SINUMERIK

SINUMERIK 828D
SINUMERIK 840D sl
SINUMERIK Operate - Turning

Control system overview
for machine tools' sales people

Valid for:
Controls
SINUMERIK 828D / SINUMERIK 840D sl
Software
CNC software version 4.7

Preface
Introduction
System overview
CNC operation with
SINUMERIK Operate
CNC operation in manual
mode (JOG)
Tool management
Data memory
Data transfer
CNC operation in automatic
mode (AUTO)
CNC functionalities
CNC programming methods
Workpiece visualization
CNC technology cycles
Complete machining
Multi-channel machining
SINUMERIK Integrate Run
MyRobot
Tools and information
Safety functions
Summary of unique features
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 SINUMERIK 828D and SINUMERIK 840D sl with SINUMERIK Operate V4.7 for turning machines.

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

Organization of the information

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

Subject to change without prior notice

Contact person at machine manufacturer

Marketing & Sales

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

Service

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

Homepage:
http://www.machinemanufacturer.com
SINUMERIK user support worldwide contact

<table>
<thead>
<tr>
<th>Name</th>
<th>First name</th>
<th>Region</th>
<th>Email</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fornillo</td>
<td>Pablo</td>
<td>Argentina</td>
<td><a href="mailto:pablo.fornillo@siemens.com">pablo.fornillo@siemens.com</a></td>
</tr>
<tr>
<td>Wallner</td>
<td>Rudolf</td>
<td>Austria, CEE</td>
<td><a href="mailto:rudolf.wallner@siemens.com">rudolf.wallner@siemens.com</a></td>
</tr>
<tr>
<td>Marino</td>
<td>Gustavo</td>
<td>Brazil</td>
<td><a href="mailto:marino.gustavo@siemens.com">marino.gustavo@siemens.com</a></td>
</tr>
<tr>
<td>Da Wei</td>
<td>Hu</td>
<td>China</td>
<td><a href="mailto:dawei.hu@siemens.com">dawei.hu@siemens.com</a></td>
</tr>
<tr>
<td>Klocova</td>
<td>Eva</td>
<td>Czech Republic</td>
<td><a href="mailto:eva.klocova@siemens.com">eva.klocova@siemens.com</a></td>
</tr>
<tr>
<td>Aakesson</td>
<td>Jerker</td>
<td>Denmark, Norway, Sweden</td>
<td><a href="mailto:jerker.akesson@siemens.com">jerker.akesson@siemens.com</a></td>
</tr>
<tr>
<td>Meriaho</td>
<td>Juha</td>
<td>Finland</td>
<td><a href="mailto:juha.meriaho@siemens.com">juha.meriaho@siemens.com</a></td>
</tr>
<tr>
<td>Chevalier</td>
<td>Francois</td>
<td>France</td>
<td><a href="mailto:francois.chevalier@siemens.com">francois.chevalier@siemens.com</a></td>
</tr>
<tr>
<td>Groezinger</td>
<td>Andreas</td>
<td>Germany</td>
<td><a href="mailto:andreas.groezinger@siemens.com">andreas.groezinger@siemens.com</a></td>
</tr>
<tr>
<td>Palla</td>
<td>Csaba</td>
<td>Hungary</td>
<td><a href="mailto:csaba.palla@siemens.com">csaba.palla@siemens.com</a></td>
</tr>
<tr>
<td>Narayanan</td>
<td>Shankar</td>
<td>India</td>
<td><a href="mailto:narayanan.shankar@siemens.com">narayanan.shankar@siemens.com</a></td>
</tr>
<tr>
<td>Arif</td>
<td>Widjyanto</td>
<td>Indonesia</td>
<td><a href="mailto:Arif_Widjyanto@CNCDESIGN.COM.AU">Arif_Widjyanto@CNCDESIGN.COM.AU</a></td>
</tr>
<tr>
<td>Ruggieri</td>
<td>Luigi</td>
<td>Italy</td>
<td><a href="mailto:luigi.ruggieri@siemens.com">luigi.ruggieri@siemens.com</a></td>
</tr>
<tr>
<td>Lee</td>
<td>Byung Chan</td>
<td>Korea</td>
<td><a href="mailto:byungchan.lee@siemens.com">byungchan.lee@siemens.com</a></td>
</tr>
<tr>
<td>Aguado</td>
<td>Mariela</td>
<td>Mexico</td>
<td><a href="mailto:mariela.aguado@siemens.com">mariela.aguado@siemens.com</a></td>
</tr>
<tr>
<td>Korte, de</td>
<td>Frank</td>
<td>The Netherlands</td>
<td><a href="mailto:frank.de.korte@siemens.com">frank.de.korte@siemens.com</a></td>
</tr>
<tr>
<td>Staworko</td>
<td>Karol</td>
<td>Poland</td>
<td><a href="mailto:karol.staworko@siemens.com">karol.staworko@siemens.com</a></td>
</tr>
<tr>
<td>Kudinov</td>
<td>Alexander</td>
<td>Russian Federation</td>
<td><a href="mailto:alexander.kudinov@siemens.com">alexander.kudinov@siemens.com</a></td>
</tr>
<tr>
<td>Lopez Gomez</td>
<td>Javier</td>
<td>Spain</td>
<td><a href="mailto:javier.lopez_gomez@siemens.com">javier.lopez_gomez@siemens.com</a></td>
</tr>
<tr>
<td>Küng</td>
<td>Hans Peter</td>
<td>Switzerland</td>
<td><a href="mailto:hans-peter.kueng@siemens.com">hans-peter.kueng@siemens.com</a></td>
</tr>
<tr>
<td>Chan</td>
<td>Kenny</td>
<td>Taiwan</td>
<td><a href="mailto:chan.kenny@siemens.com">chan.kenny@siemens.com</a></td>
</tr>
<tr>
<td>Dinler</td>
<td>Ozkan</td>
<td>Turkey</td>
<td><a href="mailto:ozkan.dinler@siemens.com">ozkan.dinler@siemens.com</a></td>
</tr>
<tr>
<td>Coombes</td>
<td>Mark</td>
<td>United Kingdom</td>
<td><a href="mailto:mark.coombes@siemens.com">mark.coombes@siemens.com</a></td>
</tr>
<tr>
<td>Neitzke</td>
<td>Wolfgang</td>
<td>USA</td>
<td><a href="mailto:wolfgang.neitzke@siemens.com">wolfgang.neitzke@siemens.com</a></td>
</tr>
<tr>
<td>Tam</td>
<td>Nguyen</td>
<td>Vietnam</td>
<td><a href="mailto:nguyen.tam@siemens.com">nguyen.tam@siemens.com</a></td>
</tr>
</tbody>
</table>

