SIEMENS

SINUMERIK 828D

Milling

Control system overview
for machine tools' sales people

07/2009

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

with a safety alert symbol, indicates that minor personal injury can result if proper precautions are not taken.

CAUTION

without a safety alert symbol, indicates that property damage can result if proper precautions are not taken.

NOTICE

indicates that an unintended result or situation can occur if the corresponding information is not taken into account.

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 for the specific task, 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 adhered to. The information in the relevant documentation must be observed.

Trademarks

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

Disclaimer of Liability

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

Siemens AG
Industry Sector
Postfach 48 48
90026 NÜRNBERG
GERMANY

℗ 07/2009

Preface

Scope of validity

This document provides you with an overview of the range of functions included in the SINUMERIK 828D Version 2.6 operator panel controller 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 will be identified as follows:
  ☑ Basic configuration
- All functions not contained in the machine's basic configuration will be identified as follows:
  ☑ Option: ...
- You can find a summary of the most important unique selling points of the SINUMERIK 828D in the chapter "Summary of highlights".
- For information on marketing options through the machine manufacturer, please see the technical description of each machine.

Subject to change without prior notice

Contact person at machine manufacturer

Marketing & Sales

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

Service

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

Homepage:

http://www.machinemanufacturer.com

Contact person at Siemens

Country	Name	e-mail	Phone
Argentina	Santiago Fernandez Veron	santiago_fernandez.veron@siemens.com	+54 (11) 4738-3348
Belgium	Pieter Vanderhaeghen	pieter.vanderhaeghen@siemens.com	+32 253-69697
Bosnia-Herzegovina	Helmut Stralz	helmut.stralz@siemens.com	+43 51 707 29115
Brazil	Gustavo Marino	marino.gustavo@siemens.com	+ 55 (11) 3908-1752
Bulgaria	Helmut Stralz	helmut.stralz@siemens.com	+43 51 707 29115
China	Yang Yifei	yifei.yang@siemens.com	+86 10 64765236
Denmark	Stefan Karlstrand	stefan.karlstrand@siemens.com	+46 500 774-148
Germany (Headquarters)	Gerhard Micka	gerhard.micka@siemens.com	+49 9131 98 3314
Finland	Juha Meriaho	juha.meriaho@siemens.com	+358 50 59 26181
France	François Chevalier	francois.chevalier@siemens.com	+33 (0)1 49 22 35 19
Great Britain	Tony Bennison	tony.bennison@siemens.com	+44 780 882 2054
India	Narayanan Shankar	narayanan.shankar@siemens.com	+91 99451 88837
Indonesia	Andy Lesmono	andy_lesmono@cncdesign.com.au	+62 21 7918 6001
Italy	Nicodemo Megna	nicodemo.megna@siemens.com	+39 335 6328927
Canada	Sagar Arora	sagar.arora@siemens.com	+1 (416) 270-4964
Korea	Kim Sung Hyun	sunghyun.kim@siemens.com	+ 82 55 268 1906
Croatia	Matjaz Mlinsek	matjaz.mlinsek@siemens.com	+386 1 47 46 152
Lithuania	Juha Meriaho	juha.meriaho@siemens.com	+358 50 59 26181
Malaysia	Ridwan Aziz	ridwan_aziz@cncdesign.com.au	+603 5621 6126
Mexico	Marco Lopez	marco.lopez@siemens.com	+52 (55) 5328-2000
Netherlands	Frank de Korte	frank.de.korte@siemens.com	+31 70 333 1568
Norway	Stefan Karlstrand	stefan.karlstrand@siemens.com	+46 500 774-148
Austria	Helmut Stralz	helmut.stralz@siemens.com	+43 51 707 29115
Portugal	Molaguero Godoy	juan.molaguero.ext@siemens.com	+34 670929001
Romania	Matjaz Mlinsek	matjaz.mlinsek@siemens.com	+386 1 47 46 152
Russia	Alexander Kudinov	alexander.kudinov@siemens.com	+7 495 737 2442
Sweden	Stefan Karlstrand	stefan.karlstrand@siemens.com	+46 500 774-148
Switzerland	Hans-Peter Kueng	hans-peter.kueng@siemens.com	+41 585 581 524
Serbia Montenegro	Helmut Stralz	helmut.stralz@siemens.com	+43 51 707 29115
Slovakia	Matjaz Mlinsek	matjaz.mlinsek@siemens.com	+386 1 47 46 152
Slovenia	Matjaz Mlinsek	matjaz.mlinsek@siemens.com	+386 1 47 46 152
Spain	Molaguero Godoy	juan.molaguero.ext@siemens.com	+34 670929001
Taiwan	Jerry Lin	jerry.lin@siemens.com	+886 4 2261 9225
Thailand	Rajeev Madhyastha	rajeev_madhyastha@cncdesign.com.au	+66(0)2993 7485
Czech	Eva Klocova	eva.klocova@siemens.com	+420 605726829
Turkey	Taner Okayi	taner.okayi@siemens.com.tr	+90 216 459 3906
Hungary	Istvan Joo	istvan.joo@siemens.hu	+36 1 471 1598
USA	Jon Cruthers	jon.cruthers@siemens.com	+1 (847) 952 4124
Vietnam	Pham-Vu Trung	pham-vu.trung@siemens.com	+84 908 346 534

Homepage:

For further information please visit ...

http://www.siemens.de/cnc4you

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

Table of contents

1 Introduction

1.1 Application

With the SINUMERIK 828D you get a CNC system which is customized for the application in turning and milling machines. CNC, PLC, user interface and axis control for six CNC measuring circuits are all combined within a single compact unit. The controller provides comprehensive CNC functions such as kinematic transformations and a powerful tool management capability. The SINUMERIK 828D offers special capabilities for milling machines, with comprehensive drilling and milling operations, including milling with workpiece planes swiveled to any degree and milling cylindrical workpieces. The performance of the controller and the new motion control allow you to achieve mirror finish surfaces for mold making applications, with a minimum machining time.

The SINUMERIK 828D has eliminated all unnecessary functionalities; this is particularly noticeable in the graphical user interface. This means it is optimally suitable for use in the workshop. Operation, programming and maintenance can quickly be mastered without any heavy training requirement.

- Optimum operator guidance thanks to CNC input screens with animated elements
- Easy data exchange thanks to USB, CF and Ethernet interfaces on the panel front
- Integral mobile radio modem for optimum process monitoring via mobile phone

1.2 Machine spectrum

The SINUMERIK 828D is perfectly designed for equipping vertical and horizontal milling centers with up to six CNC measuring circuits.

As well as the milling spindle and geometry axes (X, Y, and Z axis), other machine units can also be operated as an alternative.

This includes:

- CNC reversible clamping device (A axis) for milling and hole machining on cylindrical workpieces
- Swiveling heads or swiveling tables for milling and hole machining in swiveled machining planes

System overview

2

2.1 SINUMERIK 828D

The SINUMERIK 828D is a complete unit comprising screen, CNC keyboard and CNC electronics.

The motors can be easily connected to the digital drives via DRIVE-CLiQ. In combination with the modular structure of the SINAMICS S120 drive system, this design is conceived to ensure very simple and rugged installation with minimum wiring overhead.

The performance range of the controller has been selected to meet the requirements of standardized turning and milling machines - from one-off production runs to industrial scale manufacture.

- Digital drive controller
- Modular design for drive controller and power unit
- Up to 6 axes/spindles for milling applications
- Intelligent control functions meeting the highest standards of machining technology

Highlights

- **Maximum reliability due to compact design with few interfaces**
- **The same hardware for milling and for turning, leading to optimum spare parts management.**

2.2 SINUMERIK 828D operator panels

The operator panel consists of a hard-wearing die-cast magnesium alloy and is available in two versions, for horizontal and vertical mounting.

- 10.4" TFT color display
- Integrated QWERTY full CNC keyboard with short-stroke keys
- USB, CF card and Ethernet interfaces on the operator panel front

Highlights

- **All relevant functions at a glance, thanks to horizontal and vertical softkeys**
- **Simple data handling using easily accessible sockets for USB and compact flash card storage media on the front panel**

2.3 Performance versions

The 828D is available in two performance versions: the PPU 260/261 and the PPU 280/281. These allow optimum matching to the demands of the machine.

Scope of performance	PPU 260 / 261	PPU 280 / 281
Minimum block cycle time	~2 ms	~1 ms
CNC work memory	3 MB	5 MB
Maximum number of tools/cutting edges	128/256	256/512

Highlights

- **Outstanding performance even with the standard package**
- **Maximum mold making performance with the expansion stage PPU 280/281**

2.4 Mini handheld unit

For the purpose of setting up the machine, you can install the mini handheld unit pictured below.

Highlight

- **Operation as close as possible to the workpiece via a mobile handheld unit**

2.5 Maintenance-free operation

The SINUMERIK 828D offers maintenance-free operation:

- High reliability, as the SINUMERIK 828D does not have a hard disk, battery or fan
- Storage of part programs on a NVRAM, so no loss of data even during an extended loss of power

Highlight

- **Highest machine availability thanks to reliable hardware**

2.6 Languages of operating software

☑ Basic configuration

The following languages are available on the operator interfaces of the SINUMERIK 828D. Pressing the alt + L keys enables softkey switching between languages.

