SIEMENS

SINUMERIK

810D/840D powerline Milling with ShopMill

Control system overview for machine tools' sales people



Preface	
Introduction	1
System overview	2
Setup functions	3
Tool management	4
Program management, user memory	5
Simulation	6
Mold making	7
DIN/ISO programming	8
Graphical program input	9
Complete machining	10
In-process measuring	11
PC software	12
Option list for the SINUMERIK package	13
Summary of unique selling points	14

Safety Guidelines

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 device/system may only be set up and used in conjunction with this documentation. Commissioning and operation of a device/system may only be performed by **qualified personnel**. Within the context of the safety notes in this documentation qualified persons are defined as persons who are authorized to commission, ground and label devices, systems and circuits in accordance with established safety practices and standards.

Prescribed Usage

Note the following:

!WARNING

This device may only be used for the applications described in the catalog or the technical description and only in connection with devices or components from other manufacturers which have been approved or recommended by Siemens. Correct, reliable operation of the product requires proper transport, storage, positioning and assembly as well as careful operation and maintenance.

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.

Preface

Scope of validity

This document provides you with an overview of the functional range of ShopMill version 6.4, in use with the SINUMERIK 810D powerline or SINUMERIK 840D powerline.

The document is oriented towards vendors and dealers of machine tools.

Organization of information

- Of the varied functional possibilities of this SINUMERIK product, we only designate those qualities which are of direct value to the machine's 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: ...

- A summary of the unique selling points of ShopMill in comparison with competitors may be found in Chapter "Summary of unique selling points".
- 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 Sales

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

Web: http://www.maschinenhersteller.com

Service

Phone: +49 xxx xxx Fax: +49 xxx xxx

Email: xxx@maschinenhersteller.com

Web: http://www.maschinenhersteller.com

Homepage:

http://www.maschinenhersteller.com

Contact person at Siemens

Sales & Marketing / Specialist Support

Phone: +49 xxx xxx Fax: +49 xxx xxx

Web: http://www4.ad.siemens.de (worldwide)

Service

Phone: +49 xxx xxx Fax: +49 xxx xxx

Web: http://www4.ad.siemens.de (worldwide)

Homepage:

Visit the JobShop internet portal: http://www.siemens.com/jobshop

Table of contents

Prefac	e	3
Introdu	uction	9
1.1	Application	g
1.2	Machine spectrum	g
Systen	n overview	11
2.1	SINUMERIK 810D powerline	11
2.2	SINUMERIK 840D powerline	12
2.3	Operator panel fronts	13
2.4	Operator panel equipment	17
Setup	functions	19
3.1	Measure workpiece	19
3.2	Work offsets	20
3.3	Measure tool	20
3.4	TSM universal cycle	21
3.5	Positioning cycle	21
3.6	Face milling cycle	22
3.7	Swiveling in setup mode	22
Tool m	nanagement	23
4.1	Tool table	23
4.2	Tool monitoring, sister tools	24
4.3	Manual tool change	25
Progra	ım management, user memory	27
5.1	Program manager	
5.2	User memory and data handling	28
5.2.1	Buffered CNC work memory	28
5.2.5 5.2.6	Disk drive	
	Introdu 1.1 1.2 Syster 2.1 2.2 2.3 2.4 Setup 3.1 3.2 3.3 3.4 3.5 3.6 3.7 Tool m 4.1 4.2 4.3 Progra 5.1 5.2,5 5.2,4 5.2,5	1.2 Machine spectrum. System overview

6	Simula	ation	31
	6.1	3D simulation	31
	6.2	Quick view	32
	6.3	Simultaneous recording	32
7	Mold m	naking	33
	7.1	ShopMill applicability	33
	7.2	High speed settings	34
	7.3	Look Ahead	35
	7.4	Jerk limitation	36
	7.5	Dynamic feedforward control	37
	7.6	Online compressor	38
	7.7	CIP interpolation	39
	7.8	Block change times	39
	7.9	Block search	40
	7.10	Program control	41
8	DIN/IS	O programming	43
	8.1	DIN/ISO scope of language	43
	8.2	DIN/ISO editor / G code editor	44
	8.3	Contour calculator (DIN/ISO)	45
	8.4	Machining cycles	46
9	Graphi	ical program input	47
	9.1	Overview	47
	9.2	Dynamic broken-line graphics	48
	9.3	Online help	49
	9.4	Standard machining steps	50
	9.5	Path milling cycle	51
	9.6	Basic block display	51
	9.7	Contour calculator	52
	9.8	Contour pockets and spigots	52
	9.9	Residual material detection	53
	9.10	Multiple clamping	53
10	Comple	lete machining	55
	10.1	Swivel machining plane	55
	10.2	Five-axis machining package (TRAORI)	56
	10.3	Peripheral surface machining (TRACYL)	57
11	In-proc	cess measuring	59

