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 functions

Tool and mold making

CNC programming methods

Workpiece visualization

CNC technology cycles

Complete machining
  SINUMERIK Integrate Run MyRobot

Tools and information

Safety functions

Summary of unique selling points

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

2017
A5E41992599B AA
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 the SINUMERIK 840D sl with SINUMERIK Operate V4.7 for milling machines.
The document is focusing on vendors and dealers of machine tools.

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

Subject to change without prior notice

Contact person at machine manufacturer
Marketing & Sales
Phone: +49 xxx xxx
Fax: +49 xxx xxx
E-mail: xxx@machinemanufacturer.com

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

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

<table>
<thead>
<tr>
<th>Name</th>
<th>First name</th>
<th>Region</th>
<th>E-mail</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>Vetter</td>
<td>Philipp</td>
<td>China</td>
<td><a href="mailto:philipp.vetter@siemens.com">philipp.vetter@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>Gonda</td>
<td>Norbert</td>
<td>Hungary</td>
<td><a href="mailto:norbert.gonda@siemens.com">norbert.gonda@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>Widjayanto</td>
<td>Indonesia</td>
<td><a href="mailto:Arif_Widjayanto@CNCDESIGN.COM.AU">Arif_Widjayanto@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>Diner</td>
<td>Ozkan</td>
<td>Turkey</td>
<td><a href="mailto:ozkan.diner@siemens.com">ozkan.diner@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

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Preface</td>
<td></td>
<td>3</td>
</tr>
<tr>
<td>1</td>
<td>Introduction</td>
<td>9</td>
</tr>
<tr>
<td>2</td>
<td>System overview</td>
<td>11</td>
</tr>
<tr>
<td>2.1</td>
<td>SINUMERIK 828D</td>
<td>11</td>
</tr>
<tr>
<td>2.1.1</td>
<td>Handheld unit</td>
<td>15</td>
</tr>
<tr>
<td>2.2</td>
<td>SINUMERIK 840D sl</td>
<td>16</td>
</tr>
<tr>
<td>2.2.1</td>
<td>Panels</td>
<td>17</td>
</tr>
<tr>
<td>2.2.2</td>
<td>Operator panel equipment</td>
<td>21</td>
</tr>
<tr>
<td>2.2.3</td>
<td>Handheld units</td>
<td>23</td>
</tr>
<tr>
<td>3</td>
<td>CNC operation with SINUMERIK Operate</td>
<td>25</td>
</tr>
<tr>
<td>3.1</td>
<td>Animated elements</td>
<td>25</td>
</tr>
<tr>
<td>3.2</td>
<td>Onboard documentation</td>
<td>26</td>
</tr>
<tr>
<td>3.3</td>
<td>Multitouch operation</td>
<td>27</td>
</tr>
<tr>
<td>3.4</td>
<td>Shortcuts</td>
<td>28</td>
</tr>
<tr>
<td>4</td>
<td>CNC operation in manual mode (J OG)</td>
<td>29</td>
</tr>
<tr>
<td>4.1</td>
<td>TSM universal cycle</td>
<td>29</td>
</tr>
<tr>
<td>4.2</td>
<td>Work offsets</td>
<td>30</td>
</tr>
<tr>
<td>4.3</td>
<td>Measuring a workpiece</td>
<td>31</td>
</tr>
<tr>
<td>4.4</td>
<td>Measuring a tool</td>
<td>32</td>
</tr>
<tr>
<td>4.5</td>
<td>Logging measurement results in J OG</td>
<td>33</td>
</tr>
<tr>
<td>4.6</td>
<td>Face milling cycle</td>
<td>34</td>
</tr>
<tr>
<td>4.7</td>
<td>Retract</td>
<td>35</td>
</tr>
<tr>
<td>4.8</td>
<td>Stop and retract (ESR)</td>
<td>36</td>
</tr>
<tr>
<td>4.8.1</td>
<td>Stop and retract (ESR) - Drive-autonomous</td>
<td>36</td>
</tr>
<tr>
<td>4.8.2</td>
<td>Stop and retract (ESR) - CNC-controlled and drive-autonomous</td>
<td>37</td>
</tr>
<tr>
<td>4.9</td>
<td>Swiveling in setup mode</td>
<td>38</td>
</tr>
<tr>
<td>4.10</td>
<td>Manual machine</td>
<td>39</td>
</tr>
<tr>
<td>5</td>
<td>Tool management</td>
<td>41</td>
</tr>
<tr>
<td>5.1</td>
<td>Tool table</td>
<td>41</td>
</tr>
<tr>
<td>5.2</td>
<td>Monitoring of tool life and workpiece count</td>
<td>42</td>
</tr>
<tr>
<td>5.3</td>
<td>Replacement tools</td>
<td>43</td>
</tr>
</tbody>
</table>
# Table of contents