- Chinese Simplified
- Chinese traditional
- German
- English
- French
- Italian
- Korean
- Portuguese
- Spanish

☑ on request

Language extensions for the HMI sl operating software are available on request for the following languages:

- Danish
- Finnish
- Japanese
- Dutch
- Polish
- Romanian
- Russian
- Swedish
- Slovakian
- Czech
- Turkish
- Hungarian

Highlights

- **Operator interface in your language so that the CNC is easy to learn and safe to operate**
- **All languages are available on the control and can online be changed**

CNC operation in manual mode (JOG)

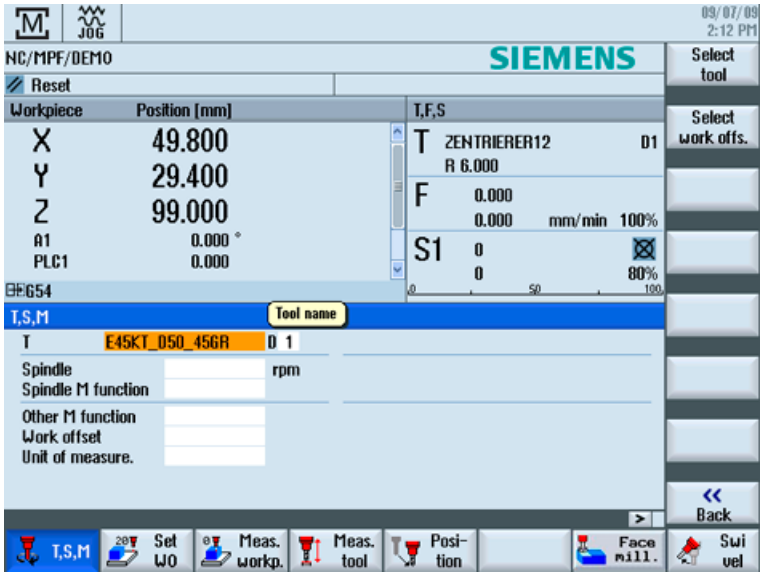
3

3.1 TSM universal cycle

☑ Basic configuration

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

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



Highlight

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

3.2 Face milling cycle

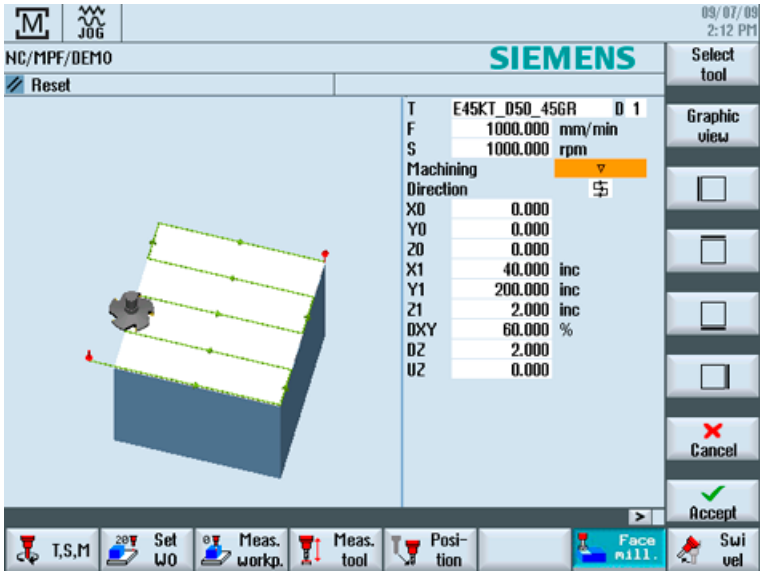
☑ Basic configuration

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

You can specify the following parameters:

- Machining strategy and direction
- Machining limitations

The input values are retained even after switching off and on again, so that the user can always restart his face milling operation with minimum manpower.



Highlight

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

3.3 Measure tool

☑ Basic configuration

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

The following variants are supported:

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

Highlight

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

3.4 Measure workpiece

☑ Basic configuration

The workpieces can be measured as follows:

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

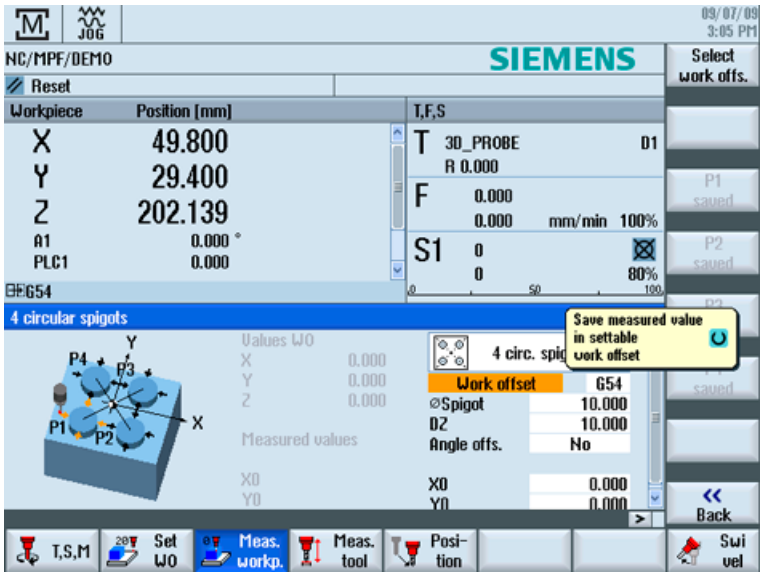
The following measuring variants are available (also if only measuring):

- Point measurement for edges
- Orienting the edge (angle)
- Inner/outer corner (3 or 4 points),

☑ Option: Extended operator functions

The following additional measurement variants are available as options:

- Orienting the edge by means of 2 holes/pins
- Rectangular or circular pockets, rectangular or circular pins
- Center point of 3 or 4 holes or pins
- Orienting the plane with three points



Highlight

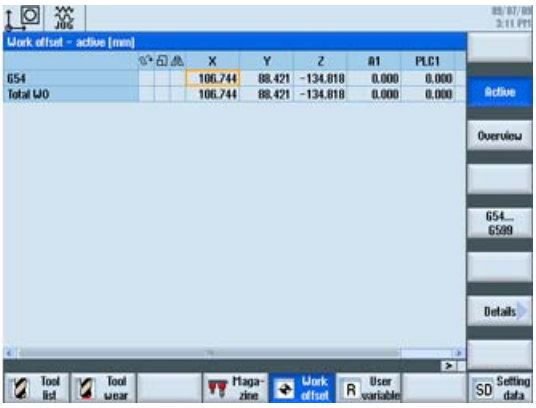
- **Time saving due to user-friendly determination of the workpiece's clamping position instead of orienting the workpiece by hand**

3.5 Work offsets

☑ Basic configuration

The following adjustable work offsets are available to you:

- A basic offset
- Maximum of 99 work offsets
  (G54, G55 ...)
- Each work offset with axis rotation and fine offset



Highlights

- **Flexible machining due to great number of adjustable work offsets**
- **Optimum overview of all active work offsets incl. swiveling**
- **Unlimited possibilities of programmable work offsets**

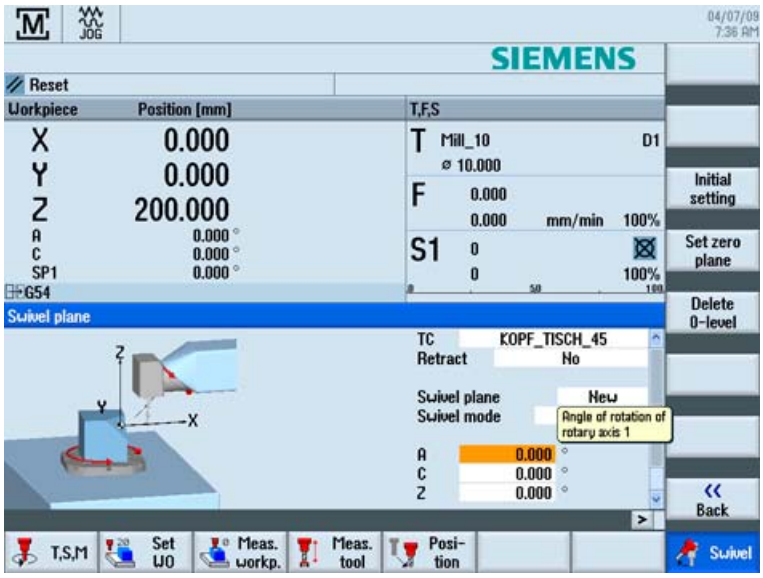
3.6 Swiveling in setup mode

☑ Basic configuration

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

- Machining inclined surfaces
- Measure with inclined tool or table

The plane can be swiveled directly including rotation of coordinates or axial swiveling.



Highlights

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

Tool management

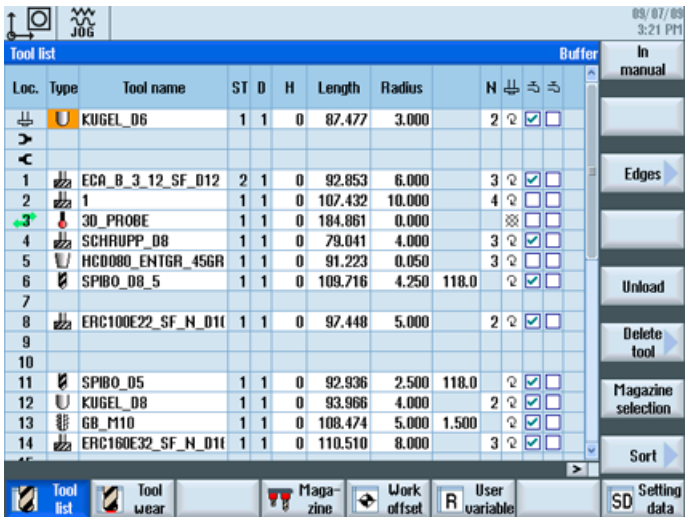
4

4.1 Tool table

☑ Basic configuration

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

- The maximum number of tools/cutting edges for the controllers is:
  - PPU 260/261: 128/256
  - PPU 280/281: 256/512
- 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



Highlights

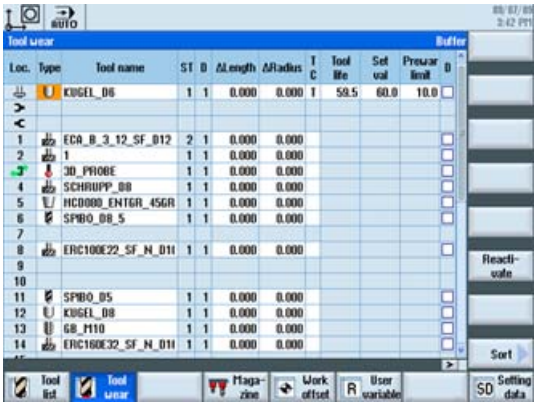
- **All tool data at a glance**
- **Simple and secure handling via unmistakable tool names**

4.2 Monitoring of tool life and quantity of workpieces

☑ Basic configuration

You can use the SINUMERIK 828D 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
- If the desired tool is not in the magazine, the SINUMERIK 828D will request a manual tool change.