Homepage:
For further information please visit ...

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

# Table of contents

- **Preface** .................................................................................................................. 3

- **Introduction** ........................................................................................................... 9

- **System overview** .................................................................................................... 11
  - 2.1 SINUMERIK 828D .................................................................................................. 11
  - 2.2 SINUMERIK 840D sl .............................................................................................. 14
    - 2.2.1 Panels .............................................................................................................. 15
    - 2.2.2 Operator panel equipment .............................................................................. 19

- **CNC operation with SINUMERIK Operate** ............................................................. 21
  - 3.1 Animated Elements .............................................................................................. 21
  - 3.2 Onboard documentation ...................................................................................... 22
  - 3.3 Multitouch operation ........................................................................................... 23
  - 3.4 Shortcuts .............................................................................................................. 24

- **CNC operation in manual mode (JOG)** .................................................................. 25
  - 4.1 TSM universal cycle ............................................................................................ 25
  - 4.2 Work offsets ......................................................................................................... 26
  - 4.3 Measure workpiece .............................................................................................. 27
  - 4.4 Measure tool ........................................................................................................ 28
  - 4.5 Logging measurement results in JOG ................................................................. 29
  - 4.6 Stock removal cycle ............................................................................................. 30
  - 4.7 Positioning cycle .................................................................................................. 31
  - 4.8 Manual machine .................................................................................................. 32

- **Tool management** .................................................................................................. 33
  - 5.1 Tool table ............................................................................................................. 33
  - 5.2 Monitoring of tool life and workpiece count ...................................................... 34
  - 5.3 Replacement tools .............................................................................................. 35
  - 5.4 Setup data ........................................................................................................... 36

- **Data memory** ......................................................................................................... 37
  - 6.1 Data storage - SINUMERIK 828D ....................................................................... 37
  - 6.2 Data storage - SINUMERIK 840D sl .................................................................. 38

- **Data transfer** ......................................................................................................... 39
  - 7.1 Program manager ............................................................................................... 39
  - 7.2 Ethernet networking ........................................................................................... 40
## Table of contents

### 8 CNC operation in automatic mode (AUTO)

8.1 Block search ................................................................. 41
8.2 Program control ............................................................ 42
8.3 Execution from external storage devices ......................... 43
8.4 Basic block display ....................................................... 44
8.5 Simultaneous recording .................................................. 45
8.6 Logging measurement results in automatic operation ............. 46

### 9 CNC functionalities

9.1 80-bit NANO floating-point accuracy ................................. 47
9.2 Block change times ....................................................... 48
9.2.1 SINUMERIK 828D ..................................................... 48
9.2.2 SINUMERIK 840D sl ................................................. 48
9.3 Jerk limitation ............................................................... 49
9.4 Dynamic feedforward control .......................................... 50