6  Data memory .............................................................................................................................. 45

6.1  Data storage - SINUMERIK 828D .......................................................................................45
6.2  Data storage - SINUMERIK 840D sl ...................................................................................46

7  Data transfer .............................................................................................................................. 47
7.1  Program Manager ..............................................................................................................47
7.2  Ethernet networking ...........................................................................................................48

8  CNC operation in automatic mode (AUTO) ..................................................................................... 49
8.1  Block search ......................................................................................................................49
8.2  Program control ..................................................................................................................50
8.3  Execution from external storage devices .............................................................................51
8.4  Basic block display .............................................................................................................52
8.5  Simultaneous recording ......................................................................................................53
8.6  Logging measurement results in automatic operation .........................................................54
8.7  Handwheel override ...........................................................................................................55

9  CNC functions ............................................................................................................................ 57
9.1  80-bit NANO floating-point accuracy ...................................................................................57
9.2  Block change times ............................................................................................................58
9.2.1  SINUMERIK 828D ..............................................................................................................58
9.2.2  SINUMERIK 840D sl ..........................................................................................................58
9.3  Jerk limiting ........................................................................................................................59
9.4  Dynamic feedforward control ..............................................................................................60

10  Tool and mold making ................................................................................................................. 61
10.1  High speed settings ............................................................................................................61
10.2  Advanced Surface and Top Surface ...................................................................................62
10.2.1  Advanced Surface ..............................................................................................................63
10.2.2  Top Surface .......................................................................................................................64
10.3  Look Ahead ........................................................................................................................66

11  CNC programming methods ......................................................................................................... 67
11.1  programGUIDE DIN/ISO and SINUMERIK high-level language ........................................68
11.1.1  Introduction ........................................................................................................................68
11.1.2  Program editor ...................................................................................................................69
11.1.3  Languages ........................................................................................................................70
11.1.4  programGUIDE input support .............................................................................................71
11.2  ShopMill - machining step programming .............................................................................72
11.2.1  Introduction ........................................................................................................................72
11.2.2  Sequence editor .................................................................................................................73
11.2.3  Interlinking of sequences ....................................................................................................74
11.2.4  Graphic view ......................................................................................................................75
<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>12</td>
<td>Workpiece visualization</td>
</tr>
<tr>
<td>12.1</td>
<td>2D simulation</td>
</tr>
<tr>
<td>12.2</td>
<td>3D simulation</td>
</tr>
<tr>
<td>12.3</td>
<td>Mold making fast view</td>
</tr>
<tr>
<td>13</td>
<td>CNC technology cycles</td>
</tr>
<tr>
<td>13.1</td>
<td>CNC technology cycles for programGUIDE and ShopMill</td>
</tr>
<tr>
<td>13.2</td>
<td>Highlights of machining cycles</td>
</tr>
<tr>
<td>13.2.1</td>
<td>Overview</td>
</tr>
<tr>
<td>13.2.2</td>
<td>Engraving cycle</td>
</tr>
<tr>
<td>13.2.3</td>
<td>Trochoidal milling</td>
</tr>
<tr>
<td>13.2.4</td>
<td>Plunge milling</td>
</tr>
<tr>
<td>13.2.5</td>
<td>Deep-hole drilling</td>
</tr>
<tr>
<td>13.3</td>
<td>Residual material detection for contour cycles</td>
</tr>
<tr>
<td>13.4</td>
<td>In-process measuring for workpiece and tool</td>
</tr>
<tr>
<td>13.5</td>
<td>Measure multiple axis kinematics</td>
</tr>
<tr>
<td>14</td>
<td>Complete machining</td>
</tr>
<tr>
<td>14.1</td>
<td>Cylinder surface transformation (TRACYL)</td>
</tr>
<tr>
<td>14.2</td>
<td>Swivel plane (CYCLE800)</td>
</tr>
<tr>
<td>14.3</td>
<td>5-axis machining package (TRAORI)</td>
</tr>
<tr>
<td>14.4</td>
<td>Milling-turning</td>
</tr>
<tr>
<td>14.4.1</td>
<td>Introduction</td>
</tr>
<tr>
<td>14.4.2</td>
<td>Tool management</td>
</tr>
<tr>
<td>14.4.3</td>
<td>Programming</td>
</tr>
<tr>
<td>14.4.4</td>
<td>Simulation</td>
</tr>
<tr>
<td>15</td>
<td>SINUMERIK Integrate Run MyRobot</td>
</tr>
<tr>
<td>15.1</td>
<td>SINUMERIK Integrate Run MyRobot / EasyConnect</td>
</tr>
<tr>
<td>15.2</td>
<td>SINUMERIK Integrate Run MyRobot / Handling</td>
</tr>
<tr>
<td>16</td>
<td>Tools and information</td>
</tr>
<tr>
<td>16.1</td>
<td>DXF reader</td>
</tr>
<tr>
<td>16.2</td>
<td>SinuTrain for SINUMERIK Operate</td>
</tr>
<tr>
<td>16.3</td>
<td>CNC4you</td>
</tr>
<tr>
<td>17</td>
<td>Safety functions</td>
</tr>
<tr>
<td>17.1</td>
<td>SINUMERIK Safety Integrated</td>
</tr>
<tr>
<td>17.2</td>
<td>Collision avoidance</td>
</tr>
<tr>
<td>18</td>
<td>Summary of unique selling points</td>
</tr>
<tr>
<td>18.1</td>
<td>SINUMERIK 828D</td>
</tr>
<tr>
<td>18.2</td>
<td>SINUMERIK 840D sl</td>
</tr>
<tr>
<td>Index</td>
<td>Index</td>
</tr>
</tbody>
</table>
Introduction

