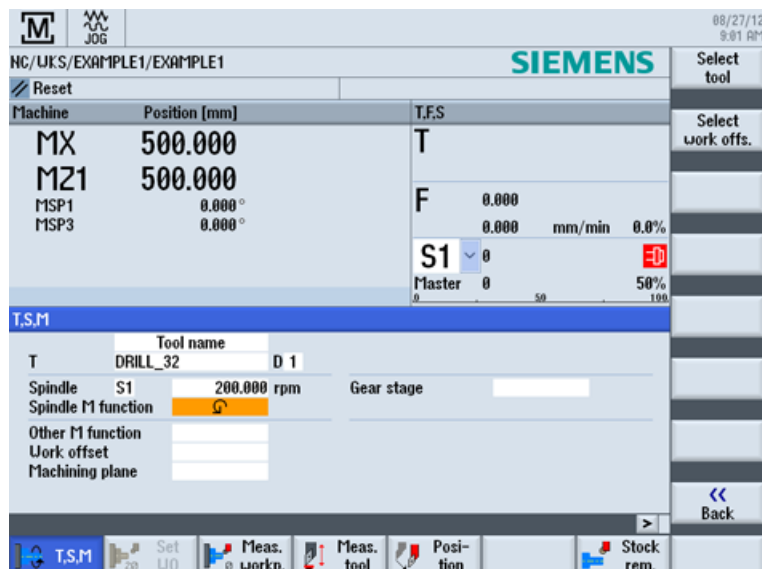
CNC operation in manual mode (JOG)

3.1 TSM universal cycle

Basic configuration

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

- Tool change with direct access via the tool table (T)
- Spindle speed and direction (S)
- M functions (M)
- Activation of work offsets



Highlight



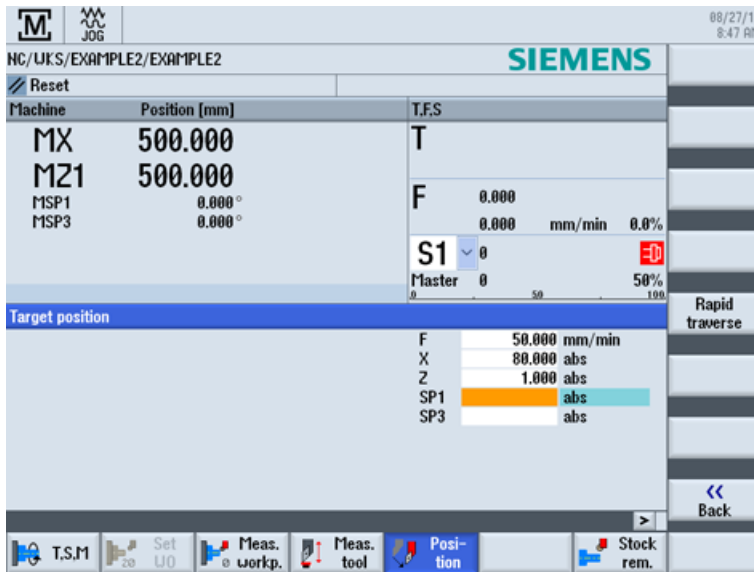
- Take over and change in tools directly from the tool table

3.2 Positioning cycle

Basic configuration

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

- Linear axes / spindles
- Feedrate / rapid traverse



Highlight



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

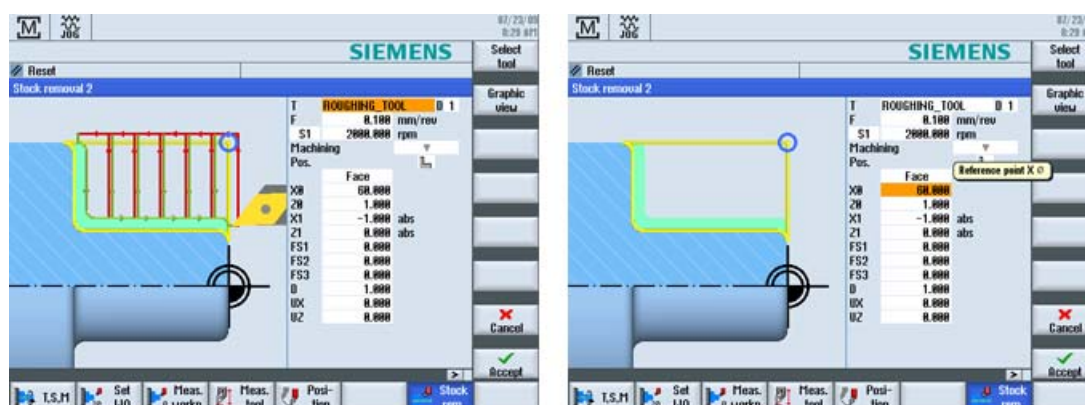
3.3 Stock removal cycle

Basic configuration

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

The following parameters can be specified:

- Roughing or finishing
- Undercut for soft collet chucks



Highlight



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

3.4 Measure tool

Basic configuration

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

The following variants are supported:

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



Highlight



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

3.5 Measure workpiece

Basic configuration

The workpieces can be measured as follows:

- Reference tool

The screenshot shows the Siemens CNC control interface in manual mode (JOG). The top bar displays the date and time (08/27/12, 9:07 AM) and the Siemens logo. Below the logo, the text 'NC/UKS/EXAMPLE1/EXAMPLE1' is visible. The main display area is divided into several sections:

- Machine Data:** Shows 'Mmachine' with 'MX' and 'M21' at '500.000' mm. It also displays 'MSP1' at '97.725°' and 'MSP3' at '0.000°'.
- Tool Data (T.F.S.):** Shows 'T ROUGHING_T80 A' with 'R0.800', 'Z39.000', and 'X55.000'. It also displays 'F 0.000' and 'S1 150' (Spindle Speed) with 'Master 200' and '75%'.
- Measure: front edge:** Shows a 3D model of a workpiece being measured. It displays 'Values UO' with 'Z 141.000' and 'Work offset G54 0.000'. It also shows 'Measured values 20'.
- Bottom Bar:** Contains icons for 'T.S.M', 'Set UO', 'Meas. workp.', 'Meas. tool', 'Position', and 'Stock rem.'.

Highlight



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

3.6 Work offsets

Basic configuration

The following adjustable work offsets are available to you:

- A basic offset
- Maximum of 99 work offsets (G54, G55 ...)
- Each work offset with axis rotation and fine offset
- Rotations, scaling and mirroring also shown as icons

	X	Y	Z	SP1	SP3
DRF	0.000	0.000	0.000	0.000	0.000
Basic reference	0.000	0.000	0.000	0.000	0.000
Total basic UO	0.000	0.000	0.000	0.000	0.000
G54	0.000	0.000	141.000	0.000	0.000
Transf. reference	0.000	0.000	0.000	0.000	0.000
Programmed UO	0.000	0.000	0.000	0.000	0.000
Cycle reference	0.000	0.000	0.000	0.000	0.000
Total UO	0.000	0.000	141.000	0.000	0.000

Highlights



- Flexible machining due to great number of adjustable work offsets
- Optimum overview of all active work offsets

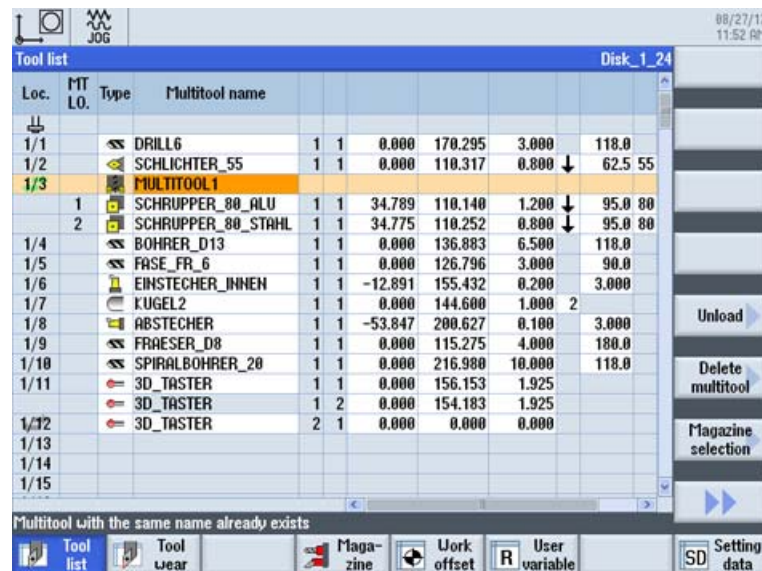
Tool management

4.1 Tool table

Basic configuration

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

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



The screenshot shows the 'Tool list' window in a CNC control system. The table contains the following data:

Loc.	MT LO.	Type	Multitool name						
1/1		DRILL6		1	1	0.000	170.295	3.000	118.0
1/2		SCHLICHTER_55		1	1	0.000	110.317	0.800	62.5 55
1/3		MULTITOOL1							
	1	SCHRUPPER_80_ALU		1	1	34.789	110.140	1.200	95.0 80
	2	SCHRUPPER_80_STAHL		1	1	34.775	110.252	0.800	95.0 80
1/4		BOHRER_D13		1	1	0.000	136.883	6.500	118.0
1/5		FASE_FR_6		1	1	0.000	126.796	3.000	90.0
1/6		EINSTECHER_INNEN		1	1	-12.891	155.432	0.200	3.000
1/7		KUGEL2		1	1	0.000	144.600	1.000	2
1/8		ABSTECHE		1	1	-53.847	200.627	0.100	3.000
1/9		FRAESER_D8		1	1	0.000	115.275	4.000	180.0
1/10		SPIRALBOHRER_20		1	1	0.000	216.980	10.000	118.0
1/11		3D_TASTER		1	1	0.000	156.153	1.925	
		3D_TASTER		1	2	0.000	154.183	1.925	
1/12		3D_TASTER		2	1	0.000	0.000	0.000	
1/13									
1/14									
1/15									

Below the table, there is a status bar with the message: "Multitool with the same name already exists". At the bottom, there are navigation buttons: Tool list, Tool wear, Magazine, Work offset, User variable, and Setting data.