### 10 CNC programming methods

10.1 programGUIDE DIN/ISO and SINUMERIK high-level language ........................................... 52
10.1.1 Introduction ............................................................................................................................ 52
10.1.2 Program editor ....................................................................................................................... 53
10.1.3 Languages ............................................................................................................................. 54
10.1.4 programGUIDE input support .............................................................................................. 55
10.2 ShopTurn machining step programming ............................... 56
10.2.1 Introduction ............................................................................................................................ 56
10.2.2 Sequence editor ..................................................................................................................... 57
10.2.3 Interlinking of sequences ....................................................................................................... 58
10.2.4 Graphical view ...................................................................................................................... 59

### 11 Workpiece visualization

11.1 2D simulation ................................................................................ 61
11.2 3D simulation ................................................................................ 62

### 12 CNC technology cycles

12.1 CNC technology cycles for programGuide and ShopTurn .................................................... 63
12.2 Highlights of machining cycles ............................................................................................ 64
12.2.1 Stock removal along contour with blank contour ................................................................. 64
12.2.2 Engraving cycle ..................................................................................................................... 65
12.2.3 Counterspindle cycle ............................................................................................................ 66
12.3 Residual material detection for contour cycles ......................................................................... 67
12.3.1 Residual material detection during turning ........................................................................... 67
12.3.2 Residual material detection during milling .............................................................................. 68
12.4 In-process measuring for workpieces and tools ........................................................................ 69
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>Complete machining .......................................................................... 71</td>
<td></td>
</tr>
<tr>
<td>13.1</td>
<td>End face machining (TRANSMIT) ...................................................... 71</td>
<td></td>
</tr>
<tr>
<td>13.2</td>
<td>Peripheral surface machining (TRACYL) ......................................... 72</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>Multi-channel machining ................................................................ 73</td>
<td></td>
</tr>
<tr>
<td>14.1</td>
<td>Overview ......................................................................................... 73</td>
<td></td>
</tr>
<tr>
<td>14.2</td>
<td>programSYNC job list ..................................................................... 74</td>
<td></td>
</tr>
<tr>
<td>14.3</td>
<td>Double editor ................................................................................ 75</td>
<td></td>
</tr>
<tr>
<td>14.4</td>
<td>Multi-channel stock removal cycle ................................................ 76</td>
<td></td>
</tr>
<tr>
<td>14.5</td>
<td>Simulation ...................................................................................... 77</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>SINUMERIK Integrate Run MyRobot .................................................. 79</td>
<td></td>
</tr>
<tr>
<td>15.1</td>
<td>SINUMERIK Integrate Run MyRobot / EasyConnect ............................... 79</td>
<td></td>
</tr>
<tr>
<td>15.2</td>
<td>SINUMERIK Integrate Run MyRobot / Handling .................................... 80</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>Tools and information .................................................................... 81</td>
<td></td>
</tr>
<tr>
<td>16.1</td>
<td>DXF reader ...................................................................................... 81</td>
<td></td>
</tr>
<tr>
<td>16.2</td>
<td>SinuTrain for SINUMERIK Operate .................................................. 82</td>
<td></td>
</tr>
<tr>
<td>16.3</td>
<td>CNC4you .......................................................................................... 83</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>Safety functions ............................................................................. 85</td>
<td></td>
</tr>
<tr>
<td>17.1</td>
<td>SINUMERIK Safety Integrated ........................................................... 85</td>
<td></td>
</tr>
<tr>
<td>17.2</td>
<td>Collision avoidance ....................................................................... 86</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>Summary of unique features ............................................................ 87</td>
<td></td>
</tr>
<tr>
<td>18.1</td>
<td>SINUMERIK 828D ............................................................................... 87</td>
<td></td>
</tr>
<tr>
<td>18.2</td>
<td>SINUMERIK 840D sl. ......................................................................... 88</td>
<td></td>
</tr>
<tr>
<td>Index</td>
<td>........................................................................................................... 89</td>
<td></td>
</tr>
</tbody>
</table>
Introduction

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

SINUMERIK Operate

Characteristic features of SINUMERIK Operate include…

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

Two options are available for the programming:

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

programSYNC – Efficient programming for multi-channel machines

Turning-milling centers with several saddles are considered to be the high-end machines in this segment. The individual tool carriers are distributed and managed by SINUMERIK in different channels. Programs must be created for each channel that then run simultaneously later during the machining. With the uniform SINUMERIK Operate user interface, Siemens provides a standard user interface which enables programs for two channels to be created simultaneously by means of a double editor and to align them with the programSYNC function. In this way, efficient programming is possible directly on the control.
2.1 SINUMERIK 828D

Alongside three high-performance CNC variants of SINUMERIK 828D, SINUMERIK 828D BASIC is a low-cost starter model in the compact class. SINUMERIK 828 therefore fits the performance requirements of standard machine concepts perfectly.