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

SINUMERIK Operate

Characteristic features of SINUMERIK Operate include...

- HMI-Advanced, ShopMill and ShopTurn are 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
- ShopMill machining step programming with graphical interactive CNC machining step editor and CNC programming without DIN-ISO knowledge for small series

SINUMERIK MDynamics

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

- Powerful CNC hardware and intelligent CNC functions for a cost-effective package price
- Very simple to operate
- NX CAD/CAM and SINUMERIK MDynamics provide an integrated, optimally matched CAD/CAM/CNC process chain
- Technology know-how concerning milling in all industry sectors, e.g. automotive, aerospace or workshop manufacturing
SINUMERIK 828D and SINUMERIK 840D sl with SINUMERIK MDynamics so offer a control configuration that covers all required application areas for using the milling technology without subsequent commissioning effort:

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

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
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 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 6 axes/spindles

**Note:** Deployment with SW26 or SW28

SINUMERIK 828D PPU 280.3, vertical
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 8 axes/spindles, depending on the deployed software

Note: Deployment with SW26, SW28 or SW28A

SINUMERIK 828D PPU 290.3, vertical
2.1.1 Handheld unit

Mini handheld unit

You can install the mini handheld unit pictured below in setup mode.
2.2 SINUMERIK 840D sl

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

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

You can find further information in catalog NC 62

Benefits

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

#### 2.2 SINUMERIK 840D sl

**OP 010C**

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

![OP 010C](image)

**OP 012**

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

![OP 012](image)

**SINUMERIK OP 015A**

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

![SINUMERIK OP 015A](image)
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 Multitouch operation (Page 27)

SINUMERIK OP 019

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

SINUMERIK OP 019 black

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

**Note:** see also Chapter Multitouch operation (Page 27)
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 Multitouch operation (Page 27)
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
2.2.3 Handheld units

Mini handheld unit

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

![Mini handheld unit image]

SINUMERIK HT 2

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

![SINUMERIK HT 2 image]
SINUMERIK HT 8

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

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

Wireless hand-held device Hubitron

For wireless operation, we recommend the Hubitron HBG 800-DP hand-held device.

Further information:
www.hubitron.de (www.hubitron.de)
3.1 Animated elements

<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 config.</td>
<td>Basic config.</td>
<td>Basic config.</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 config.</td>
<td>Basic config.</td>
</tr>
</tbody>
</table>

To illustrate which parameters affect what in machining operations, SINUMERIK Operate offers a new input support function with animated element sequences. For instance, the difference between chip breakage and chip removal when drilling or the precise probe sequence for a corner measurement can be shown.