Highlights



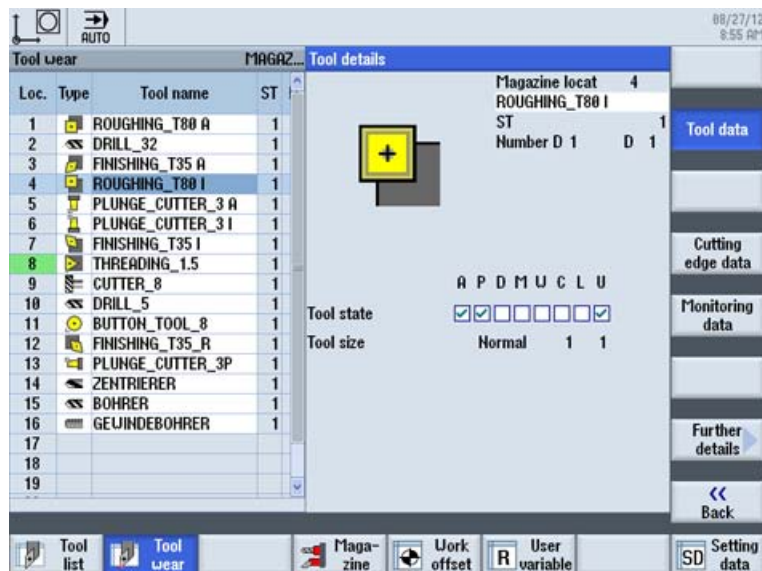
- All tool data at a glance
- Simple and secure handling via unmistakable tool names

4.2 Monitoring of tool life and workpiece count

Basic configuration

You can use SINUMERIK 840D sl to monitor the service life of your tools and the number of tool changes. 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
- If the desired tool is not in the magazine, the SINUMERIK 840D sl will request a manual tool change.



Highlights



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

4.3 Replacement tools

Basic configuration

If needed, you can even use the SINUMERIK 840D sl to manage replacement tools (sister tools). Tools with the same name are created as replacement tools. The replacement tools are identified with an incrementing number in the ST column.

Highlight

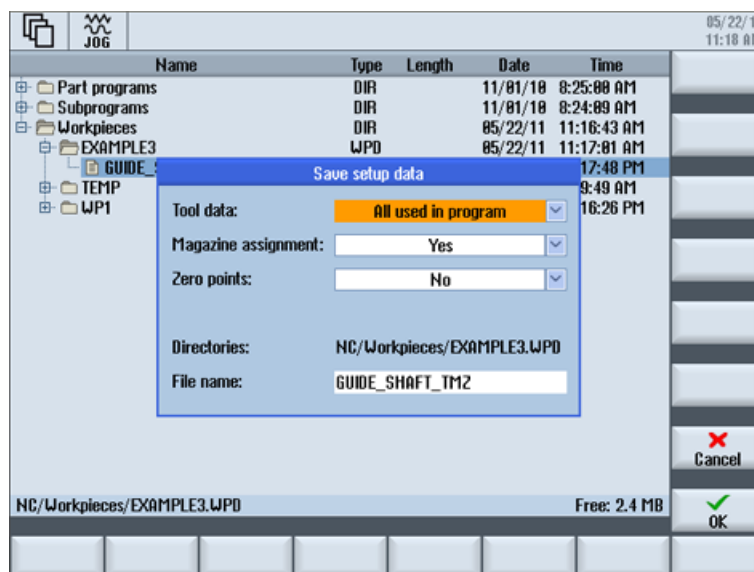


- Automatic tool exchange for unmanned operation

4.4 Setup data

Basic configuration

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



Highlight



- Time savings when you save the part programs

User memory

5.1 Buffered CNC work memory

SINUMERIK 840D sl
NCU 710

Basic configuration: 3 MByte, expandable to
9 MByte

SINUMERIK 840D sl
NCU 720 and NCU 730

Basic configuration: 3 MByte, expandable to
15 MByte

Highlight



- Exceptionally large storage space in basic configuration already

5.2 CompactFlash card

Option: Additional user memory on the
CompactFlash card of the NCU

For SINUMERIK 840D sl, the NCU has a compact flash card for the system software. On it, at least 256 MB up to 2 GB may be used optionally as additional user memory. This option is possible in conjunction with the TCU and offers an alternative to the PCU 50 hard disk.

Executing the part program directly from the CompactFlash Card is possible.

Highlight



- Powerful and reliable solution for handling larger part programs

5.3 Hard disk

Option: PCU 50

A hard disk with 12 GB of user memory is available on the PCU 50 for expansion of the CNC memory.

Highlights



- Hard disk can still be used as a data carrier

5.4 Card reader

Option: Card reader

Using the SINUMERIK card reading unit in combination with CF, SD and MMC cards, data can be archived and user data exchanged. The connection is made via a USB interface. The card reader can be booted. Cards can be inserted and removed during operation.



Highlights



- Insertion and removal of cards during operation possible
- Simple data exchange without opening the control cabinet

5.5 USB Hub 4

Option: USB Hub 4

Using the Industrial USB Hub 4, you can connect up to four I/O devices such as e.g. USB stick and keyboard. The Industrial USB Hub 4 differs from commercially available USB hubs mainly in the fact that it is suitable for use in rugged industrial environments (IP65).

Highlight



- **The system to be operated can be more easily, safely and reliably handled**

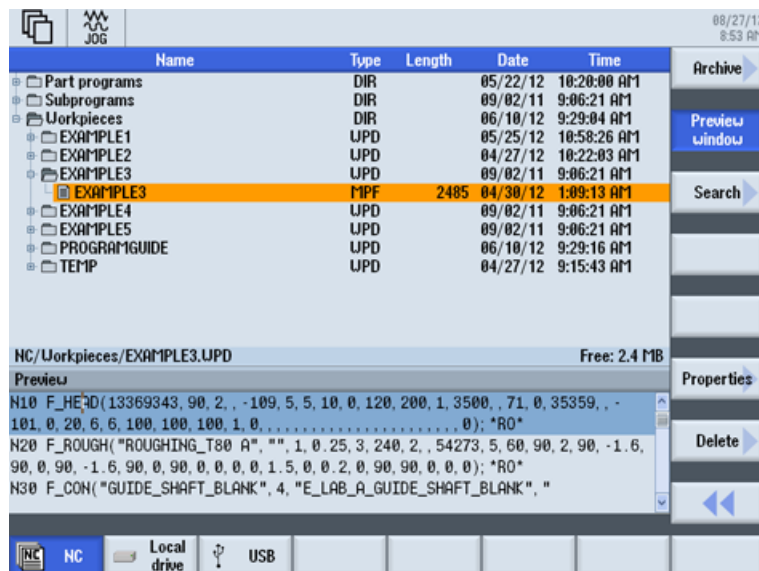
Data transfer

6.1 Program manager

Basic configuration

The program manager offers you an optimum overview of the directories and programs, and very easy-to-use file handling. It supports plain names of up to 24 characters for directories and files. Subdirectories can also be managed on external storage media such as CF cards and USB sticks.

All storage media including the network drives are displayed in the program manager. The part programs can be edited in all media.



Highlights



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

6.2 Data transfer using the USB memory stick

- ☑ Option: Only USB memory stick required

A USB memory stick can be used on the front side of the operator panel.

User data stored on the PC can be quickly transferred to the internal CNC memory. Execution from USB memory sticks is not possible for security reasons. Instead, we offer a CompactFlash Card solution.



Highlights



- User-friendly solution, as memory slot is located at front
- Extremely cost-efficient, as no software option required

6.3 RCS Commander

- ☑ Option: RCS Commander on CD-ROM
- ☑ Option: RCS Host remote diagnostics function

The RCS Commander is a powerful free-of-charge tool for the PC. It allows you to move data easily into the CNC controller using drag & drop. In addition, it offers a convenient means of viewing the contents of the CNC screen display on the PC. Simply connect your PC or Notebook on which the RCS Commander is installed to the Ethernet interface on the front panel. SINUMERIK 840D sl automatically configures the network. No knowledge of networks is required.