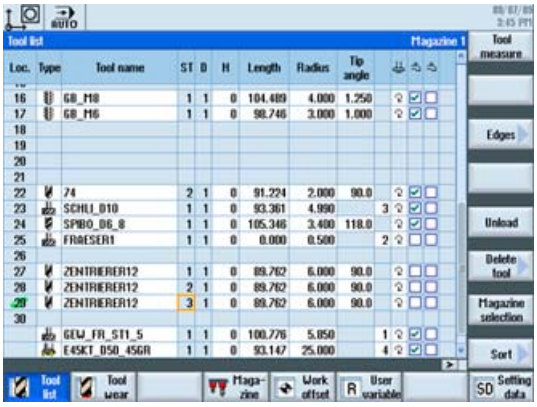
Highlights

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

4.3 Replacement tools

☑ Option: Replacement tools for tool management

If needed, you can even use the SINUMERIK 828D to manage replacement tools (sister tools). Tools with the same name are created as replacement tools. The replacement tools are identified with an incrementing number in the ST column.



Highlight

- **Automatic tool exchange for unmanned operation**

User memory

5

5.1 Buffered CNC work memory

☑ Basic configuration

	PPU 260 / 261	PPU 280 / 281
CNC work memory	3 MB	5 MB

Highlight

- **Exceptionally large storage space in basic configuration already**

5.2 Memory expansion by compact flash card

☑ Basic configuration
CF card not included in the scope of delivery

A compact flash card slot is located directly at the operator panel front of the SINUMERIK 828D.

- Cover can be closed while the card is inserted in order to protect the unit from dust.
- No special software necessary for reading/writing compact flash cards via PC

Note:
Please ensure you use robust, high-quality compact flash cards for industrial use.

Highlight

- **Commercially available mass storage as low-cost memory expansion**

Data transfer

6

6.1 Interfaces

☑ Basic configuration

The SINUMERIK 828D has the following interfaces available on the front panel of the unit. You can access the respective storage media using the program manager.

Highlights

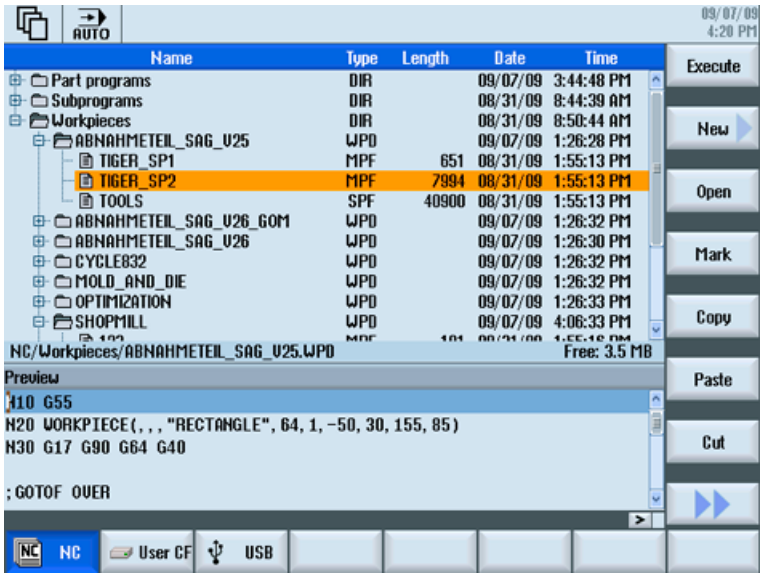
- **Freedom in the selection of the mass storage device**
- **Optimum accessibility for data transfer directly at the front of the operator panel**

6.2 Program manager

☑ Basic configuration

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

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



Highlight

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

6.3 Data transfer by serial interface

☑ Basic configuration

The SINUMERIK 828D permits easy data transfer to and from PCs, using the RS232C interface.

The main application for this is data archiving of part programs

Note:
You can use the RS232C interface either as a serial interface or the modem connection for Easy Message.

Highlight

- **Easy and tried and tested data transfer, even using the serial interface**

6.4 Data transfer using a USB memory stick or compact flash card

☑ Basic configuration

The SINUMERIK 828D has a USB stick socket on the front panel and another on the rear panel. The socket for a compact flash card is located on the front panel.

- Storage media can be inserted or removed during operation, i.e. the machine does not have to be restarted in order for the storage medium to be recognized.
- Loading, editing and executing of part programs from the storage medium
- When executing part programs from a storage medium there is no loss of speed (DNC operation), in which case executing from a CF card is recommended
- No special software is necessary for reading/writing the storage medium stick on the PC

Highlight

- **Efficient and reliable solution for handling a large volume of user data**
- **Freedom in the selection of the mass storage device**
- **Part programs can also be edited directly on the storage medium**

6.5 RCS Commander

☑ Basic configuration

Installation of the RCS Commander from the CD-ROM (included in the scope of delivery)

☑ Option: RCS Host remote diagnostics function

The RCS Commander is a powerful free-of-charge tool for the PC. It allows you to move data easily into the CNC controller using drag & drop. In addition, it offers a convenient means of viewing the contents of the CNC screen display on the PC. Simply connect your PC or Notebook on which the RCS Commander is installed to the Ethernet interface on the front panel. The network configuration will automatically include the SINUMERIK 828D. No knowledge of networks is required.

The SINUMERIK 828D also supports remote diagnostics via the Ethernet (see options). When connecting the PC to several machines, only one PC license is necessary for remote diagnostics via Ethernet. All machine tool controller diagnostic functions are also available in remote diagnostics.

Highlight

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

6.6 Ethernet networking

☑ Option: Control additional drives using Ethernet

The SINUMERIK 828D is set up for Ethernet (TCP/IP) networking (RJ45 connection).

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

Note:
In addition to the Ethernet interface for a point to point connection to a PC (RCS Commander), the SINUMERIK 828D also has a second Ethernet interface for a fixed factory network.

Highlight

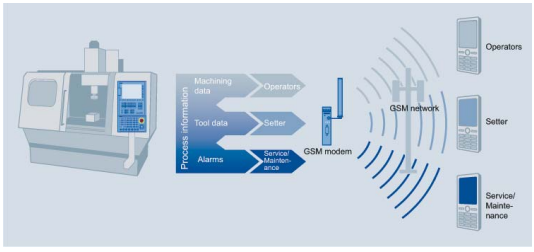
- **Easy and economical connection via Ethernet (TCP/IP) to Windows PCs or Unix workstations**
- **No software needs to be installed on the servers**

6.7 Easy Message

☑ Basic configuration, SIM card not included in the scope of delivery
☑ Option: Mobile-radio modem

Easy Message allows process information such as the degree of wear of tools to be sent simply by SMS to your cellular phone. The mobile phone modem with its associated mobile phone antenna ensures optimum transmission characteristics even in a harsh industrial environment.

- Personalized messaging thanks to user management
- Any text messages can be sent out directly from the CNC part program
- Short reaction time when performing service work is achieved by transmitting fault messages or maintenance information by SMS



Highlight

- **The right information to the right person thanks to user management**
- **Short response times allow perfect service**

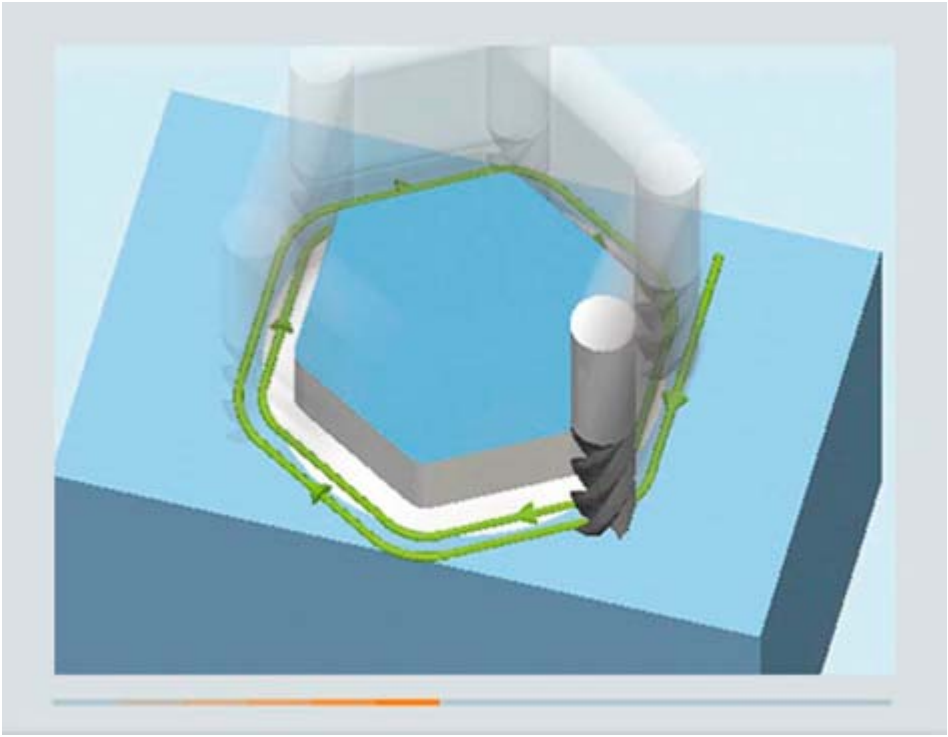
Graphical support functions

7

7.1 Animated Elements

☑ Basic configuration

For indication of which parameters affect what in the machining, the SINUMERIK 828D 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.