Benefits

- Process reliability during the setup
- Increased reliability 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

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

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

**Benefit**

- Shortcuts in SINUMERIK Operate avoid the need for complicated menu operations and provide functions not previously expected from a CNC
CNC operation in manual mode (JOG)

4.1 TSM universal cycle

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

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

Benefit

- User-friendly manual input function with dialog prompting
4.2 Work offsets

The following work offsets are possible:

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

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

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

**Benefits**

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

The workpieces can be measured as follows:

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

The following measuring cycles are available:

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

The measurement results can be output in a measuring log (see Chapter Logging measurement results in JOG (Page 33)).

Benefits

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

<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 config</td>
<td>Basic config</td>
<td>Basic config</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 config</td>
<td>Basic config</td>
</tr>
</tbody>
</table>

The tool compensation values can be directly determined in setup mode.
The following variants are supported:
- Manual or switching probe
- Scratching with tool at known workpiece geometry

The measurement results can be output in a measuring log (see Chapter Logging measurement results in JOG (Page 33)).

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

The results for measuring in JOG can be logged. The standard log contains the measurement results of the most recently performed measuring method. The function is available as milling technology for the workpiece and tool measurement. Text format or table format can be selected for the output format.

The measuring log comprises the following data:

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

Benefit

- Simple logging of measured values in log files
4.6 Face milling cycle

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

You can specify the following parameters:

- Machining strategy and direction
- Machining limitations

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

Benefit

- Preparation of workpiece without having to create a part program
4.7 Retract

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

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

**Benefit**

- Machining can be continued at the point of interruption
4.8 Stop and retract (ESR)

4.8.1 Stop and retract (ESR) - Drive-autonomous

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

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

- Retract
- Extended stop
- Generator operation

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

**Benefit**

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

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D ADVANCED</th>
</tr>
</thead>
<tbody>
<tr>
<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: M61</td>
<td>Option: M61</td>
</tr>
</tbody>
</table>

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

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

**Benefit**

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

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

- Machining inclined surfaces
- Measure with inclined tool or table

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

Benefits

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

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

The following functions are available:

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

Benefit

- Simple and intuitive operation of cycle-controlled milling machines
4.10 Manual machine
## Tool management

### 5.1 Tool table

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D ADVANCED</th>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
<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: e.g. face milling cutter, taps and 3D probes
  - Clear tool name in plain text, example: CUTTER_HEAD_63MM
  - Max. of 9 cutting edges per tool
  - Tool length and diameter
  - Nose angle for drills or number of teeth for milling tools
  - Spindle direction and coolant (level 1 and 2) and up to four additional functions
- Direct transfer of the tool from the list in the program or for measurement
- 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

<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>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>BASIC</td>
<td>BASIC</td>
<td></td>
</tr>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td></td>
</tr>
</tbody>
</table>

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

- Monitor cutting time (T) in minutes or number of exchanges (C)
- Prewarning limit for timely preparation of new tools
- Provided the desired tool is not in the magazine, SINUMERIK Operate will request a manual tool change.

**Benefits**

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

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

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

### Benefit

- **Automatic exchange of identical tools for unmanned operation**
# 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)</td>
</tr>
<tr>
<td>828D</td>
<td>Network, USB storage media, compact flashcard</td>
</tr>
<tr>
<td>828D ADVANCED</td>
<td></td>
</tr>
</tbody>
</table>

USB / CF card can be used for data transport or for execution with EXTCALL

<table>
<thead>
<tr>
<th>Option P77</th>
<th>100 MB</th>
</tr>
</thead>
</table>

| SW | 3 MB | 5 - 10 MB |

Internal memory can be expanded via option P77 → 100 MB

External storage via option P75 → can be expanded almost without limit
## 6.2 Data storage - SINUMERIK 840D sl

<table>
<thead>
<tr>
<th>Internal memory</th>
<th>External storage</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>NCU</td>
<td>NCU</td>
</tr>
<tr>
<td>Option P77 + PCU</td>
<td></td>
</tr>
<tr>
<td>Option P77 + option P12</td>
<td></td>
</tr>
<tr>
<td>Option P77</td>
<td>100 MB</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>CNC user memory (option D00)</th>
<th>NCU 710.3B: 2 to 16 MB</th>
<th>NCU 720.3B and NCU 730.3B: 2 to 22 MB</th>
</tr>
</thead>
</table>