- Panel-based compact CNC
- Technologies: Turning, milling, G-Tech
- Up to 10 axes/spindles and 2 help axes
- Up to 2 machining channels
- 10.4" color display / 15.6" touch display
- SIMATIC S7-200 PLC-based

You can find further information in catalog NC 82

Benefits

- Maximum robustness and low maintenance
- Reliable CNC, even in a harsh environment
- 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 SINUMERIK 828D

SINUMERIK 828D BASIC PPU 24x.3

The SINUMERIK 828D BASIC is an operator-panel CNC that combines all the components of a CNC in a single unit:

- CNC, PLC, HMI
- 10.4" TFT color display
- Full CNC keyboard
- 1 machining channel
- Closed-loop control for up to 5 axes/spindles

SINUMERIK 828D BASIC PPU 240.3, horizontal

SINUMERIK 828D PPU 28x.3

The SINUMERIK 828D PPU 28x.3 is an operator-panel CNC that combines all the components of a CNC in a single unit:

- CNC, PLC, HMI
- 10.4" TFT color display
- Full CNC keyboard
- 1 machining channel
- Closed-loop control for up to 8 axes/spindles

Note: Use with 26x or 28x software variant

SINUMERIK 828D PPU 280.3, horizontal
SINUMERIK 828D ADVANCED PPU 290.3

The SINUMERIK 828D ADVANCED PPU 290.3 is an operator-panel CNC that combines all the components of a CNC in a single unit:

- CNC, PLC, HMI
- 15.6” multitouch display in 16:9 format
- Full CNC keyboard
- Up to 2 machining channels
- Closed-loop control for up to 10 axes/spindles, depending on the software used

Note: Use with 26x, 28x or 28xA software variant

SINUMERIK 828D PPU 290.3
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 Industry 4.0 automation concepts
2.2.1 Panels

SINUMERIK OP 08T

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

SINUMERIK OP 010

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

SINUMERIK OP 010S

- Operator panel 310 mm wide, 10.4" TFT display (resolution 640 × 480 pixels)
- Mechanical keys
- With USB interface for a memory stick at the front
- Separate CNC keyboard and machine control panel
2.2 SINUMERIK 840D sl

**OP 010C**

- Operator panel 483 mm wide, 10.4" TFT display (resolution 640 x 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 x 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 x 768 pixels)
- Version with membrane keyboard with 62 keys
- With USB interface at the front
- Integrated mouse
SINUMERIK OP 015 black

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

**Note:** see also Chapter AUTOHOTSPOT

SINUMERIK OP 019

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

SINUMERIK OP 019 black

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

**Note:** see also Chapter AUTOHOTSPOT
SIMATIC Industrial Thin Client

- Touch operation
- Connection via Ethernet

**Versions:**
- SIMATIC ITC 1200, 12" widescreen TFT display (resolution 1280 x 800 pixels)
- SIMATIC ITC 1500, 15" widescreen TFT display (resolution 1280 x 800 pixels)
- SIMATIC ITC 1900, 19" widescreen TFT display (resolution 1366 x 768 pixels)

**Note:** see also Chapter AUTOHOTSPOT
2.2.2 Operator panel equipment

PCU 50

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

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

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

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

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

Benefits

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

<table>
<thead>
<tr>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
</tr>
</thead>
<tbody>
<tr>
<td>BASIC</td>
<td>BASIC</td>
<td>ADVANCED</td>
</tr>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>BASIC</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

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

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 290.3, 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:

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

Benefit

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

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

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

**Benefit**

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

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D ADVANCED</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl BASIC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

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

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

**Benefit**

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

The following work offsets are possible:

- **Settable work offsets:**
  
  It is possible to enter as many as 100 work offsets (G54 to G57, G505 to G599), offset coordinates, angles and scaling factors.

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

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

**Benefits**

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

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D ADVANCED</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

The workpieces can be measured as follows:

- Reference tool

Benefit

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

<table>
<thead>
<tr>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
</tr>
</thead>
<tbody>
<tr>
<td>BASIC</td>
<td>BASIC</td>
<td>ADVANCED</td>
</tr>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>BASIC</td>
<td></td>
</tr>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

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

The following variants are supported:

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

The measurement results can be output in a measurement report (see Logging measurement results in JOG (Page 29)).