The SINUMERIK 840D sl also supports remote diagnostics via the Ethernet (see options). When connecting the PC to several machines, only one PC license is necessary for remote diagnostics via Ethernet. All machine tool controller diagnostic functions are also available in remote diagnostics.

Highlights



- Simple data transfer by drag & drop
- Remote diagnostics means shorter response times and reduced service costs
- Easy transfer of the contents of the screen display from the CNC to the PC; ideal for training purposes etc. (connecting a projector to the PC)

6.4 Ethernet networking

Basic configuration

The SINUMERIK 840D sl is set up for Ethernet (TCP/IP) networking (RJ45 connection).

- The data transfer rate is 10/100 Mbps.
- Remote access to the controller 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.

Highlight

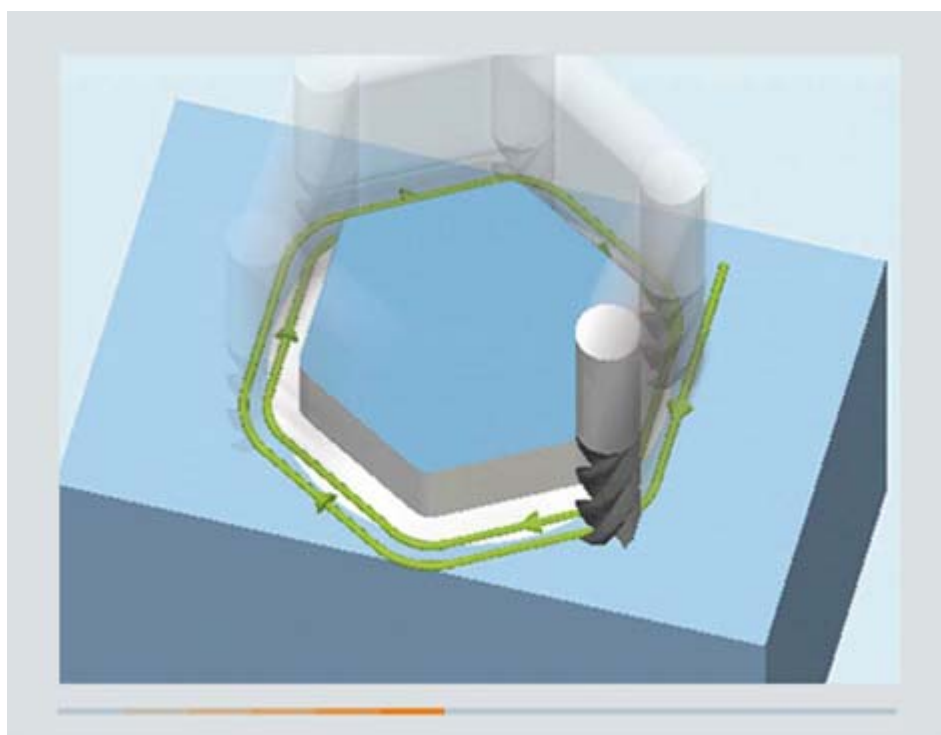


- Favorably-priced and simple connection to Windows PCs via Ethernet (TCP/IP)
- No software needs to be installed on the servers

Graphical support functions

7.1 Animated Elements

To illustrate which parameters affect what in machining operations, the SINUMERIK 840D sl offers a new input support function with animated element sequences. For instance, the difference between chip breakage and chip removal when drilling or tapping can be shown.



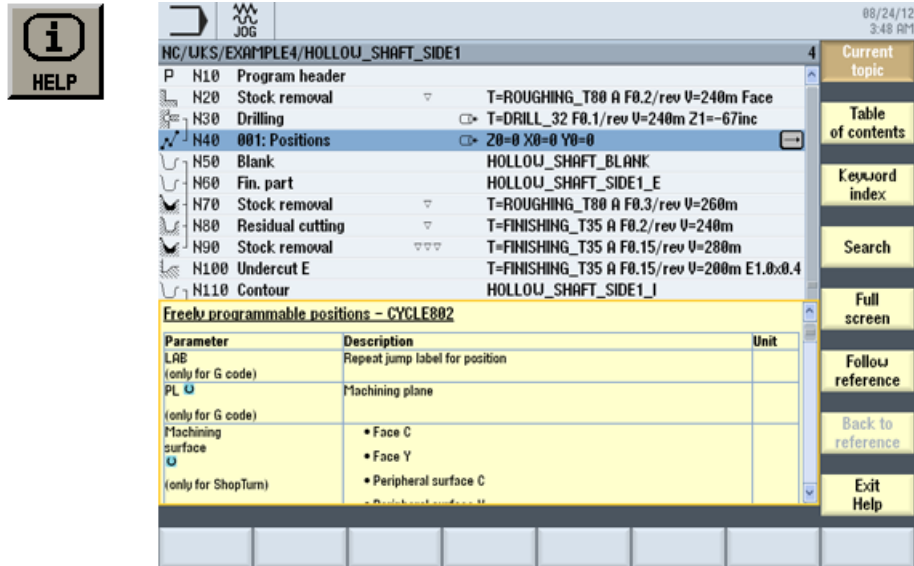
Highlight



- Process reliability during the setup
- Increased reliability at program input by easily understood depiction of selection options

7.2 Onboard documentation

For each input field in the operating screens, SINUMERIK 840D sl automatically displays help in the form of a "cursor text". The SINUMERIK 840D sl provides further information in the form of a complete context-sensitive help system with many useful details and graphics.



Highlight



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

CNC operation in automatic mode (AUTO)

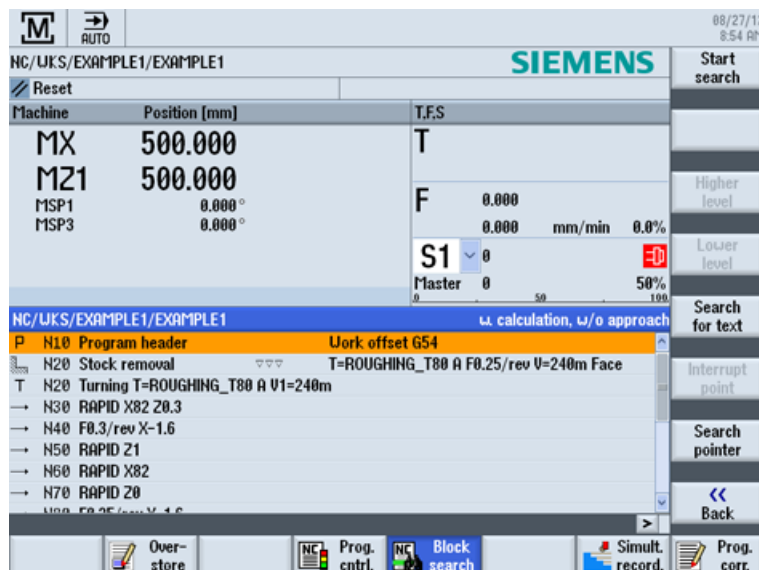
8.1 Block search

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 are 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 interruption point
- To any CNC block in the DIN/ISO programs
- To any subroutine levels in DIN/ISO programs
- In work plan programs
- In position patterns when programming work plans
- Accelerated block search in large mold making programs



Highlights



- 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

8.2 Basic block display

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.

NC/UKS/EXAMPLE2/EXAMPLE2		Basic blocks
N30	Contour	DRIVE_SH X48.4 Z-13.415
N40	Stock removal	T=ROUGH Z-42.968
N50	Residual cutting	T=FINISHI G03 X59.692 Z-50.77 K-11.232 I-7.308
N60	Stock removal	T=FINISHII G01 X68.058 Z-66.567
N70	Thread long.	T=THREAC Z-78.8 X77.6
T	N80 T=0	G03 X80 Z-78.854 K-13.4 I-0.002
END	End of program	

Highlights



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

8.3 Program control

Basic configuration

Single block

Single block mode can be activated for startup of the program. For this purpose a program stop occurs after each traversing block.

Work plan programs maintain the alternative of stopping processing after each plane infeed.

Program test

Programs can be checked before processing in a program test mode. The program is executed to completion with stationary axes. This is especially meaningful in connection with the simultaneous recording option (real-time simulation).

Reduced rapid traverse

You have the facility of additionally limiting the traversing speed for rapid traverse so that when running-in a new program with rapid traverse, no undesirable high traversing speeds occur. In the rapid traverse mode, the traversing speed of the axes is reduced to the percentage value (0-100%) entered in RG0.

Program editing

In machine condition STOP, the program can be edited directly at the location of the fault, e.g. erroneous DIN/ISO blocks or wrongly parameterized sequences. After correcting the program you can continue machining.

Repositioning to the contour (REPOS)

In machine condition STOP, the machining axes may be moved to and away from the workpiece surface with the handwheel or the direction keys.