- Internal memory can be expanded via option P77 + PCU → 40 GB
- Internal memory can be expanded via option P77 + P12 → 6 GB
- Internal memory can be expanded via option P77 → 100 MB

1) HMI user memory, alternative to PCU

- Execution from external storage devices (EES, option P75)
- Network, USB storage media, compact flashcard

- Execution from the CNC expanded user memory (option P77)
- External storage via option P75 → can be expanded almost without limit
Data transfer

7.1 Program Manager

<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>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<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 names with as many as 24 characters for directories and files
- Manage subdirectories on external storage media, local drives and on the NC
- Store and display files of any type (e.g. *.png, *.pdf, *.dxf, *.xml)
- Manage and open DXF files
- Display all storage media in the program manager (with details of the storage capacity), including the network drives.
- Edit part programs on 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
7.2 Ethernet networking

The SINUMERIK controls are prepared for networking via Ethernet (TCP/IP) (RJ 45 connection).

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

Highlight

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

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D ADVANCED</th>
<th>SINUMERIK 840D sl BASIC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

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

The following search variants are available:

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

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.

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

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

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

**Benefits**

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

<table>
<thead>
<tr>
<th></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>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></td>
<td>BASIC</td>
<td>BASIC</td>
</tr>
<tr>
<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.

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
8.4 Basic block display

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

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

This function is available to you for programGUIDE (screenshot on left) and also for ShopMill (screenshot on right).

Benefit

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

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

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

While machining the workpiece the tool paths can be recorded on the screen of the control in the plan view, 3-side view or in 3D view. Workpiece depiction and views correspond to the graphic simulation.

Benefit

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

**Benefit**

- Small corrections and feeds of the tool in the tool direction are possible using a handwheel.
CNC operation in automatic mode (AUTO)

8.7 Handwheel override
9.1 80-bit NANO floating-point accuracy

<table>
<thead>
<tr>
<th></th>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D ADVANCED</th>
<th>SINUMERIK 840D si BASIC</th>
<th>SINUMERIK 840D si</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

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
CNC functions

9.2 Block change times

9.2.1 SINUMERIK 828D

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

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 limiting

<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>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td></td>
</tr>
</tbody>
</table>

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:

- Higher path accuracy through compensation of contouring errors
10.1 High speed settings

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

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

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

The following settings are possible:

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

Highlight

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

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

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

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

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

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

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

<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 si BASIC</th>
<th>SINUMERIK 840D si BASIC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option: S07</td>
<td>Option: S07</td>
</tr>
</tbody>
</table>

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

Enter the following settings for Advanced Surface:
- Tolerance of the machining axes
- Machining type
  - Finishing
  - Rough-finishing
  - Roughing
  - Deselection
- Multiple axis program yes/no

**Benefit**

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

10.2.2 Top Surface

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D ADVANCED</th>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>not available</td>
<td>Option: S17</td>
<td>Option: S17</td>
<td></td>
</tr>
<tr>
<td>Option: S17</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

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

The dynamic response is also optimized:
- Improved observance of the acceleration and jerk limits
- Lower vibration excitation of the machine

In addition to selecting the machining types (finishing, rough-finishing, roughing), the following settings are possible:
- Smoothing yes/no
- Multiple axis program yes/no
- Contour and orientation tolerance
The contour tolerance is shown as magnifying glass.

Standard values:
- Roughing 0.1
- Rough-finishing 0.05
- Finishing 0.01

The smoothing is also shown in the magnifying glass:
- Smoothing adds shine to the surface.
- Without smoothing, high-precision contours appear perfectly.

**Benefits**

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

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

Highlight

* Shorter machining times through optimum speed control
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.

**ShopMill machining step programming**

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

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

**Benefit**

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

11.1.1 Introduction

<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>

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.
11.1.2 Program editor

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

The following functions are included in the program editor:

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

Benefits

- Time saving by using a powerful editor when programming
- Even large part programs (many MB large) can be edited extremely fast
11.1.3 Languages

The CNC interpreter of 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
11.1.4 programGUIDE input support

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

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

Benefits

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

11.2.1 Introduction

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

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

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

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

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

Benefits

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

In ShopMill, associated sequences are interlinked with each other. The interlinked sequences are performed consecutively at the appropriate contours or pattern positions. In the following example, the sequences centering, deep-hole drilling and tapping are applied to 6 holes on the pitch circle pattern position.