**Benefit**

- User-friendly functions for determining the tool dimensions directly in the machine
4.5 Logging measurement results in JOG

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D ADVANCED</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

The results for measuring in JOG can be logged. The standard log contains the measurement results of the most recently performed measuring method. The function is available as turning technology for the tool measurement.

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

The measuring log comprises the following data:

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

**Benefit**

- Simple logging of measured values in log files
4.6 Stock removal cycle

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

The following parameters can be specified:

- Roughing or finishing
- Undercut for soft collet chucks

**Benefit**

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

<table>
<thead>
<tr>
<th></th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>SINUMERIK 840D sl</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td></td>
</tr>
</tbody>
</table>

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

- Linear axes / spindles
- Feedrate / rapid traverse

Benefit

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

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
- Setting the endstop
- Turning a straight line / circle
- Drilling, including centering, deep-hole drilling, tapping
- Milling, including face milling, pocket, multiple edge spigot

Benefit

- Simple and intuitive operation of cycle-controlled turning machines
5.1 Tool table

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D ADVANCED</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl BASIC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

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 (routher, 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
- 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

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

Benefits

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

<table>
<thead>
<tr>
<th></th>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D ADVANCED</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option</td>
<td>M78</td>
<td>M78</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

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

Benefit

- Automatic exchange of identical tools for unmanned operation
5.4 Setup data

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

Benefit

- Time savings when you save the part programs
## Data memory

### 6.1 Data storage - SINUMERIK 828D

<table>
<thead>
<tr>
<th>Internal memory</th>
<th>External storage</th>
</tr>
</thead>
<tbody>
<tr>
<td>828D BASIC</td>
<td>Execution from external storage (EES) (option P75) Network, USB storage media, compact flashcard</td>
</tr>
<tr>
<td>828D</td>
<td></td>
</tr>
<tr>
<td>828D ADVANCED</td>
<td></td>
</tr>
<tr>
<td>USB / CF card can be used for data transport or for execution with EXTCALL</td>
<td></td>
</tr>
<tr>
<td>Option P77</td>
<td>100 MB</td>
</tr>
<tr>
<td>SW</td>
<td></td>
</tr>
<tr>
<td>3 MB</td>
<td></td>
</tr>
<tr>
<td>5 - 10 MB</td>
<td></td>
</tr>
<tr>
<td>Internal memory can be expanded via option P77 → 100 MB</td>
<td>External storage via option P75 → can be expanded almost without limit</td>
</tr>
</tbody>
</table>
### 6.2 Data storage - SINUMERIK 840D sl

<table>
<thead>
<tr>
<th>Internal memory</th>
<th>NCU</th>
<th>NCU</th>
<th>NCU</th>
<th>NCU + PCU</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option P77 + PCU</td>
<td></td>
<td></td>
<td></td>
<td>up to 40 GB</td>
</tr>
<tr>
<td>Option P77 + option P12 ¹</td>
<td></td>
<td></td>
<td></td>
<td>up to 6 GB</td>
</tr>
<tr>
<td>Option P77</td>
<td></td>
<td></td>
<td></td>
<td>100 MB</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>External storage</th>
<th>Execution from external storage devices (EES, option P75)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Network, USB storage media, compact flashcard</td>
</tr>
<tr>
<td></td>
<td>Execution from the CNC expanded user memory (option P77)</td>
</tr>
<tr>
<td></td>
<td>External storage via option P75 → can be expanded almost without limit</td>
</tr>
</tbody>
</table>

| CNC user memory (option D00) | NCU 710.3B: 2 to 16 MB  
| NCU 720.3B and NCU 730.3B: 2 to 22 MB |

1) HMI user memory, alternative to PCU
7

Data transfer

7.1 Program manager

<table>
<thead>
<tr>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
</tr>
</thead>
<tbody>
<tr>
<td>BASIC</td>
<td>BASIC</td>
<td>ADVANCED</td>
</tr>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>BASIC</td>
<td>BASIC</td>
</tr>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

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

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

Benefits

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

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D ADVANCED</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option: P01</td>
<td>Option: P01</td>
<td>Option: P01</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option: P17</td>
<td>Option: P17</td>
</tr>
</tbody>
</table>

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

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

Benefits

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

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

The following search variants are available:

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

You can individually configure the block search:

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

Benefits

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

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

- **PRT** – No axis motion
  The program is completely executed with the axes stationary, e.g. for the program test.

- **DRY** – Dry run feedrate
  The traversing velocities programmed in conjunction with G1, G2, G3, CIP and CT are replaced by a defined dry run feedrate.

- **RG0** – Reduced rapid traverse
  You define the reduced rapid traverse in the settings for automatic operation.