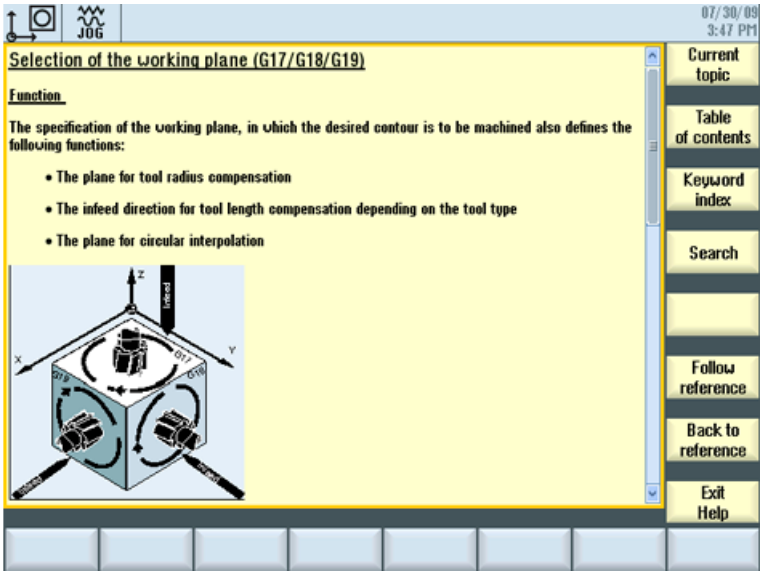
Highlight

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

7.2 Onboard documentation

☑ Basic configuration

For each input field in the operating screens, SINUMERIK 828D automatically displays help in the form of a "cursor text". The SINUMERIK 828D provides further information in the form of a complete context-sensitive help system with many useful details and graphics.



Highlight

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

CNC operation in automatic mode (AUTO)

8

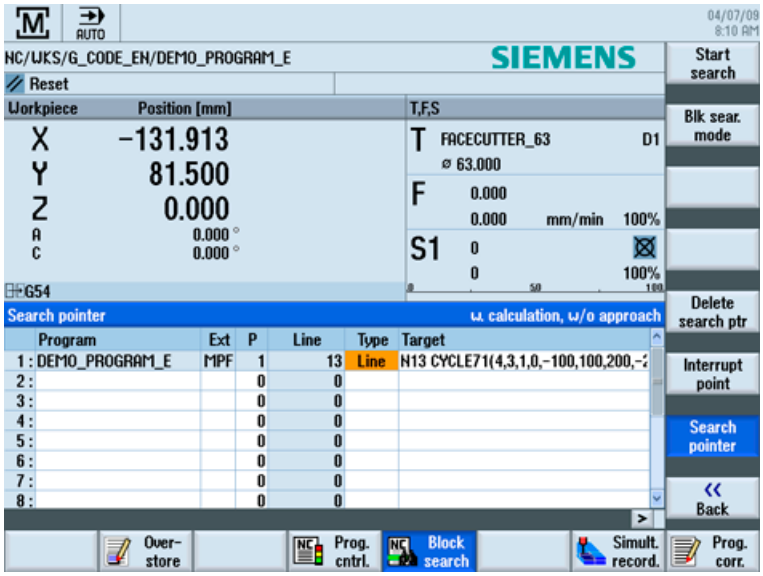
8.1 Block search

☑ Basic configuration

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

The following search variants are available:

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



Highlights

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

8.2 Program control

☑ Basic configuration

Single block

Single block mode can be activated for startup of the program. For this purpose a program stop occurs after each traversing block.
Work plan programs maintain the alternative of stopping processing after each plane infeed.

Program test

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

Reduced rapid traverse

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

Program editing

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

Repositioning to the contour (REPOS)

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

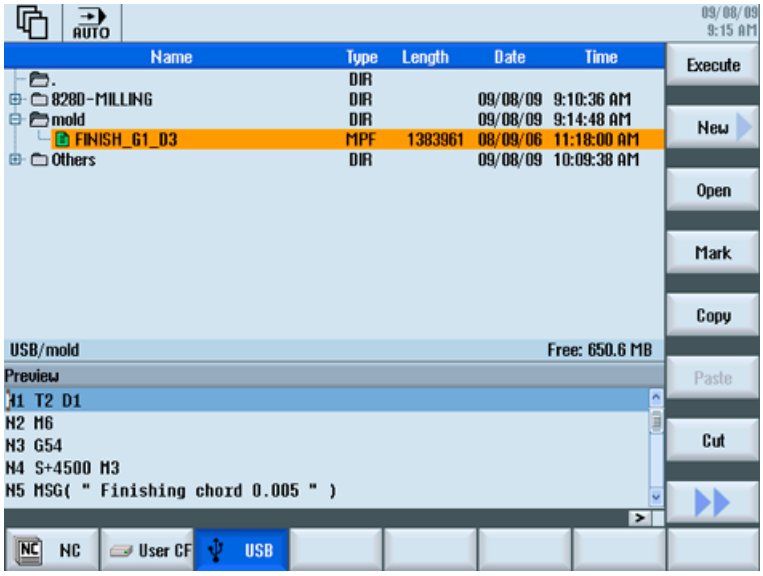
Highlights

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

8.3 Execution from external source

☑ Basic configuration

The SINUMERIK 828D offers you the facility to select and execute part programs directly from the CF card or via the network.



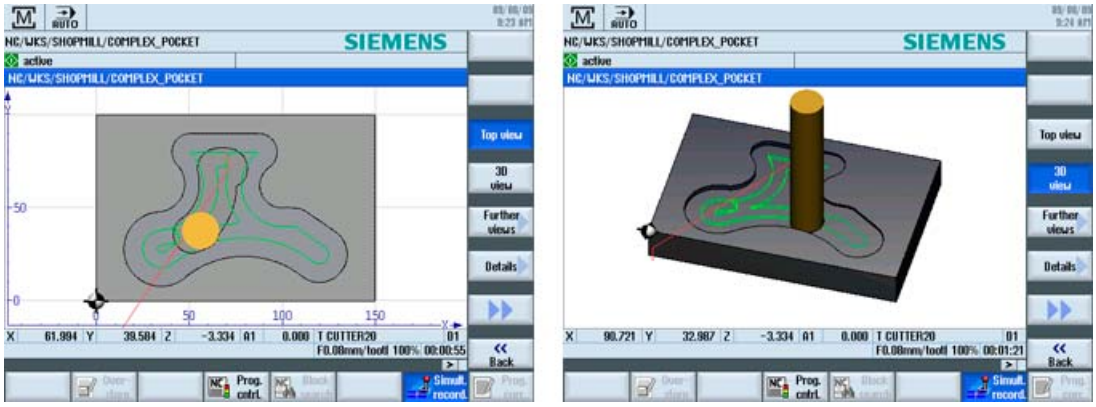
Highlight

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

8.4 Simultaneous recording

☑ Option: Simultaneous recording

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



Highlight

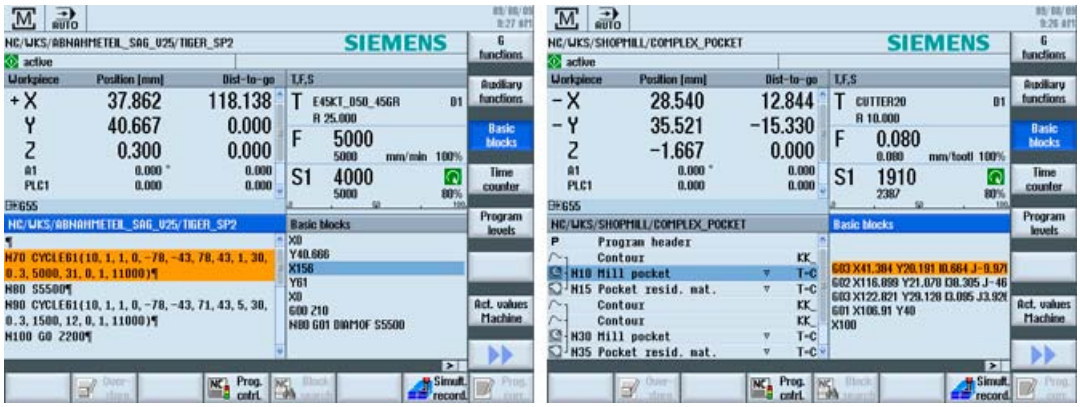
- **Machining can also be monitored in a complex machine room**

8.5 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).



Highlight

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

CNC performance capability and optimization functions

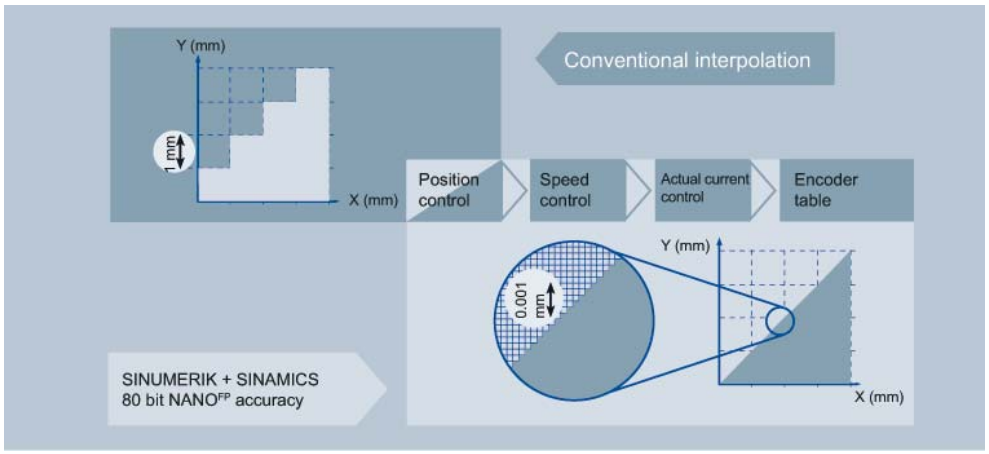
9

9.1 80bit NANO FP accuracy

☑ Basic configuration

The accuracy of the workpiece is determined by more factors than just the mechanical characteristics of the machine. The CNC controller also contributes to a critical degree towards the precision of the workpieces. The SINUMERIK 828D offers a multitude of CNC functions for this purpose.