Highlights

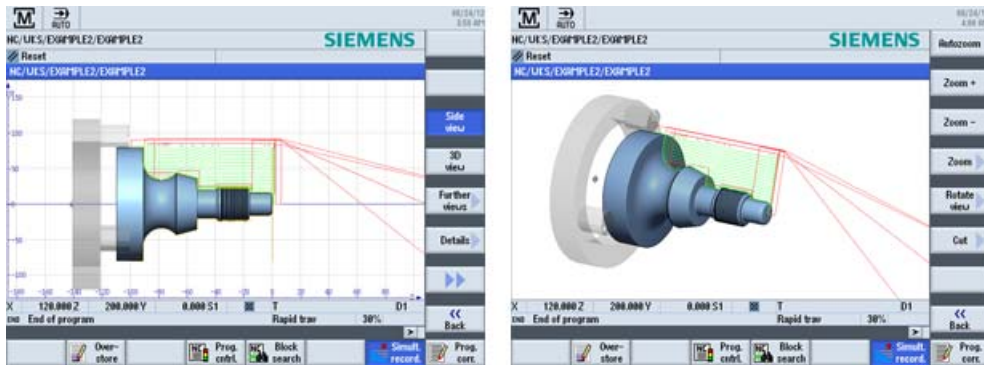


- Secure positioning of new part programs
- Continue machining quickly after interruptions

8.4 Simultaneous recording

Option: Simultaneous recording

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



Highlight



- Machining can also be monitored in a complex machine room

CNC performance capability and optimization functions

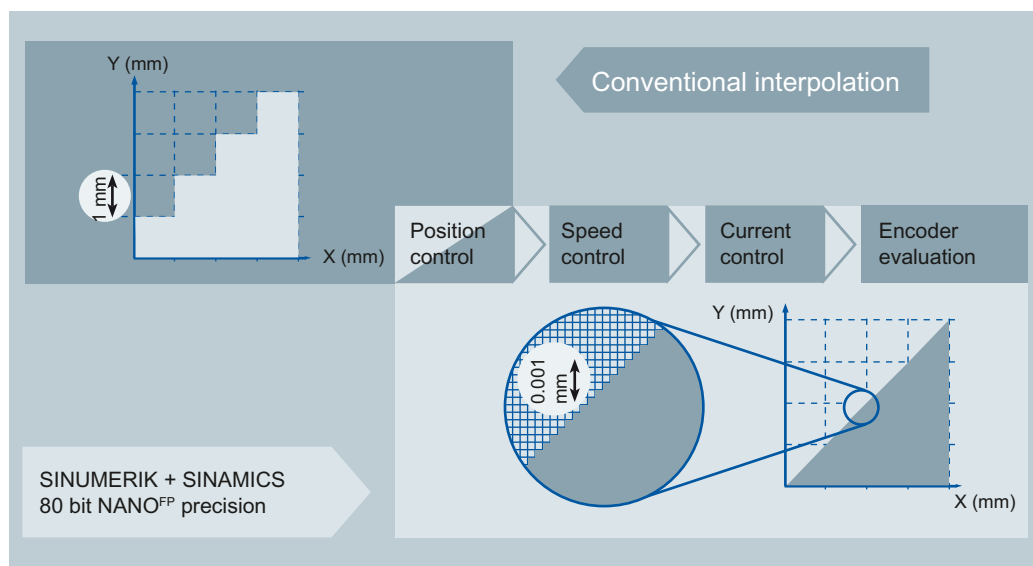
9

9.1 80bit NANO^{FP} accuracy

☑ Basic configuration

The accuracy of the workpiece is determined by more factors than just the mechanical characteristics of the machine. The CNC controller also contributes to a critical degree towards the precision of the workpieces. The SINUMERIK 840D sl offers a multitude of CNC functions for this purpose.

The SINUMERIK 840D sl and the SINAMICS drive calculate using 80-bit 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 power and closed-loop speed control and also for sensor evaluation of the drive.



Highlight

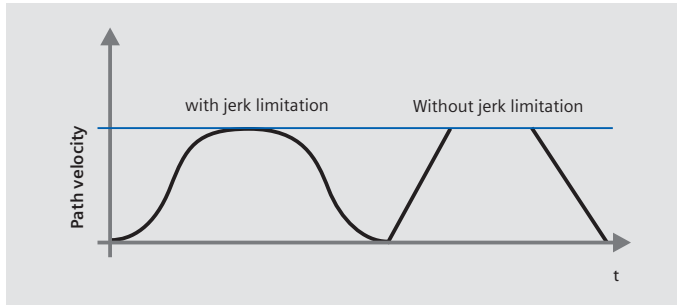


- Maximum precision in the workpiece results based on extremely high calculation accuracy

9.2 Jerk limitation

Basic configuration

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



Highlights



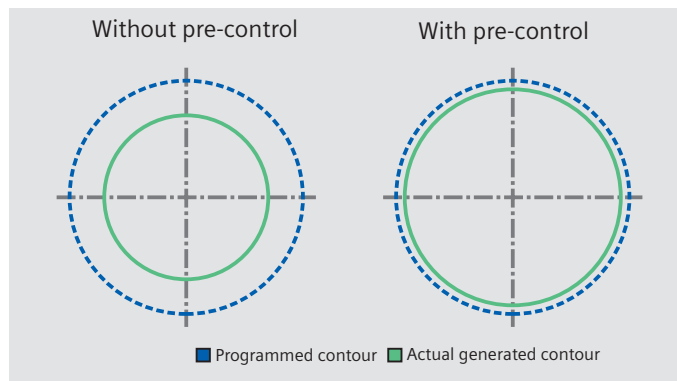
- Longer machine lifespan through protection of the mechanical components
- Higher path accuracy through softer acceleration

9.3 Dynamic feedforward control

☑ 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 tool path feedrates. This is clarified with a circularity test on the machine.

Example:



Highlight

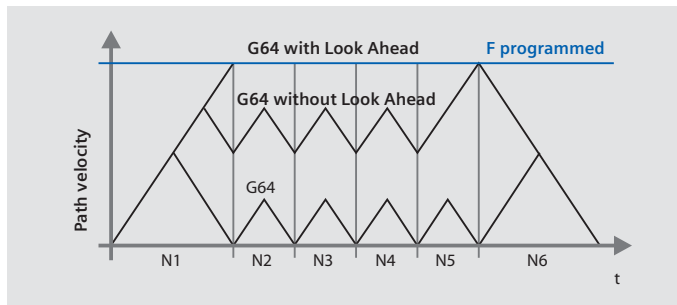


- Higher path accuracy through compensation of contouring errors

9.4 Look Ahead

Basic configuration

The »Look Ahead« function is a means of optimizing the machining speed by »looking ahead« over a parameterizable number of traversing blocks. With tangential block transitions, the axis is accelerated and decelerated beyond block boundaries, so that no drops in velocity occur.



Highlight



- Shorter machining times through optimum velocity control

CNC programming methods

10.1 Overview of programming methods

The SINUMERIK 840D sl offers you a choice of the following programming methods:

programGUIDE and SINUMERIK CNC programming

With programGUIDE you obtain the perfect combination of the SINUMERIK CNC programming language and the parameterization of technology cycles. The wide choice of technology cycles and the ease of parameterization allows you to reduce the programming time. The parameter input is supported by Animated Elements.

The SINUMERIK language statements with CNC high-level language elements offer you a very high degree of flexibility and guarantee minimum machining time.

programmGUIDE and SINUMERIK CNC programming are particularly suitable for medium series and large series production.

ShopTurn - Machining step programming

Machining operations such as stock removal, grooving or thread cutting are shown in ShopTurn in the form of worksteps. In this way CNC programs – even for complex machining operations – are very compact and easily read. Associated sequences are automatically interlinked and can be assigned any position patterns. ShopTurn offers you the shortest programming times even for highly demanding machining tasks. The parameter input is supported by Animated Elements.

ShopTurn is particularly well suited for small series production.

ISO dialect and SINUMERIK CNC programming language

The SINUMERIK 840D sl allows you to perform ISO programming using the SINUMERIK CNC programming language combined with or exclusively in ISO dialect.

The online ISO dialect interpreter offers you the opportunity to use CNC programs from other manufacturers.

Step for step you can increase the performance capability by using SINUMERIK CNC programming.

Highlights



- Whether you use programGUIDE or ShopTurn – in either case the full range of technological cycles, position patterns and geometries is available to you
- Compatibility with the ISO dialect of other controller manufacturers is feasible

10.2 programGUIDE and SINUMERIK CNC programming

10.2.1 Introduction

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

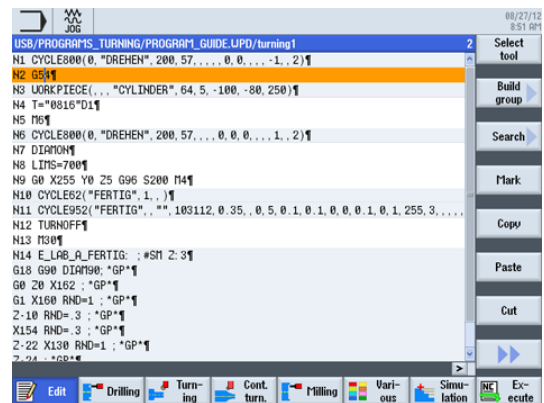
Programming with programGUIDE is available in the basic scope of the SINUMERIK 840D sl.