Benefit

- Reduced programming time due to linking of machining steps
11.2.4 Graphic view

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

- Plan view of workpiece
- Front view of drilling operations

Benefit

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

11.2 ShopMill - machining step programming
Workpiece visualization

12.1 2D simulation

<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>

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

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

Benefits

- Maximum process reliability through simulation using real geometry values
- Perfect clarity by showing the workpiece dimensions with a scale
- Parallel simulation (background simulation) is possible in conjunction with the NCU 720 and NCU 730, i.e. simulating a part program while another part program is being simultaneously machined.
12.2 3D simulation

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

- **Reliability:**
  3 viewing planes and solid model of the finished part, with zoom to details and free rotation of the viewing angle

- **Support:**
  - Simulation speed controllable by override
  - Single block operation and start/stop available at any time

- **Checking:**
  Automatic calculation of machining time

### Benefits

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

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

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

Benefit

- More safety when handling mold making programs
13.1 CNC technology cycles for programGUIDE and ShopMill

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

Benefits

- Significant simplification of programming, even for complex jobs, using CNC technology cycles
- Consistency of cycles for programGUIDE and ShopMill
13.2 Highlights of machining cycles

13.2.1 Overview

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

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

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

- **Turning technology:**
  Groove, undercut, cutting with relief cut, thread undercut, thread cutting, chaining of threads, thread recutting

A selection of machining cycles is explained in more detail below.
13.2.2 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**
13.2 Highlights of machining cycles

13.2.3 Trochoidal milling

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

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

**Benefits**

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

<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</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

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

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

**Benefits**

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

<table>
<thead>
<tr>
<th></th>
<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>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

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

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

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

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

You can also choose between the following drilling strategies:

- With or without predrilling
- With or without pilot hole
- Soft first cut yes/no
- Chip breaking/removal
- Chip breaking and swarf removal
- 1 cut - drill in one step to the end depth
- Swarf removal to the piloting depth / safety clearance
- Retraction to the piloting depth / retraction plane
- Position pattern

Benefit

- Generate drill holes with more than one feed to any positions
13.3 Residual material detection for contour cycles

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

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

Benefits

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

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D ADVANCED</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option: P28</td>
<td>Option: P28</td>
<td>Option: P28</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option: P28</td>
<td>Option: P28</td>
</tr>
</tbody>
</table>

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

You can perform the following measuring tasks:

- Workpiece measurement: Correction of work offsets, correction of tool geometry or only measuring
- Tool measurement: Correction of tool geometries
- Display of measurement results
- Logging of measurement results
The following workpiece measuring versions are available:

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

Benefits

- Reliable quality of the manufactured parts by automatic measurement in the machine
- Fast programming for complex measuring tasks thanks to input screens with graphic support
- Measuring cycles are now also available for ShopMill sequence programs
13.5 Measure multiple axis kinematics

The kinematic measuring cycle CYCLE996 of SINUMERIK 840D sl allows axis kinematics of machine tools - equipped with several rotary axes - to be measured.

- Application during start-up and commissioning and for the control of the machine: The cycle makes control easier and improves the quality of the process as compensations of the rotary axis vectors - due, for example, to temperature variations, mechanical damage or other influential factors - can be checked automatically.

- Input of compensation values after the measuring process for the digital alignment of the rotary axis or acquisition of the measured values for documentation purposes. Tolerance values are freely selectable.

Benefits

- Simplest measuring cycle to measure machines with rotary axes in the shortest time
- Simple measurement or direct correction with freely selectable tolerance values
Complete machining

14.1 Cylinder surface transformation (TRACYL)

<table>
<thead>
<tr>
<th>SINUMERIK 828D Basic</th>
<th>SINUMERIK 828D</th>
<th>SINUMERIK 828D Advanced</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option: M27</td>
<td>Option: M27</td>
<td>Option: M27</td>
</tr>
</tbody>
</table>

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

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

**Programming in the run-off**

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

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

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

**Milling grooves with parallel walls**

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

**Benefits**

- Additional business through expansion of workpiece spectrum
- Reduction of set-up times by complete machining on one machine
14.2 Swivel plane (CYCLE 800)

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

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

Benefit

- Programming with standard cycles and easy transformation on the inclined plane through the swivel cycle
14.3 5-axis machining package (TRAORI)