The SINUMERIK 828D and the SINAMICS drive calculate using 80-bit floating point accuracy. This enables a calculation accuracy much less than a nanometer. This exactness is available not only for closed loop position control but also for power and closed-loop speed control and also for sensor evaluation of the drive.



Highlight

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

9.2 Block change times

☑ Basic configuration

The classic block cycle time plays a secondary role in the SINUMERIK 828D due to the high-performance velocity control functions.

Grouping linear blocks to splines greatly lowers the internal amount of data. This, in turns, greatly reduces the block change time.

This is demonstrated by a specific test situation: A SINUMERIK 828D, PPU 280/281 with an active online compressor processes 10,000 G01 blocks in approx. 10 seconds. This corresponds to a block change time of approx. 1 msec.

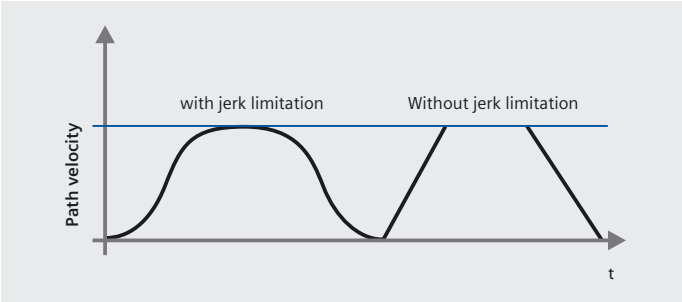
Highlight

- **Internal reduction of block change times by reducing the data volume**

9.3 Jerk limitation

☑ Basic configuration

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



Highlights

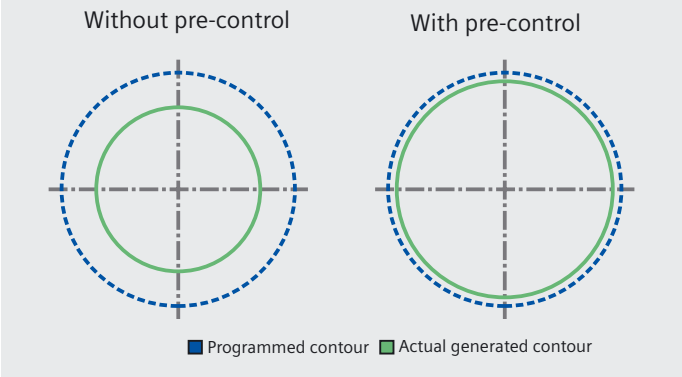
- **Longer machine lifespan through protection of the mechanical components**
- **Higher path accuracy through softer acceleration**

9.4 Dynamic feedforward control

☑ Basic configuration

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

Example:



Highlight

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

Mold making

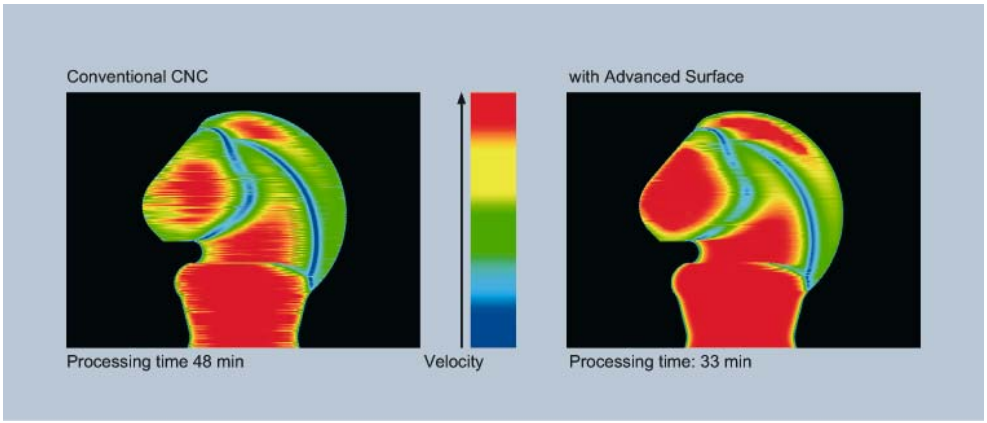
10

10.1 Advanced Surface

☑ Basic configuration

The SINUMERIK 828D Advanced Surface facility allows you to generate high-quality surfaces on mold making workpieces.

- **Perfect surface**
  The SINUMERIK 828D 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 ensures 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 tolerant Advanced Surface filter algorithms guarantee you the best workpiece surfaces and shortest machining times, after just a single optimization of the system.



Highlight

- **Improved workpiece surface with reduced machining time**

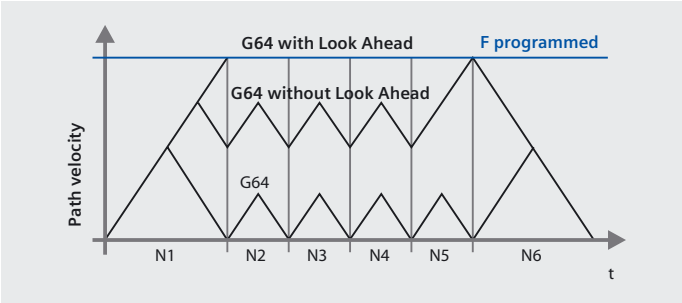
10.2 Look Ahead

☑ Basic configuration

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

Depending on the performance variant, the SINUMERIK 828D offers up to 150 Look Ahead blocks.

Performance variant	Number of Look Ahead blocks
PPU 260 / 261	100
PPU 280 / 281	150



Highlight

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

CNC programming methods

11

The SINUMERIK 828D offers you a choice of the following programming methods:

programGUIDE and SINUMERIK CNC programming

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

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

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

ShopMill

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

ShopMill is particularly well suited to small series production.

ISO dialect and SINUMERIK CNC programming language

The SINUMERIK 828 allows you to perform ISO programming using the SINUMERIK CNC programming language combined with or exclusively in ISO dialect.

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

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

Highlights

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

11.1 programGUIDE and SINUMERIK CNC programming

11.1.1 Introduction

☑ Basic configuration

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

- DIN/ISO editor
- Languages
- programGUIDE input support

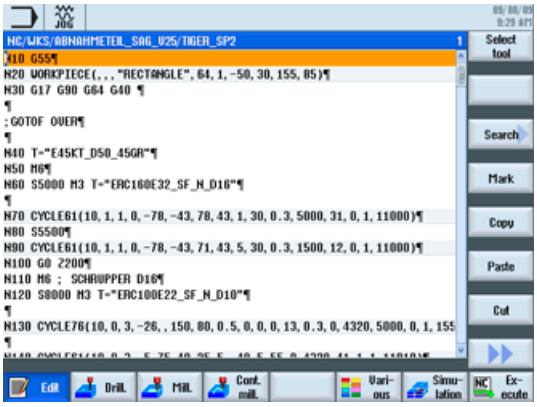
Programming with programGUIDE is available in the basic scope of the SINUMERIK 828D.

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 are available for the most complex machining.

The following functions are included in the program editor:

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



Highlights

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

11.1.3 Languages

The CNC interpreter of the SINUMERIK 828D can also process more complex CNC commands, in addition to DIN 66025 standard commands. The commands are presented in clearly readable form.

The following commands are available:

- **G-code**
  G-code in accordance with DIN 66025 and in ISO dialect mode
- **G functions**
  G0, G1, G2, G71 ...
- **Language commands (extended G functions)**
  CIP, SOFT, BRISK, FFWON ...
- **Frame operations (programmable work offsets)**
  The workpiece coordinate system can be shifted, scaled, mirrored or rotated with the commands TRANS, SCALE, MIRROR, ROT.
- **R parameters (arithmetic parameters)**
  300 predefined R parameters are available as arithmetic parameters (floating point format).
- **User variables**
  The user can define his own variables by name and type.
- **System variables**
  System variables can be read/written in all programs. They enable access to work offsets, tool offsets, axis positions, measurement values, control conditions etc.
- **Calculation operations**
  The following mathematical calculation operations are available for linking the variables:
  calculation operations + - * / sin cos exp etc.
  logical operations == <> >= etc.
- **Program control structures**
  BASIC-style language commands are available for flexible programming of the user cycles: IF-ELSE-ENDIF, FOR, CASE ...

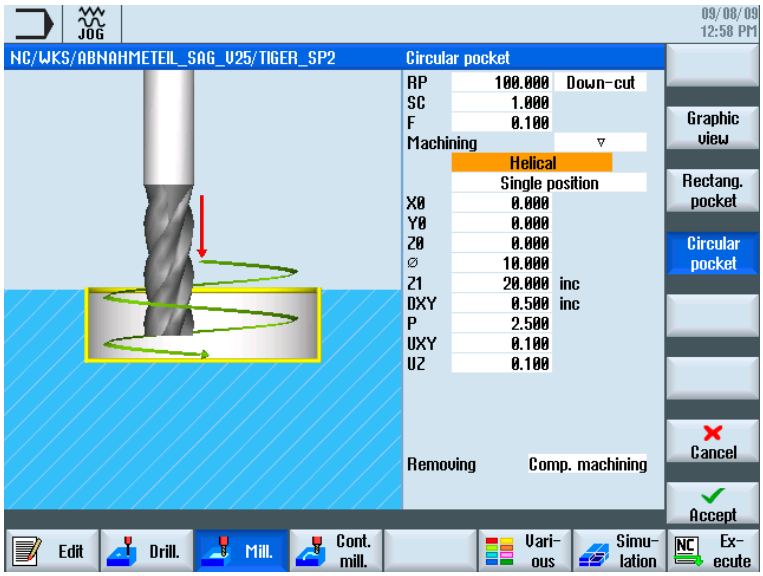
Highlights

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

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 continuity.

Of course, the commands for tool, feedrate and spindle speed are still programmed in the DIN/ISO editor.