10.2.2 Program editor

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

The following functions are included in the program editor:

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



Highlights



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

10.2.3 Languages

The SINUMERIK 840D sl's CNC interpreter 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**
The user can define his 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.
- **Calculation operations**
The following mathematical calculation operations are available for linking the variables:
calculation 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 ...

Highlights

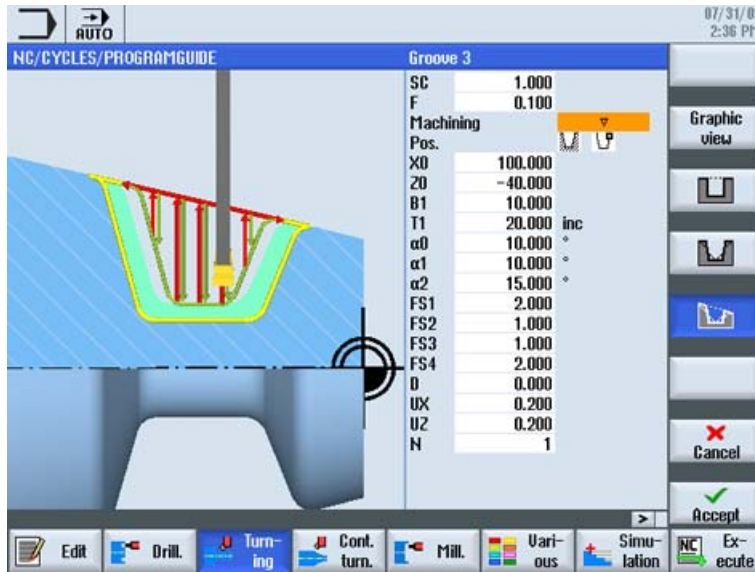


- Established programming according to DIN 66025
- Unbeatable range of commands for flexibility and time saving while programming

10.2.4 programGUIDE input support

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

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



Highlights



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

10.3 ShopTurn

10.3.1 Introduction

Option: Machining step programming ShopTurn

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

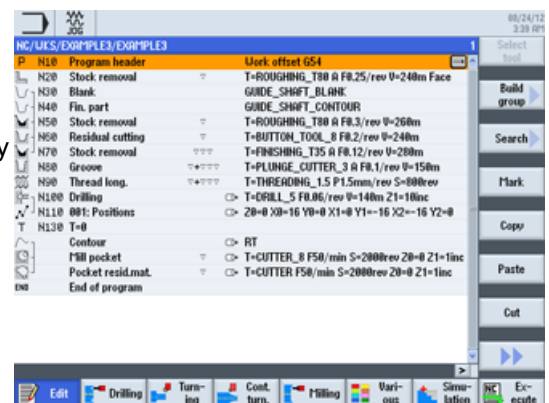
- Sequence editor
- Interlinking of sequences
- Broken-line graphics

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

10.3.2 Sequence editor

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

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



Highlights

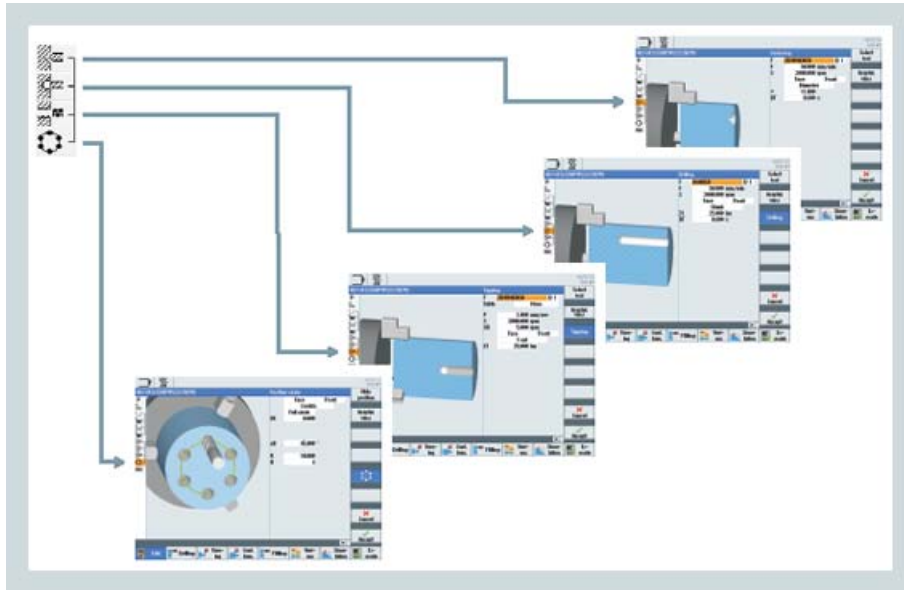


- Intuitive program input, without knowledge of DIN/ISO or Operating Manual
- Compact, clearly arranged machining programs
- Reducing the programming time by graphical input masks and copying / inserting machining steps

10.3.3 Interlinking of sequences

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

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



Highlight



- Reduced programming time due to linking of machining steps

10.3.4 Broken-line graphics

While programming, the previously entered sequences will be continuously displayed to scale. A simulation is not required for this. The switch over between the sequence program and the broken-line graphics is performed by the "Graphics View" softkey.

- Turning view
- Front face and peripheral side



Highlight



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

10.4 Online ISO dialect interpreter

Basic configuration

It is always useful to be able to speak a foreign language. This is true even for a global player such as the SINUMERIK 840D sl. If you prefer classic ISO programming you can continue to use it. You can even mix ISO programming with the SINUMERIK CNC programming language. This enables you to increase the productivity and flexibility of your machine step by step.

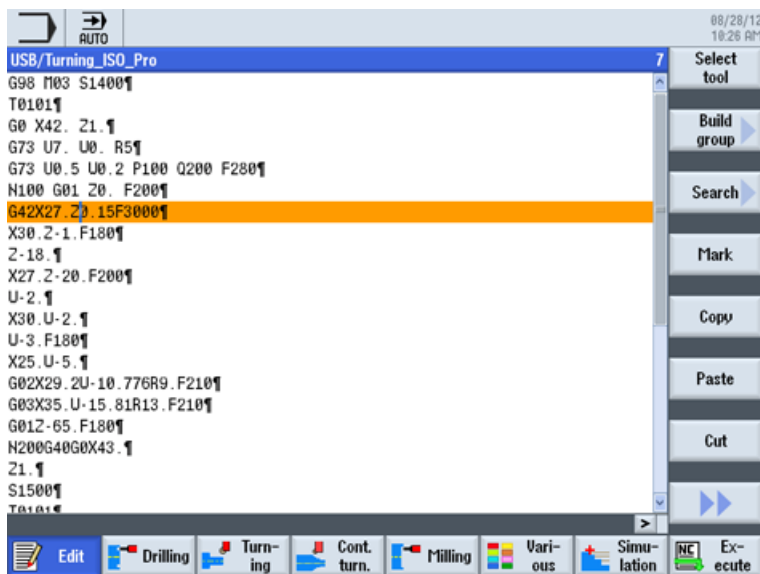
In the controller, G commands from Siemens are interpreted as standard.

ISO dialect codes and Siemens codes can be mixed within a part program, but not within an NC block.

The switch over between Siemens operating mode and ISO dialect is performed using the following two G commands:

- G290 - "Siemens" NC programming language active
- G291 - "ISO dialect" NC programming language active

The performance capability of the ISO dialect extends even as far as using the cycles G73 to G89, such as cycle G84 for tapping.



Highlights



- Even first-time users can initially continue programming the way they are accustomed to
- ISO dialect and SINUMERIK CNC programming languages can be mixed within part programs

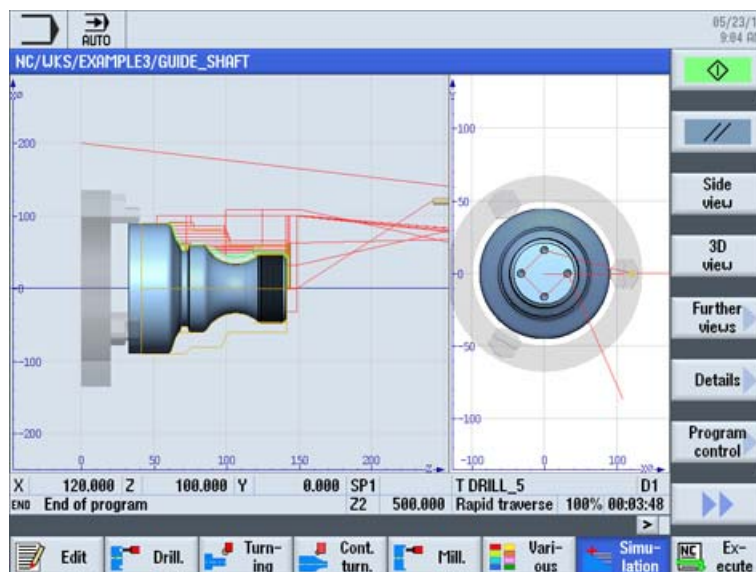
Simulation

11.1 2D simulation

Basic configuration

The SINUMERIK 840D sl 2D simulation offers you the facility to make optimum and reliable preparations for machining workpieces, including the detection of collisions. Calculating the machining time also supports optimum calculation of tooling costs.