- **M01** – Programmed stop 1
  The processing of the program stops at every block in which supplementary function M01 is programmed. In this way you can check the intermediate result when machining a workpiece.

- **SKP** – Skip block
  Skip blocks are skipped during machining.

- **MRD** – Displaying measurement result
  The display of the measurement result can be enabled or disabled during program execution.

**Benefits**

- Secure positioning of new part programs
- Continue machining quickly after interruptions
### 8.3 Execution from external storage devices

<table>
<thead>
<tr>
<th>Option</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>BASIC</td>
<td>BASIC</td>
<td>BASIC</td>
</tr>
<tr>
<td></td>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>Option</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>BASIC</td>
<td>BASIC</td>
<td>ADVANCED</td>
</tr>
<tr>
<td></td>
<td>Option: P75</td>
<td>Option: P75</td>
<td>Option: P75</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Option</th>
<th>SINUMERIK 840D sl</th>
<th>SINUMERIK 840D sl</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>BASIC</td>
<td>BASIC</td>
<td>BASIC</td>
</tr>
<tr>
<td></td>
<td>Option: P75</td>
<td>Option: P75</td>
<td>Option: P75</td>
</tr>
</tbody>
</table>

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
- Block search for large programs on external storage media
### 8.4 Basic block display

![Basic block display](image)

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

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

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

**Benefit**

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

<table>
<thead>
<tr>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
</tr>
</thead>
<tbody>
<tr>
<td>BASIC</td>
<td>ADVANCED</td>
<td></td>
</tr>
<tr>
<td>Option: P22</td>
<td>Option: P22</td>
<td>Option: P22</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl</th>
<th>SINUMERIK 840D sl</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>BASIC</td>
<td>Option: P22</td>
<td></td>
</tr>
<tr>
<td>Option: P22</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

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

Benefit

- Machining can also be monitored in a complex machine room
8.6 Logging measurement results in automatic operation

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
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 based on extremely high calculation accuracy
9.2 Block change times

9.2.1 SINUMERIK 828D

<table>
<thead>
<tr>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
</tr>
</thead>
<tbody>
<tr>
<td>BASIC</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
<tr>
<td>ADVANCED</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>PPU 241.3/240.3</th>
<th>PPU 281.3/280.3</th>
<th>PPU 290.3</th>
</tr>
</thead>
<tbody>
<tr>
<td>SW24</td>
<td>SW26</td>
<td>SW28</td>
</tr>
<tr>
<td>~3 ms</td>
<td>~2 ms</td>
<td>~1 ms</td>
</tr>
</tbody>
</table>

Benefit

- Minimum block change times for the associated performance versions

9.2.2 SINUMERIK 840D sl

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>BASIC</td>
<td>Basic configuration</td>
</tr>
<tr>
<td></td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>NCU 710.3B PN</th>
<th>NCU 720.3B PN</th>
<th>NCU 730.3B PN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.2 ms</td>
<td>0.5 ms</td>
<td>0.3 ms</td>
</tr>
</tbody>
</table>

Benefit

- Minimum block change times for the associated performance versions
9.3 Jerk limitation

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
9.4 Dynamic feedforward control

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:

![Diagram showing comparison between Without pre-control and With pre-control](image)

**Benefit**

- Higher path accuracy through compensation of contouring errors
CNC programming methods

SINUMERIK Operate provides the following programming methods for selection:

**DIN-ISO programming with programGUIDE**
CNC text editor with programGuide cycle support, and DIN-ISO and readable CNC high-level language commands for mid-sized and large series
The wide choice of technology cycles and the ease of parameterization allows you to reduce the programming time.

**ShopTurn - Machining step programming**
with graphical interactive CNC machining step editor and CNC programming without DIN-ISO knowledge for small series.
Machining operations such as stock removal, grooving or thread cutting are shown in ShopTurn in the form of worksteps. In this way CNC programs – even for complex machining operations – are very compact and easily read. Associated sequences are automatically interlinked and can be assigned any position patterns.
ShopTurn offers you the shortest programming times even for highly demanding machining tasks. The parameter input is supported by Animated Elements.

**Benefit**

- Whether you use programGUIDE or ShopTurn – in either case the full range of technological cycles, position patterns and geometries is available to you
10.1 programGUIDE DIN/ISO and SINUMERIK high-level language

10.1.1 Introduction

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 are available for the most complex machining. The following functions are included in the program editor:

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

Benefits

- Time saving by using a powerful editor when programming
- Even large part programs allow extremely fast editing in MB size
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 ShopTurn cycles input screens, so as to ensure optimum continuity.