Highlights

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

11.2 ShopMill

11.2.1 Introduction

☑ Option: ShopMill machining step programming

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

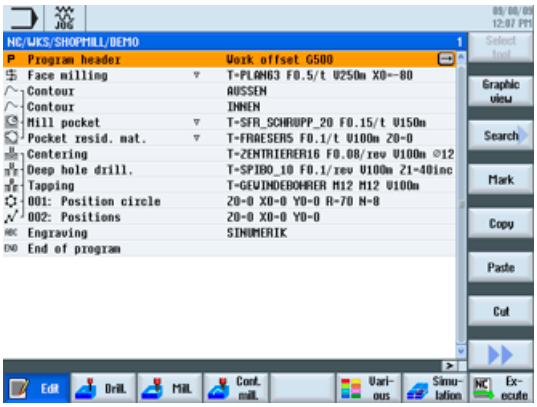
- Machining plan editor
- Interlinking of sequences
- Broken-line graphics

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

11.2.2 Machining plan editor

The graphical programming is performed via a graphic interactive machining plan 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 technical and geometric parameters are entered in screen forms. Simple, intuitive programming with sequences can always be expanded very flexibly by inputting DIN/ISO blocks and control functions.



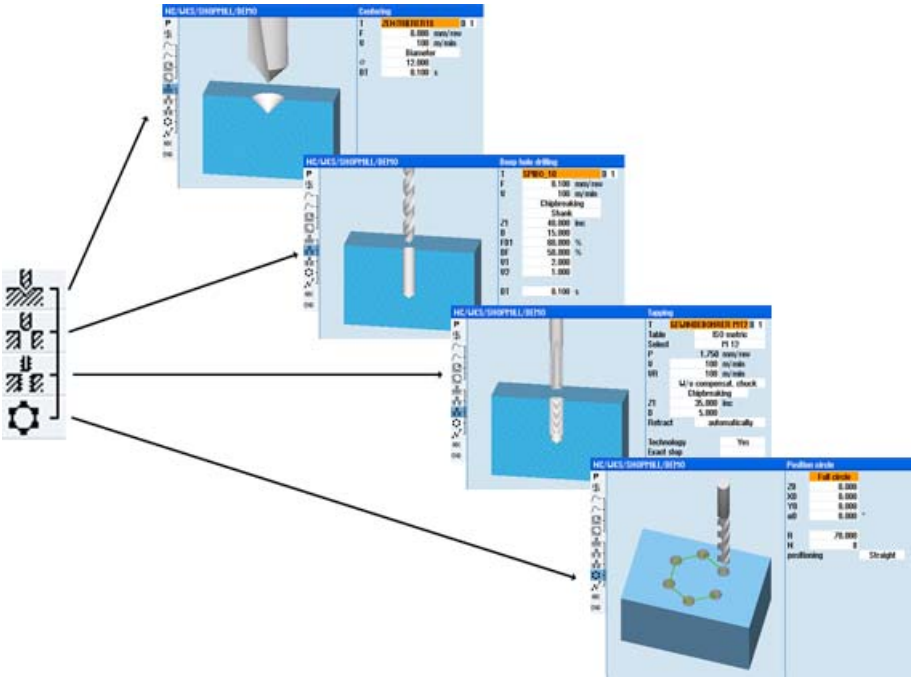
Highlights

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

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.



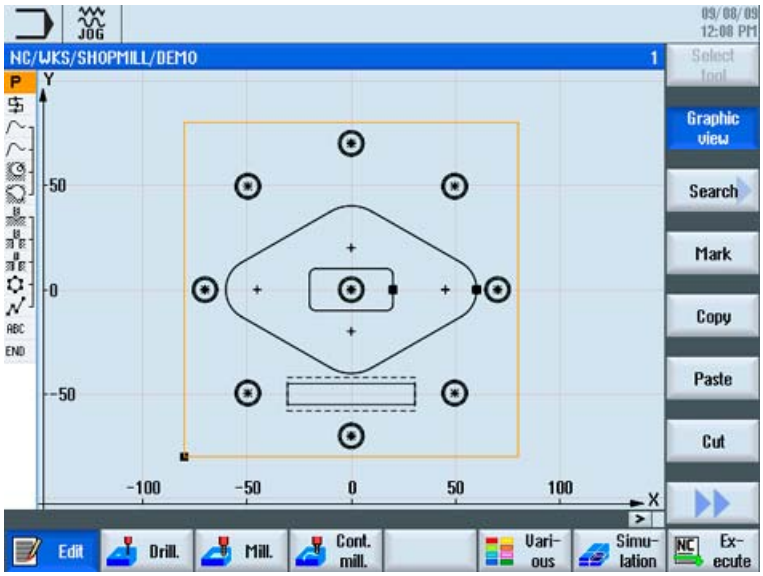
Highlight

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

11.2.4 Broken-line graphics

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

- Plan view of workpiece
- Front view of drilling operations



Highlight

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

11.3 Online ISO dialect interpreter

☑ Basic configuration

It is always useful to be able to speak a foreign language. This is true even for a global player such as the SINUMERIK 828D. If you prefer classic ISO programming you can continue to use it. You can even mix ISO programming with the SINUMERIK CNC programming language. This enables you to increase the productivity and flexibility of your machine step by step.

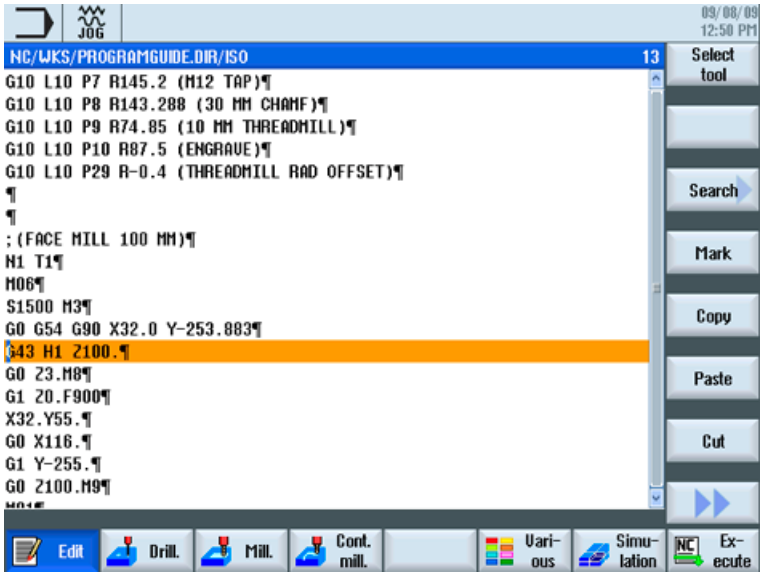
In the controller, G commands from Siemens are interpreted as standard.

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

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

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

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



Highlights

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

Simulation

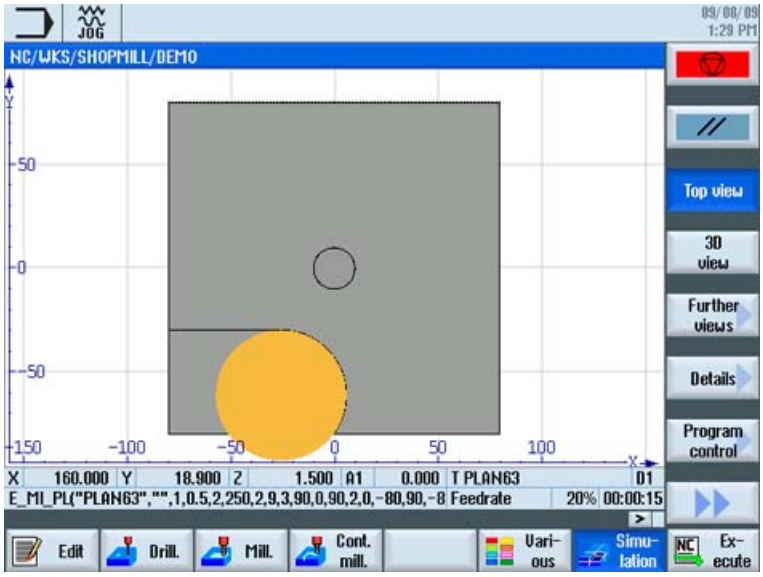
12

12.1 2D simulation

☑ Basic configuration

The SINUMERIK 828D 2D simulation offers you the facility to make optimum and reliable preparations for machining workpieces, including detection of collisions. Calculating the machining time also supports optimum calculation of tooling costs.

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



Highlights

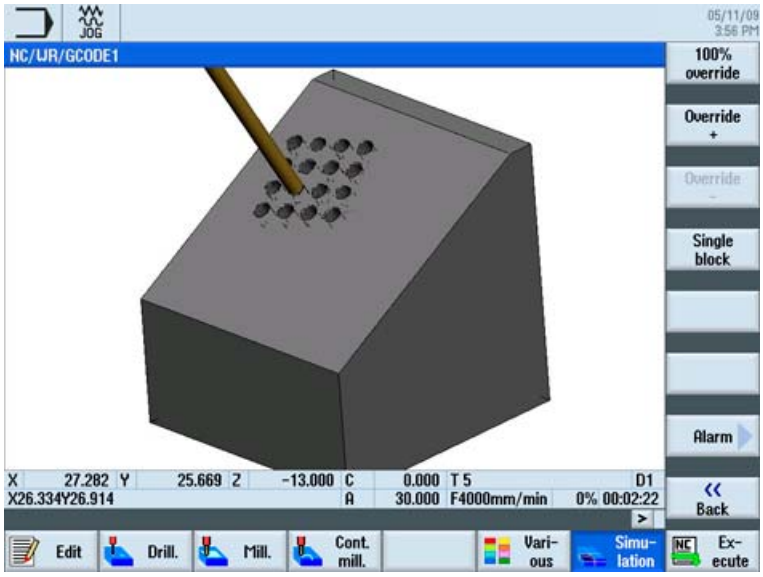
- **Maximum process reliability through simulation using real geometry values**
- **Perfect clarity by showing the workpiece dimensions with a scale**

12.2 3D simulation

☑ Option: 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



Highlights

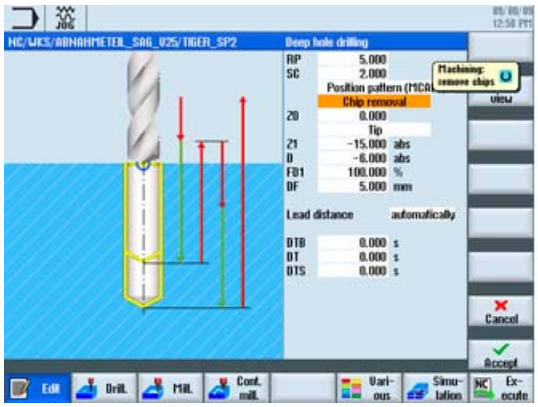
- **Particularly realistic simulation through representation of the tool**
- **Optimum help and reliability in programming and in quotation costing**

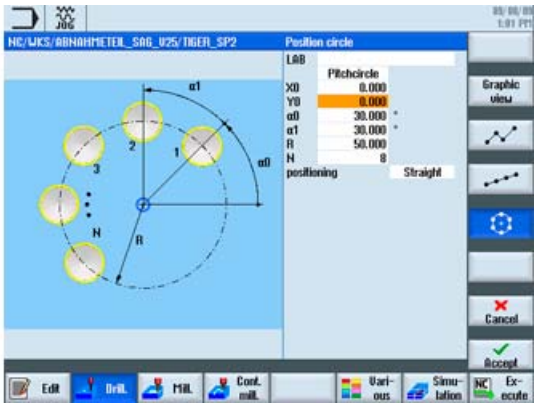
CNC technology cycles

13

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.