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



Highlights



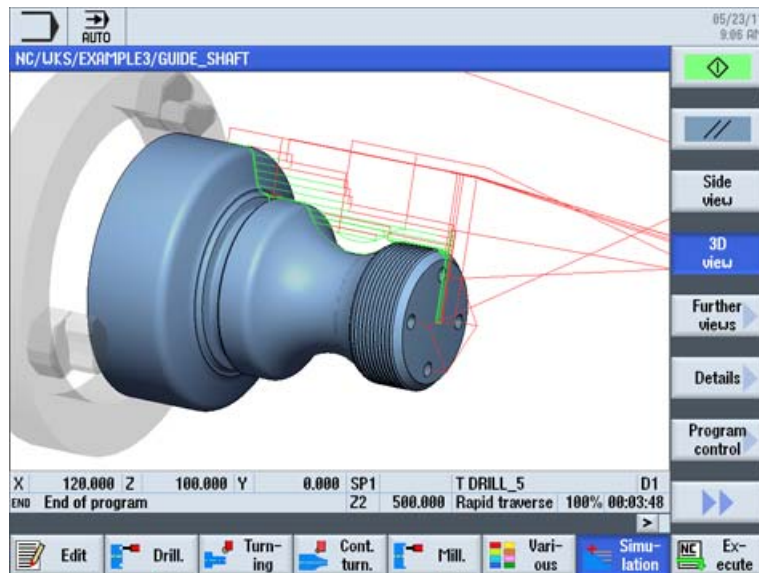
- Maximum process reliability through simulation using real geometry values
- Perfect clarity by showing the workpiece dimensions with a scale

11.2 3D simulation

Option: 3D simulation

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

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



Highlights



- Particularly realistic simulation through representation of the tool
- Optimum help and reliability in programming and in quotation costing

CNC technology cycles

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



The SINUMERIK 840D sl offers you a unique range of CNC technology cycles for standard machining – including an engraving cycle. The assignment to the machining positions is performed very simply using a wide selection of ready position patterns.

For sustained accuracy of workpieces in an ongoing machining process, the SINUMERIK 840D sl supports you with the measuring cycles' optional package.

Thanks to the integrated geometry processor, you can create even complex contours directly at the CNC controller. In this case, partially defined contour elements are automatically calculated. In addition you can use the optional CAD reader to process DXF files.

The geometry processor supports you when inputting contours. The stock removal motions are generated fully automatically by the SINUMERIK 840D sl. To be able to achieve maximum productivity, you can pre-machine using a large plate angle. The optional identification of residual material permits selective remachining of the remaining material using a small plate angle.

Highlights



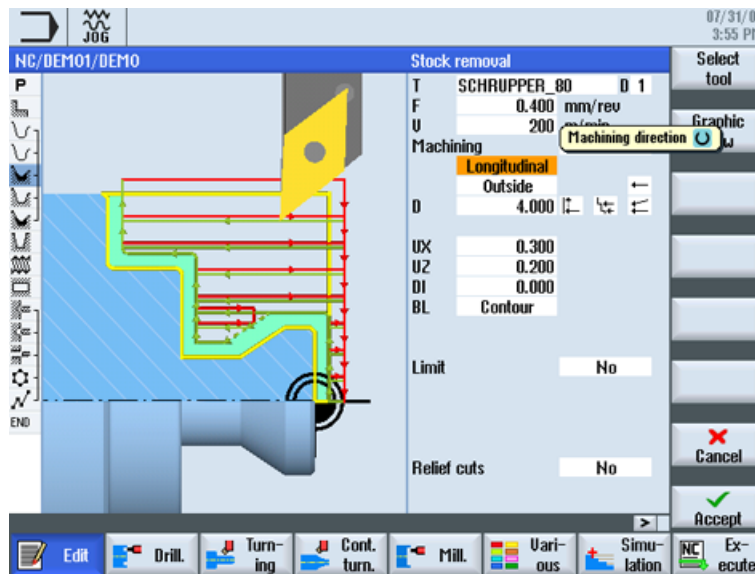
- Significant simplification of programming, even for complex jobs, using CNC technology cycles
- Continuity of cycles for programGuide and ShopTurn

12.1 Highlights of machining cycles

12.1.1 Stock removal along contour with blank contour

Basic configuration

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



- Processing any contour calculator geometry
- Cylindrical blank, freely-defined blank, blank as allowance of finished-part contour
- Longitudinal / face / contour-parallel roughing on outside and inside
- Processing sloping contours (relief cuts)
- Consideration of tool's setting and plate angle
- Grooving any contours on outside, inside or end face
- Plunge turn any contours on outside, inside or end face
- Finishing with negative allowances (for machining electrodes)
- Roughing with feed interruption to reduce flow chips
- Optional number of cutting passes with orientation to workpiece edges
- Finishing with alternating cutting depth for longer tool life
- Arbitrary limitation to the machining segment with automatic blank actualization

Highlights



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

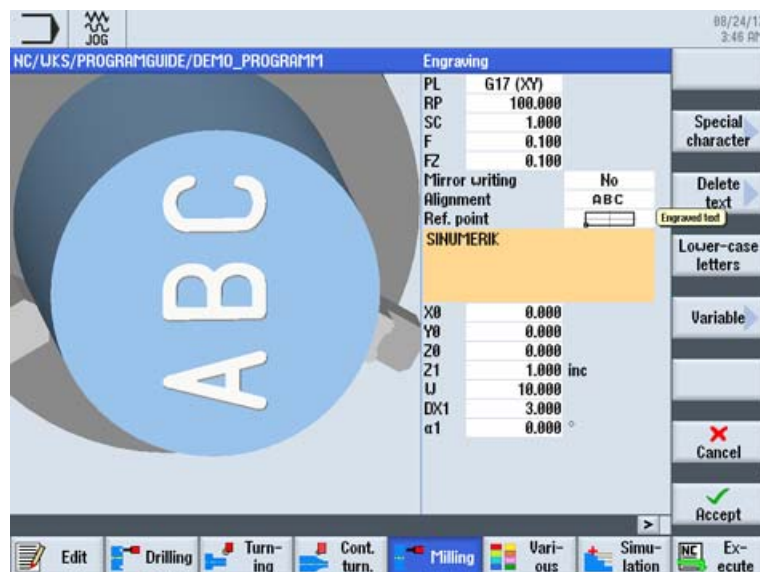
12.1.2 Engraving cycle

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"`).



Highlights

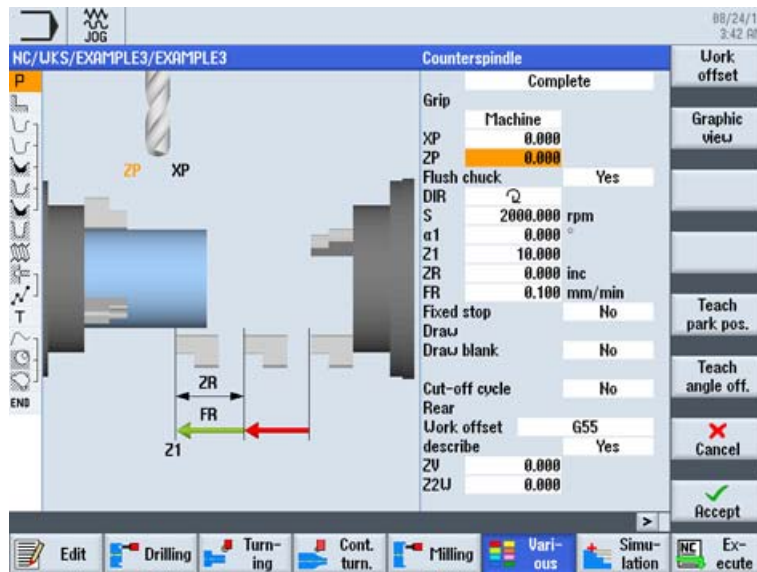


- Reduction of set-up times by complete machining on one machine
- Simple program input of engraving

12.1.3 Counterspindle cycle

Basic configuration

The SINUMERIK 840D sl allows the use of a fully-fledged counterspindle. The main spindle and counterspindle can be operated under conditions of angular synchronism.