<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: S33 (Package)</td>
<td>Option: S33 (Package)</td>
</tr>
</tbody>
</table>

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

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

Benefits

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

14.4.1 Introduction

<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>Basic configuration</td>
<td>Basic configuration</td>
</tr>
</tbody>
</table>

The milling technology in ShopMill and programGUIDE provides comprehensive technology cycles for turning and contour turning. Among others, the following functions are available for milling-turning:

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

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

Benefits

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

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

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

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

Benefits

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

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

Benefit

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

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

Benefits

- Maximum process reliability through simulation using real geometry values
Complete machining

14.4 Milling-turning
15.1 SINUMERIK Integrate Run MyRobot / EasyConnect

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D ADVANCED</th>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl ADVANCED</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td>Basic configuration</td>
<td></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.
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.
Tools and information

16

16.1 DXF reader

<table>
<thead>
<tr>
<th>SINUMERIK 828D BASIC</th>
<th>SINUMERIK 828D ADVANCED</th>
<th>SINUMERIK 840D sl BASIC</th>
<th>SINUMERIK 840D sl</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option: P56</td>
<td>Option: P56</td>
<td>Option: P56</td>
<td></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 (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 (http://www.siemens.com/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

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

You can activate 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 selling points

18.1 SINUMERIK 828D

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

Compact

- Rugged and maintenance-free design
- All relevant functions at a glance on the 10.4”/15.6” 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

Powerful

- 80-bit NANO floating-point accuracy for maximum precision of the workpiece results
- Advanced Surface, a unique calculation procedure to improve the workpiece surface and at the same time significantly reduce machining time
- 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
- ShopMill sequence 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 SINUMERIK 840D sl control has the following outstanding features for milling technology:

Open

- Standard and integrated 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 ShopMill machining step programming, programGUIDE and DIN/ISO programming

Flexible

- Optimum milling performance by being able to flexibly select the hardware that matches the machine
- High performance cycles for automatic workpiece and tool measurement
- Optimum functional scope of SINUMERIK MDynamics, milling package for 3-axis and 5-axis machining

Powerful

- 80-bit NANO floating-point accuracy for maximum precision of the workpiece results
- Top Surface offers enhanced surface quality, optimized dynamic response as well as improved user friendliness, in particular for poor data quality from the CNC programs provided by CAD/CAM systems
- Simplest monitoring of 5-axis machines with a kinematic measuring cycle
Index

5
5-axis machining package, 90, 93

8
80-bit NANO floating-point accuracy, 57

A
Advanced Surface, 62
Animated elements, 25

B
Basic block display, 52
Block change times, (SINUMERIK 828D), (SINUMERIK 840D sl)
Block search, 49

C
Collision monitoring, 106

D
Deep-hole drilling, 86
DIN/ISO language, 70
DIN/ISO programming, 70

E
Engraving cycle, 83

F
Feedforward control, 60

G
G-code editor, 69
Groove side offset, 91

H
Handwheel override, 55
High Speed Settings, 61

I
In-process measurement, 88

M
Measuring a tool, 32
Measuring cycle
  Multiple axis kinematics, 90
Mini handheld unit, 15, 23
Monitoring of tool life and workpiece count, 42
Multiple axis kinematics
  Measuring cycle, 90

O
Online Help, 26

P
Plunge milling, 85
Program editor, 69
Program manager, 47

R
Replacement tools, 43
Residual material identification, 87
RTCP, 93

S
Safety functions, 105
Safety Integrated, 105
Sequence editor, 73
Setup swivel cycle, 38
SIMATIC ITC 1200, 20
SIMATIC ITC 1500, 20
SIMATIC ITC 1900, 20
Index

Simulation
2D, 77
3D, 78
   Processing time, 78
SINUMERIK CNC4you, 103
SINUMERIK OP 010, 17
SINUMERIK OP 010C, 18
SINUMERIK OP 010S, 17
SINUMERIK OP 012, 18
SINUMERIK OP 015 black, 19
SINUMERIK OP 019, 19
SINUMERIK OP 019 black, 19
SINUMERIK OP 08T, 17
SinuTrain, 102
Speed control, 59
Swivel cycle, 92
Swivel machining plane, 92

T

Tool management, 41
Top Surface, 62
TRAORI, 90, 93
Trochoidal milling, 84
TSM universal cycle, 29

U

Unique features, (SINUMERIK 828D), (SINUMERIK 840D sl)