The SINUMERIK 828D offers you a unique range of CNC technology cycles for standard machining – including an engraving cycle. The assignment to the machining positions is performed very simply using a wide selection of ready position patterns, even with swiveled planes for milled workpieces.

The integrated swivel cycle allows the machining plane to be swiveled to any degree – no need for CAD/CAM or pocket calculator. This means that the programming of oblique surfaces is not a problem.

For sustained accuracy of workpieces in an ongoing machining process, the SINUMERIK 828D supports you with the measuring cycles' optional package.

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

The geometry processor supports you when inputting pocket and island contours. The material removal paths are generated fully automatically by the SINUMERIK 828D. So as to achieve maximum productivity, you can pre-machine using a large milling tool. The optional identification of residual material permits selective remachining of the remaining corners using a small tool.

Highlights

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

13.2 Highlights of machining cycles

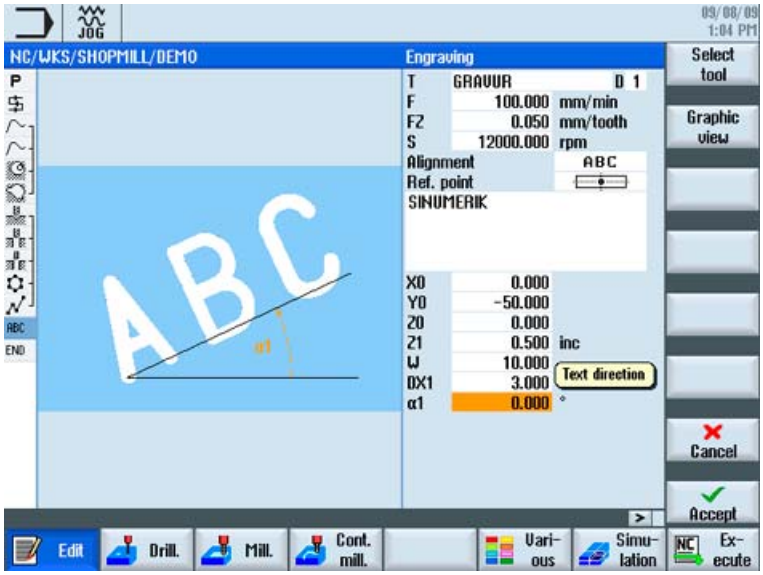
13.2.1 Engraving cycle

☑ Basic configuration

The engraving cycle is used to engrave a text on a workpiece along a line or arc. You can enter the text as fixed text or assign it via a variable as variable text.

Examples of variable texts:

- Date and time
  The values for the date and time are read from the CNC.
- Quantity
  The "Quantity" variable is available as a pre-defined user variable
- Numbers
  When outputting numbers (e.g. measurement results), you can select the output format (digits before and after the point) of the number to be engraved.
- Text
  Instead of entering a fixed text in the engraving text field, you can specify the text to be engraved via a text variable (e.g., _VAR_TEXT="ABC123").



Highlights

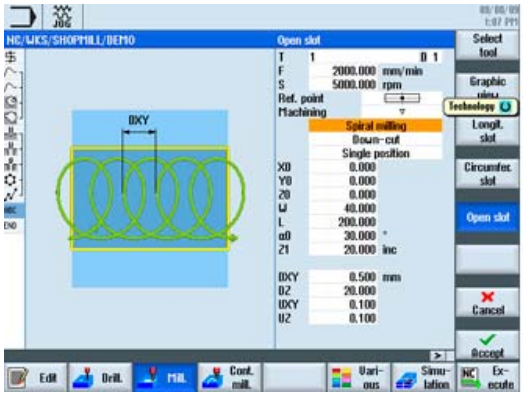
- **Reduction of set-up times by complete machining on one machine**
- **Simple program input of engraving**

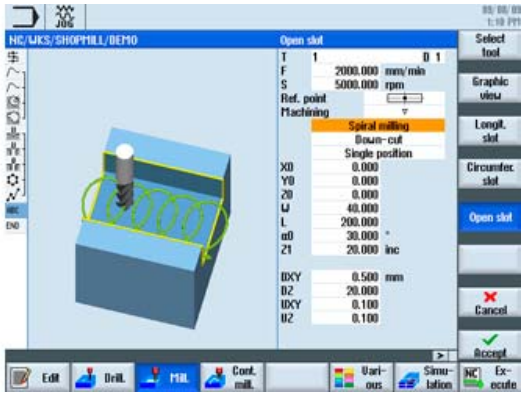
13.2.2 Trochoidal milling

☑ Basic configuration

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

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





Highlights

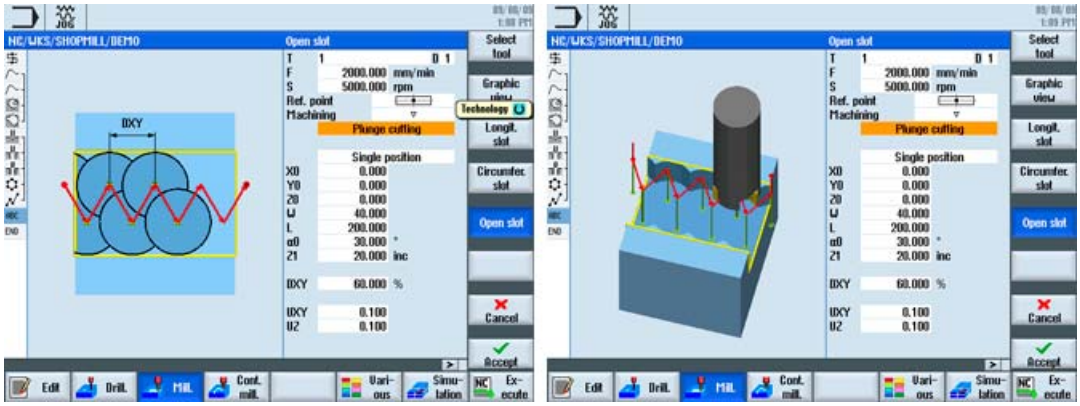
- **Innovative CAM function now available directly on the controller**
- **Reduction in the machining time for slot milling by up to 50%**

13.2.3 Plunge milling

☑ Basic configuration

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

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



Highlights

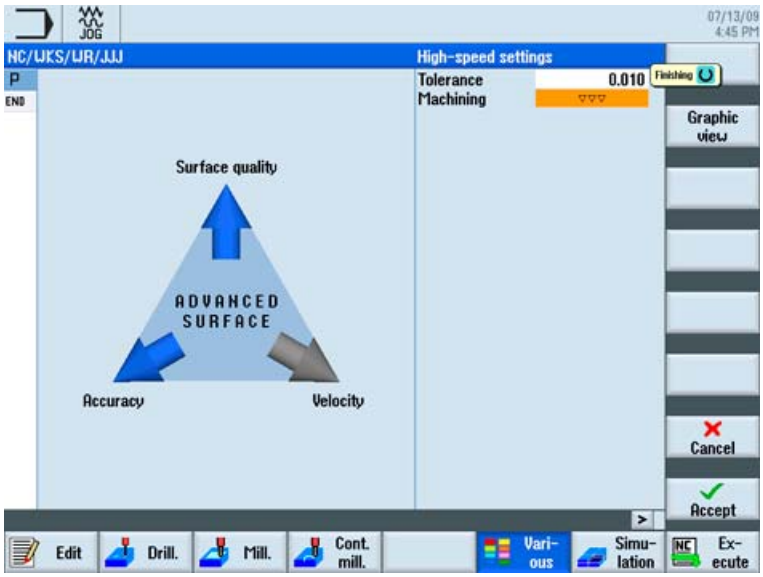
- **Less vibrations and deeper cutting depth thanks to the new machining strategy plunge milling**
- **Reduced cutting pressure and distortion enable higher productivity when machining thin-walled workpieces**

13.2.4 High speed settings

☑ Basic configuration

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

- This cycle is called up within the DIN/ISO editor or in ShopMill. Calling up this function activates Advanced Surface.
- For simplification, the selection is limited to the roughing, pre-finishing, finishing machining types (or deselection) and the tolerance band from the CAD/CAM system.