DIN/ISO programming

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

Machining step programming

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

Highlights



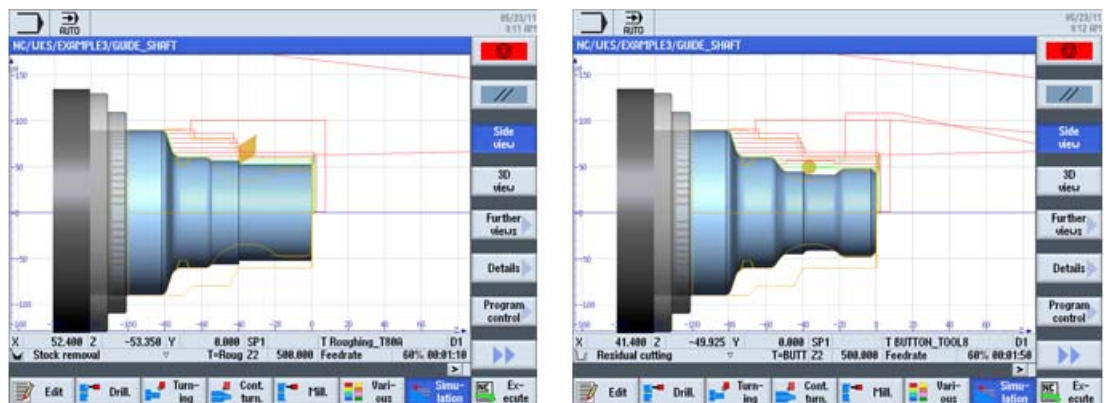
- Simple and secure programming of all counterspindle functions
- High quality of workpieces by workpiece transfer in synchronous spindle mode

12.2 Residual material detection

12.2.1 Residual material detection during turning

Option: Residual material detection

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



Highlight



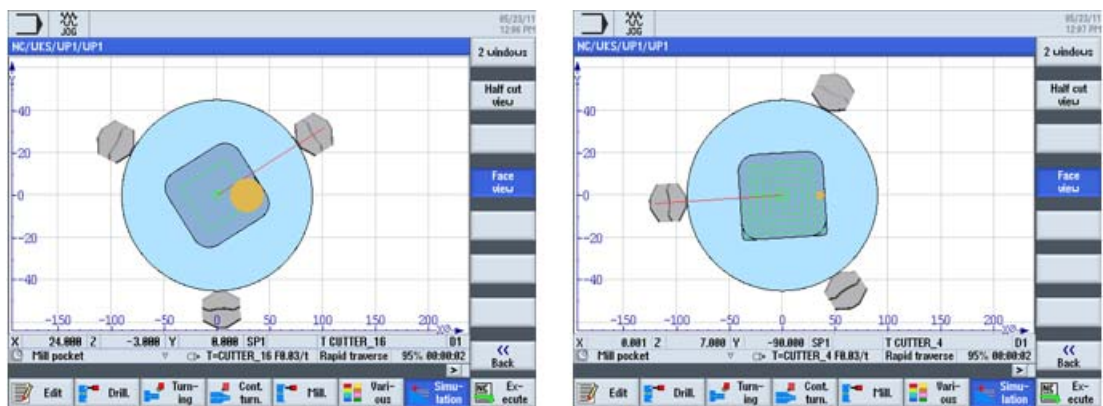
- Time saving through avoiding idle cuts during residual stock removal

12.2.2 Detection of residual material when milling

Option: Residual material detection

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

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



Highlight



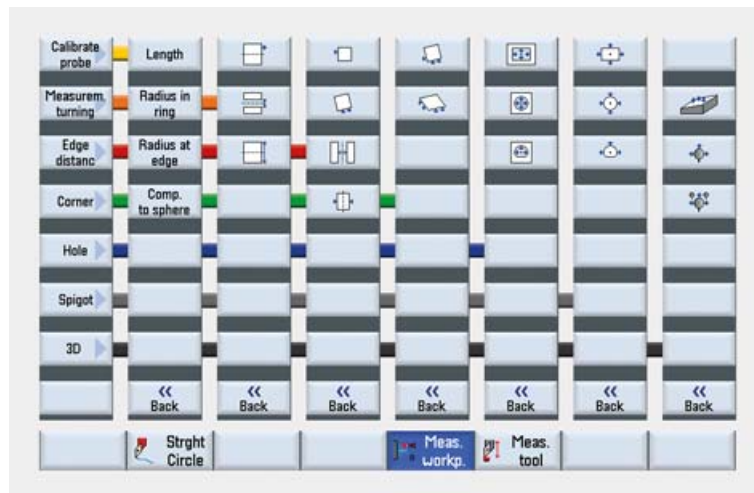
- Shorter machining times through the use of a large tool for the substantial part of the stock removal and a smaller tool for the remaining residual material
- Avoidance of non-cutting movements while achieving extremely simple programming

12.3 In-process measuring for workpieces and tools

Option: Measuring cycles

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

The following cycles are available for workpiece measurement:



The following measurement variants are available for tool measurement:

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

The following measuring tasks can be made:

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

Highlights



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

Complete machining

13.1 End face machining (TRANSMIT)

- ☑ Option: TRANSMIT and peripheral surface transformation

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

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

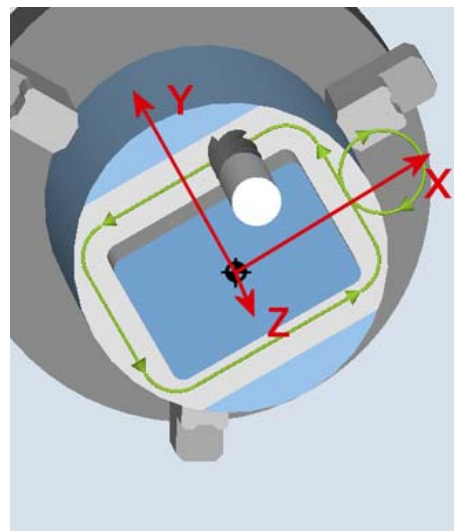
The path movements are conducted with the linear axes X / Z and the rotary axis C.

Machine without Y axis

- Machining with TRANSMIT

Machine with Y axis

- Machining with Y axis
- Machining with TRANSMIT



Highlights



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

13.2 Peripheral surface machining (TRACYL)

- ☑ Option: TRANSMIT and peripheral surface transformation

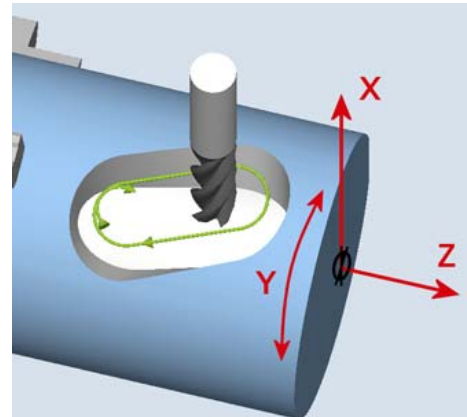
Using the peripheral surface transformation TRACYL, drilling and milling machining can be executed on the peripheral surface of workpieces in the main and counterspindle.

Machine without Y axis

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

Machine with Y axis

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



Highlights



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

Multi-channel machining

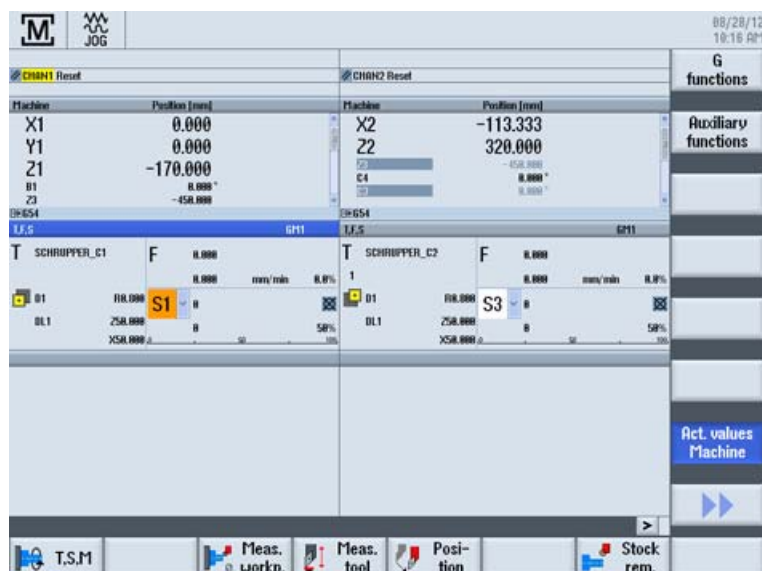
14.1 Overview

Option: programSYNC (multi-channel)

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

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

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



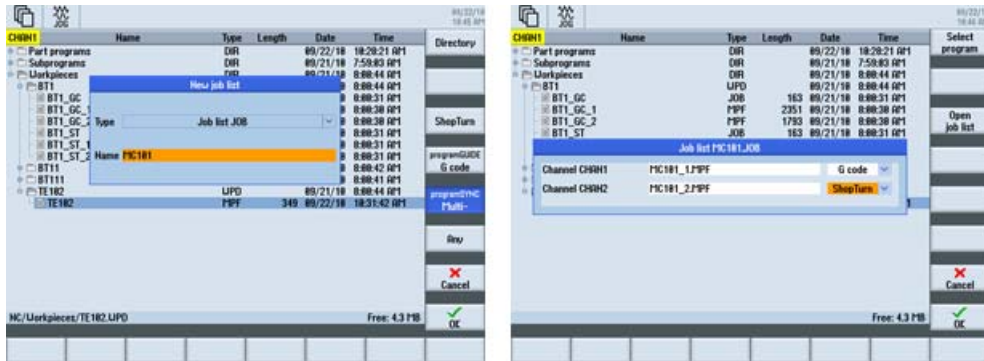
Highlight



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

14.2 programSYNC job list

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



Highlight



- Simple program management in Windows Explorer style.