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

Benefits

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

10.2.1 Introduction

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

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

These functions are part of the machining step programming options package in ShopTurn.
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 turning, centering, drilling, tapping) or geometric data required for the sequences (position patterns or contours). Graphical programming offers, in comparison to DIN/ISO programming, a compact and comprehensible program overview.

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

Benefits

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

In ShopTurn, associated sequences are interlinked with each other. The interlinked sequences are performed consecutively at the appropriate contours or pattern positions. In the following example, the sequences centering, drilling and tapping are applied to 4 holes on the pitch circle pattern position.

Benefit

- Reduced programming time due to linking of machining steps
10.2.4 **Graphical view**

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

- Turning view
- Front face and peripheral side

**Benefit**

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

10.2 ShopTurn machining step programming
11.1 2D simulation

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

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

**Benefits**

- Maximum process reliability through simulation using real geometry values
- Perfect clarity by showing the workpiece dimensions with a scale
- Parallel simulation (background simulation) is possible in conjunction with the NCU 720 and NCU 730, i.e. simulating a part program while another part program is being simultaneously machined.
11.2 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
  - 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.
12.1 CNC technology cycles for programGuide and ShopTurn

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

Benefits

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

12.2.1 Stock removal along contour with blank contour

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

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

Benefits

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

12.2.2 Engraving cycle

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 Highlights of machining cycles

12.2.3 Counterspindle cycle

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D ADVANCED</th>
</tr>
</thead>
<tbody>
<tr>
<td>not available</td>
<td>Option: M75 (SW 28x)</td>
<td>Option: M75</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration 1)</td>
<td>Basic configuration 1)</td>
</tr>
</tbody>
</table>

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

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

DIN/ISO programming

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

Machining step programming

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

Benefits

- Simple and secure programming of all counterspindle functions
- High quality of workpieces by workpiece transfer in synchronous spindle mode
12.3 Residual material detection for contour cycles

12.3.1 Residual material detection during turning

<table>
<thead>
<tr>
<th>Option: P13</th>
<th>Option: P13</th>
<th>Option: P13</th>
</tr>
</thead>
<tbody>
<tr>
<td>SINUMERIK 828D BASIC</td>
<td>SINUMERIK 828D</td>
<td>SINUMERIK 828D ADVANCED</td>
</tr>
<tr>
<td>Option: P13</td>
<td>Option: P13</td>
<td></td>
</tr>
<tr>
<td>Option: P13</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

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

**Benefit**

- Time saving through avoiding idle cuts during residual stock removal
12.3.2 Residual material detection during milling

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D ADVANCED</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option: P13</td>
<td>Option: P13</td>
<td>Option: P13</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option: P13</td>
<td>Option: P13</td>
</tr>
</tbody>
</table>

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

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

Benefits

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

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

The following cycles are available for workpiece measurement:

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

The following measuring tasks can be made:

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

Benefits

- Reliable quality of the manufactured parts by automatic measurement in the machine
- Fast programming for complex measuring tasks thanks to input screens with graphic support
12.4 In-process measuring for workpieces and tools
13.1 End face machining (TRANSMIT)

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

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

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

Machine without Y axis
- Machining with TRANSMIT

Machine with Y axis
- Machining with Y axis
- Machining with TRANSMIT

Benefit
- Full functional range for drilling and milling on the end face
13.2 Peripheral surface machining (TRACYL)

**Benefits**

- Full functional range for drilling and milling on the peripheral surface
- Reduction of set-up times by complete machining on one machine
14

Multi-channel machining

14.1 Overview

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D ADVANCED</th>
</tr>
</thead>
<tbody>
<tr>
<td>not available</td>
<td>not available</td>
<td>Option: P05</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl BASIC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option: M27</td>
<td>Option: M27</td>
</tr>
</tbody>
</table>

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

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

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

Benefit

- SINUMERIK supports the easy handling of complex machines.
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.

**Benefit**

- 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.

**Benefit**

- Easy creation of time-optimized programs through synchronization of wait marks and determination of the processing time of the respective blocks
14.4 Multi-channel stock removal cycle

With the intelligent multi-channel stock removal cycle CYCLE952, contours can be machined in a variety of ways. Powerful functions facilitate the workpiece machining:

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

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

Benefit

- Cycle support enables efficient programming of complex machining tasks for multi-channel turning machines
14.5 Simulation

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

Benefit

- With the workpiece simulation, SINUMERIK offers optimum help and safety for programming - even during parallel machining
15.1 SINUMERIK Integrate Run MyRobot / EasyConnect

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D ADVANCED</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

The prepared Run MyRobot / EasyConnect configuring interface permits the connection of handling robots to machine tools.