Highlight

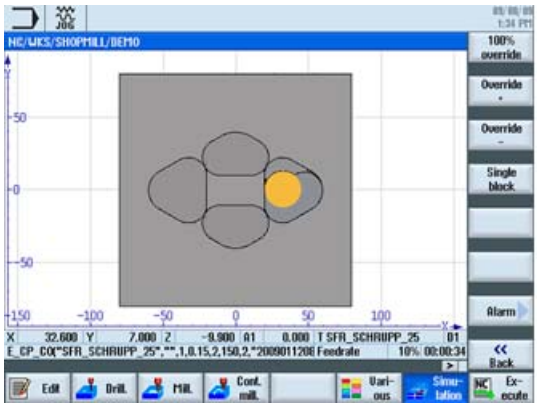
- **Simple and easily understandable parameterization of the required machining type (roughing, pre-finishing or finishing) with an interactive screen**

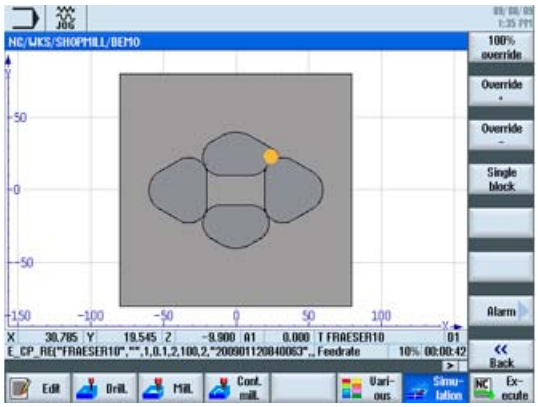
13.3 Residual material identification

☑ Option: Residual material identification

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

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





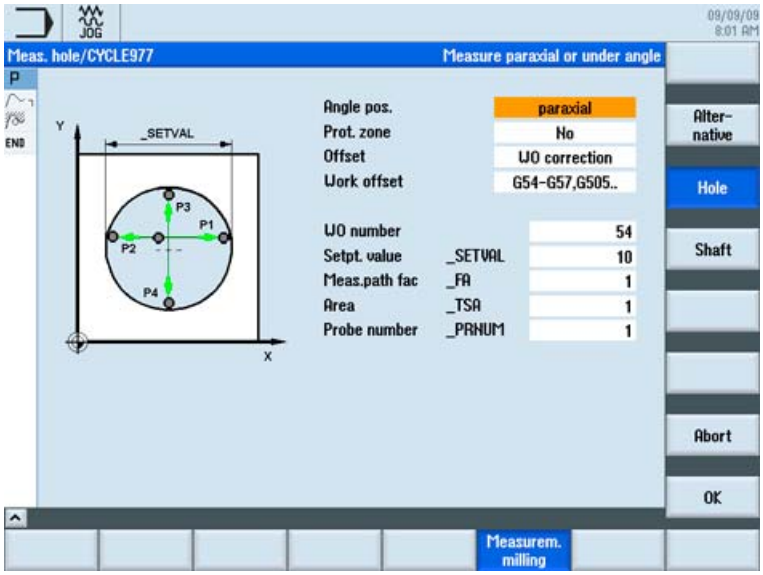
Highlights

- **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 Process measurements for workpieces and tools

☑ Option: Measuring cycles

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



The following measuring tasks can be made:

- 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 measuring variants are available:

- Hole, spigot, corner, rectangle, slot, bar, edge, face
- Measure under any surface angle
- Measure in swiveled machining planes
- Orienting the plane with three points

Highlights

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

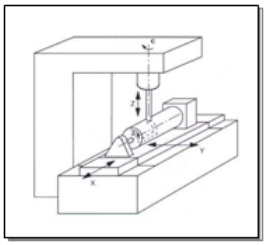
Complete machining

14

14.1 Peripheral surface machining (TRACYL)

☑ Option: Peripheral surface transformation

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

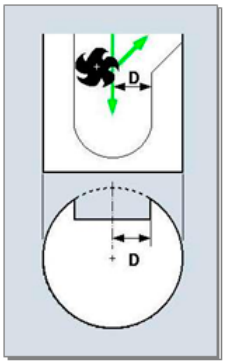
The axis behaves like a Y axis while programming in the run-off. All plane machining can also be executed in the run-off.

- 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.



Highlights

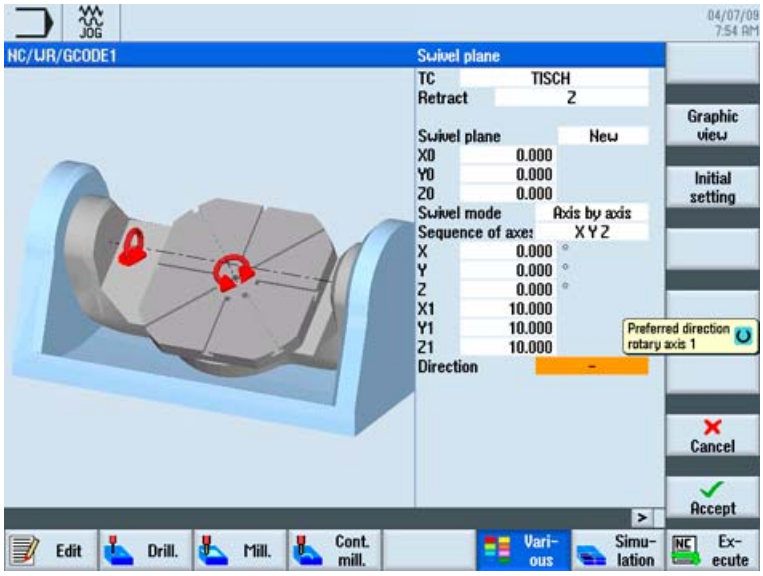
- **Additional business through expansion of workpiece spectrum**
- **Reduction of set-up times by complete machining on one machine**

14.2 Swivel machining plane

☑ Basic configuration

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

- A prerequisite is that the machine is equipped with a swivel device (swivel head and/or swivel table).
- The swivel cycle is available for sequence programming and 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



Highlight

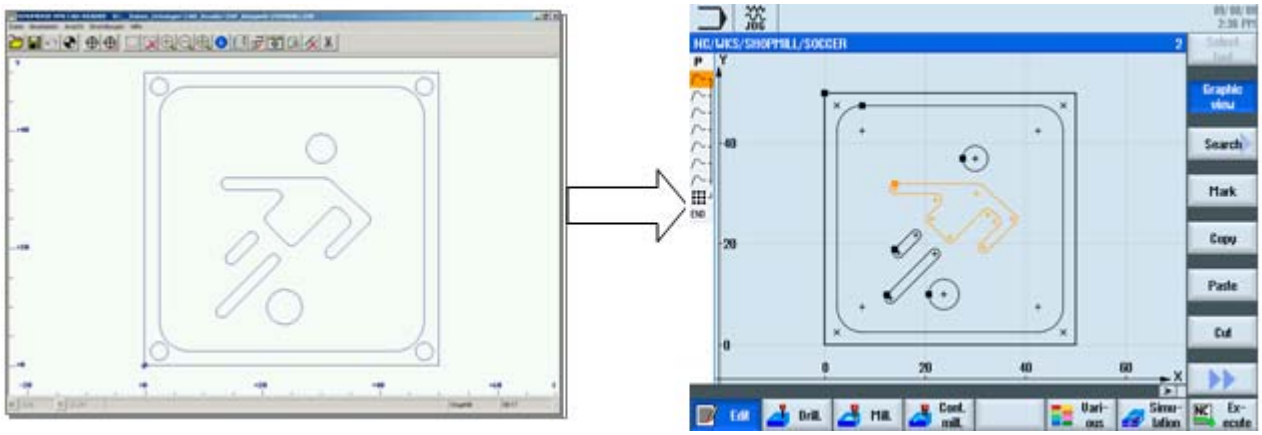
- **Programming with standard cycles and easy transformation on the inclined plane through the swivel cycle**

PC software

15

15.1 CAD reader for PC

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



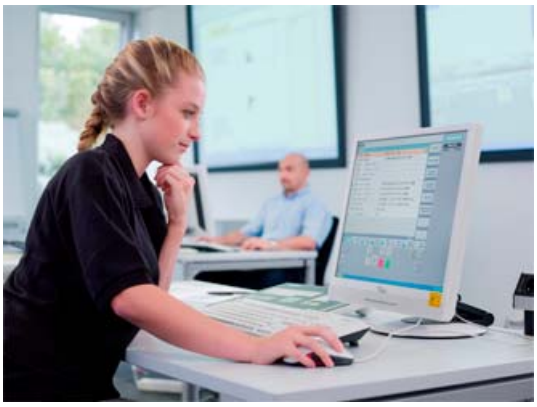
Highlight

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

15.2 SinuTrain

SinuTrain on your PC behaves in exactly the same way as your SINUMERIK 828D on the machine. This allows you to prepare part programs on the PC without having to occupy the machine. In addition, SinuTrain is an ideal training system for CNC training.

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



Highlight

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

15.6 Computer-based training

Multi-media initial study of milling technology.

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

Highlight

- **Graphically supported instruction software for beginners**

Option list for the SINUMERIK package 16

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

Programming support	
ShopMill machining step programming	6FC5800-0AP17-0YB0
Residual material identification and machining for contour pockets	6FC5800-0AP13-0YB0
Simulation	
3D simulation, machined part	6FC5800-0AP25-0YB0
Simultaneous recording (real-time simulation of current machining)	6FC5800-0AP22-0YB0
Tools	
Replacement tools for tool management	6FC5800-0AM78-0YB0
Transformations	
Peripheral surface transformation	6FC5800-0AM27-0YB0
Measuring functions/measuring cycles	
Measuring cycles for drilling/milling and turning (calibrate workpiece probe, workpiece measurement, tool measurement)	6FC5800-0AP28-0YB0
Extended operator functions	6FC5800-0AP16-0YB0
Communication/data management	
Controlling up to 4 additional drives using Ethernet	6FC5800-0AP01-0YB0
Languages	
Additional languages for the HMI sl operating software, without license, e.g. Danish, Finnish, Dutch, Polish, Romanian, Russian, Swedish, Slovakian, Czech, Turkish and Hungarian	On request
Diagnostic functions	
RCS Host remote diagnostics function	6FC5800-0AP30-0YB0
RCS Commander (viewer function) RCS Commander for PC/PG (on CD-ROM, included in the scope of delivery of the 828D)	

Summary of the highlights

17

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

Compact

✓ Maximum performance from the smallest possible dimensions

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

Strong

✓ The most powerful CNC functions

- 80bit NANO^{FP} accuracy for the maximum precision in 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

✓ Simple operation & programming

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

Index

R

S

T

U

V

W