14.3 Double editor

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

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



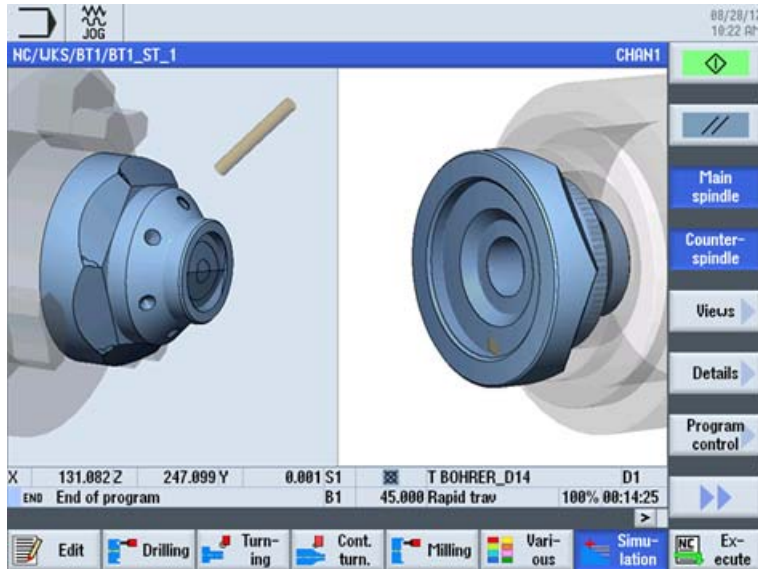
Highlight



- SINUMERIK supports the easy handling of complex machines.

14.4 Simulation

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



Highlight

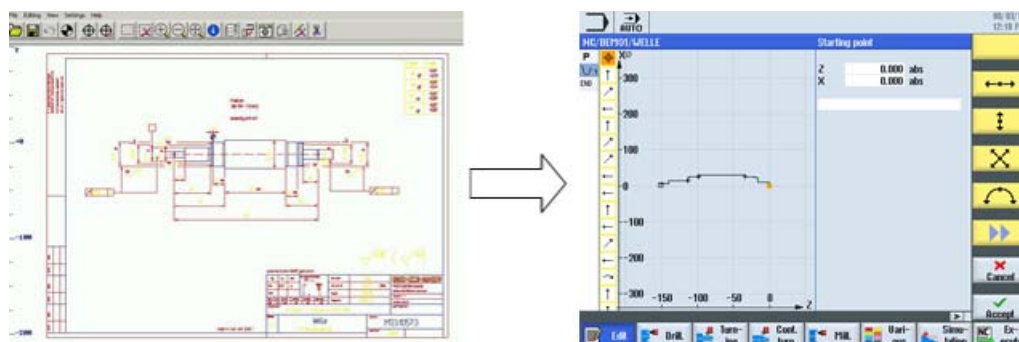


- With the workpiece simulation, SINUMERIK offers optimum help and safety for programming - even during parallel machining.

PC software

15.1 CAD reader for PC

Contours and position patterns can be converted on the PC from DXF files into a format understandable to the controller with the software package "CAD Reader for PC". The contours can be remachined in the contour calculator of the controller.



Highlight



- Save time by converting DXF files into contours and position patterns

15.2 SinuTrain

SinuTrain on your PC behaves in exactly the same way as your SINUMERIK 840D sl on the machine. This allows you to prepare part programs on the PC without having to occupy the machine. In addition, SinuTrain is an ideal training system for CNC training.

- Full functional scope
- Networking of several student and trainer units possible



Highlight



- PC software for training and work preparation without occupying the machine

15.3 Computer-based training

Multi-media initial study of milling technology.

- Programming exercises with guided examples
- Multi-lingualism
- Realistic machine



Highlight



- Graphically supported instruction software for beginners

Integrated safety functions

☑ Option: Safety Integrated

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 according to EU standard EN 954-1 and safety integrated level SIL2 of IEC 61508, ISO 13849-1 and NRTL. As a consequence, the essential requirements relating to functional safety can be simply and cost-effectively implemented.

The following belong to functional safety for machine tools:

- Safety-relevant monitoring of velocity and standstill (zero speed)
- Safety-relevant work and protective zone demarcation
- Direct connection of safety-relevant sensors/actuators
- Internal logical combination of safety-related signals

Highlights



- **High level of flexibility:** With Safety Integrated, setup with an open guard door is possible
- **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:** Reducing the hardware

Option list for the SINUMERIK package

The basic options and their Siemens order numbers are listed in the following:

Programming support

Machining step programming ShopTurn	6FC5800-0AP17-0YB0
Residual material detection and machining for contour pockets and stock removal	6FC5800-0AP13-0YB0
programSYNC (multi-channel)	6FC5800-0AP05-0YB0

Simulation

3D simulation, machined part	6FC5800-0AP20-0YB0
Simultaneous recording (real-time simulation of current machining)	6FC5800-0AP24-0YB0

Transformations

TRANSMIT and peripheral surface transformation	6FC5800-0AM27-0YB0
--	--------------------

Measuring functions/measuring cycles

Measuring cycles for drilling/milling and turning (calibrate workpiece probe, workpiece measurement, tool measurement)	6FC5800-0AP28-0YB0
--	--------------------

User memory

Additional user memory or CompactFlash Card	6FC5800-0AP12-0YB0
Additional 2 MB user memory in the NCU	6FC5800-0AD00-0YB0

Languages

Additional languages for the operating software, without license, e.g. Danish, Finnish, Dutch, Polish, Romanian, Russian, Swedish, Slovakian, Czech, Turkish and Hungarian	6FC5800-0AN00-0YB0
--	--------------------

Diagnostic functions

RCS Host remote diagnostics function	6FC5800-0AP30-0YB0
RCS Commander (viewer function)	
RCS Commander for PC/PG (on CD-ROM)	

Summary of unique features

The operator panel control SINUMERIK 840D sl with SINUMERIK Operate has the following notable features:

Open

- Universal operator interface for milling, turning and additional technologies
- Simple handling of tool and magazine data through clear and powerful tool management
- Wide range of programming options using ShopTurn machining step programming, programGUIDE and DIN/ISO programming

Flexible

- High performance cycles for automatic workpiece and tool measurement
- Full freedom of data transfer via USB, CF card and Ethernet, directly at the operator panel
- Can be used for simple 2-axis turning up to 5-axis multi-channel and simultaneous machining

High

- programSYNC for multi-channel programming permits the user-friendly creation and optimization of the programs for the respective channels
- Free input of blank contours allows more efficient machining thanks to orientation on actually available material
- Powerful transformations for end faces and envelope faces of turned workpieces and inclined machining of turning-milling workpieces

Index

8

80bit NANO FP accuracy, 43

A

Animated Elements, 37

B

Basic block display, 40

Block search, 39

C

CAD Reader, 71

CNC memory, 29

CNC technology cycles

 programGuide, 57

 ShopTurn, 57

CompactFlash Card, 29

Contour stock removal cycle, 58

D

Data transfer

 RCS Commander, 34

Detection of residual material when milling, 62

DIN/ISO language, 49

DIN/ISO programming, 49

DXF converter, 71

E

Engraving cycle, 59

F

Feedforward control, 45

Flashcard, 29

Front end machining, 65

G

G code editor, 48

H

Hard disk, 30

I

In-process measurement, 63

M

Machine spectrum, 10

Measure tool, 22

Monitoring of tool life and workpiece count, 26

Multi-channel machining

 Blocks, 69

 Job list, 68

 Machine basic screen, 67

 programSYNC, 68

 Simulation, 70

 Wait marks, 69

O

Online Help, 38

Operator panel fronts, 12

Operator Panels, 12

Ordering data, 75

P

PC software, 71

Peripheral surface machining, 66

Program control, 41

Program editing, 41

Program editor, 48

Program manager, 33

Program test, 41

R

RCS Commander, 34
Reduced rapid traverse, 41
Replacement tools, 27
Repositioning to the contour, 41
Residual material detection during turning, 61

S

Safety Functions, 73
Safety Integrated, 73
Sequence editor, 51
Simulation
 2D, 55
 3D, 56
 Processing time, 56
Simultaneous recording, 42
Single block, 41
SINUMERIK 840D sl, 11
SinuTrain, 72
Stock removal cycle, 21

T

Tool management, 25
TRACYL, 66
TRANSMIT, 65
TSM universal cycle, 19

U

Unique selling points, 77
User memory, 29

V

Velocity control, 44

W

Work offsets, 24