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

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

**Benefit**

- The prepared Run MyRobot / EasyConnect configuration interface provides a universal and manufacturer-independent interface for the low-effort automation of machine tools.
15.2 SINUMERIK Integrate Run MyRobot / Handling

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.
16

Tools and information

16.1 DXF reader

<table>
<thead>
<tr>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D</th>
</tr>
</thead>
<tbody>
<tr>
<td>BASIC</td>
<td>BASIC</td>
<td>ADVANCED</td>
</tr>
<tr>
<td>Option: P56</td>
<td>Option: P56</td>
<td>Option: P56</td>
</tr>
</tbody>
</table>

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

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](https://www.siemens.com/sinutrain)

### Benefits

- Controller-identical PC software for training and work preparation with configuration of the real machine on the PC
- Preparation of the part program anywhere without needing a machine
- Prediction of the production time
16.3 CNC4you

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

CNC4you portal:
http://www.siemens.de/cnc4you
Safety functions

17.1 SINUMERIK Safety Integrated

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D ADVANCED</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option: refer to the catalog</td>
<td>Option: refer to the catalog</td>
<td>Option: refer to the catalog</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option: refer to the catalog</td>
<td>Option: refer to the catalog</td>
</tr>
</tbody>
</table>

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 the safe monitoring of speed 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

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D ADVANCED</th>
</tr>
</thead>
<tbody>
<tr>
<td>not available</td>
<td>not available</td>
<td>not available</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option: S02</td>
<td>Option: S02</td>
</tr>
</tbody>
</table>

The SINUMERIK Collision Avoidance option provides optimum protection against unintentional collisions of moving machine components with stationary machine components.

You can active the collision monitoring in the Machine operating area for the JOG, MDA and Automatic operating modes.

The Automatic operating mode provides for logging a 3D visualization of the collision monitoring.

---

Note

The use of collision monitoring requires the availability of the relevant machine data and the associated visualization.

---

Benefit

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

18.1 SINUMERIK 828D

The SINUMERIK 828D operator panel controller has the following notable features:

Compact

- Robust and maintenance-free design
- All relevant functions at a glance on the 10.4” color screen
- Full-function QWERTY CNC keyboard for user-friendly programming at the machine
- Full freedom of data transfer via USB, CF card and Ethernet, directly at the operator panel

High

- 80-bit NANO floating-point accuracy for maximum precision of the workpiece results
- 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 oblique machining of milling workpieces
- Simple handling of tool and magazine data through clear and powerful tool management

Simple

- Animated Elements: unique facility to display machining parameters with animated sequences
- ShopTurn machining step programming: shortest programming times and clear CNC programs with technological sequences
- Common user interface for milling and turning
- Easy Message: simple process monitoring by SMS
18.2 SINUMERIK 840D sl

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
80-bit NANO floating-point accuracy, 47

I
In-process measurement, 69

A
Animated Elements, 21

M
Measure tool, 28
Monitoring of tool life and workpiece count, 34
Multi-channel machining
  Blocks, 75
  Job list, 74
  Machine basic screen, 73
  programSYNC, 74
  Simulation, 77
  Wait marks, 75

B
Basic block display, 44
Block change times, (SINUMERIK 828D), (SINUMERIK 840D sl)
Block search, 41

O
Online Help, 22

C
CNC technology cycles
  programGuide, 63
  ShopTurn, 63
Collision monitoring, 86
Contour stock removal cycle, 64
Contour-grooving, 64

P
Peripheral surface machining, 72
Plunge-turning, 64
Positioning cycle, 31
Program editor, 53
Program manager, 39

D
Detection of residual material when milling, 68
DIN/ISO language, 54
DIN/ISO programming, 54

R
Replacement tools, 35
Residual material detection during turning, 67

E
End face machining, 71
Engraving cycle, 65

S
Safety functions, 85
Safety Integrated, 85
Sequence editor, 57
SIMATIC ITC 1200, 18
SIMATIC ITC 1500, 18
SIMATIC ITC 1900, 18
Simulation
  2D, 61

F
Feed interruption, 64
Feedforward control, 50

G
G code editor, 53
3D, 62
  Processing time, 62
Simultaneous recording, 45
SINUMERIK CNC4you, 83
SINUMERIK OP 010, 15
SINUMERIK OP 010C, 16
SINUMERIK OP 010S, 15
SINUMERIK OP 012, 16
SINUMERIK OP 015 black, 17
SINUMERIK OP 019, 17
SINUMERIK OP 019 black, 17
SINUMERIK OP 08T, 15
SinuTrain, 82
  Speed control, 49
Stock removal cycle, 30

T
  Tool management, 33
  TRACYL, 72
  TRANSMIT, 71
  TSM universal cycle, 25

U
  Unique selling points, 87, 88