12	PC sof	itware	61
	12.1	CAD reader for PC	61
	12.2	ShopMill on PC, SinuTrain	62
	12.3	ShopMill for self-study	62
13	Option	list for the SINUMERIK package	63
14	Summ	ary of unique selling points	65
	Index		67

Introduction

1.1 Application

ShopMill is a customized technology package for all standard CNC milling machines and machining centers.

ShopMill enables the simple operation of machine tools, supporting all operator actions with graphic help displays.

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

ShopMill offers three different options for programming:

The DIN/ISO editor is used to create DIN/ISO programs on the machine and correct externally created DIN/ISO programs.

The work plan editor is used for graphic programming on the machine. Here are the typical applications for finishing individual parts and small series. For example, DIN/ISO sections may be flexibly inserted for complex part machining steps.

ShopMill offers a uniform control configuration which covers all required areas of application without additional start-up and training costs:

- Easy-to-use interface for all machine functions
- DIN/ISO programming offline via CAD/CAM system (e.g. mold making applications)
- DIN/ISO programming on the machine
- Graphic programming
- · Measuring functions for workpieces and tools

1.2 Machine spectrum

ShopMill is perfectly suited for the following types of machines:

- Vertical machining centers with three geometry axes
- · Vertical machining centers with additional reversible clamping device
 - Machining cylindrical workpieces (peripheral surface transformation)
 - Machining on multiple sides (swiveled plane)
- Vertical machining centers with additional swiveling device
 - Swivel head, swivel table or mixed kinematics

System overview

2.1 SINUMERIK 810D powerline

The SINUMERIK 810D powerline is a fully digital CNC controller for standard machine tools and handling tasks.

- Three integrated power modules (optionally expandable to 6 power modules)
- Digital drive control
- Up to 6 axes, including a maximum of 1 spindle and 1 auxiliary spindle
- Up to 2 machining channels
- NC interpreter for DIN and high-level language elements
- Intelligent control functions for complex machining tasks

You can find further information in catalog NC 60





1PH7 spindle motor



1FK7 servo motors



- Cost-effective thanks to integrated power modules
- Very dynamic due to digital drive control
- Can be used for a wide variety of applications due to the use of a large number of CNC functions
- Space-saving design for even the smallest control cabinets

2.2 SINUMERIK 840D powerline

The SINUMERIK 840D powerline is a modular CNC for complex machine tools, mold making, rotary indexing machines and transfer lines.

- · Digital drive controller
- Modular design for drive controller and power unit
- Up to 31 axes/spindles in up to 10 processing channels
- NC interpreter for DIN and high-level language elements
- Intelligent control functions meeting the highest standards of machining technology
- You can find further information in catalog NC 60





1PH7 spindle motor



1FT6 servo motors



- Flexible application and guaranteed future through modular drive design and comprehensive CNC functions
- Extremely high dynamics, precision and surface quality through use of high performance processors
- High degree of expandability for complex production machines

2.3 Operator panel fronts

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

OP 010C

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





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

OP 010

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





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

OP 010S

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





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

OP 015

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





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

2.4 Operator panel equipment

The ShopMill user interface can be installed on either of the following PC modules:

PCU20

- Fewer parts for specifically flat design
- Competitive purchase price and inexpensive replacement parts
- CNC memory expandable using Compact Flash cards



Highlights



PCU50

Rugged and cost-optimized solution, as there is no hard disk

For customers who require a hard disk and a Windows operating system, we offer the PCU 50.

- Windows XP operating system
- · Additional CNC memory through hard disk
- · Additional PCI slots on board





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

Setup functions

3.1 Measure workpiece

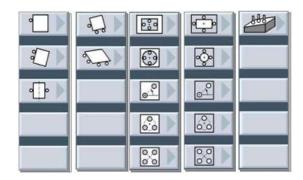
☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Basic configuration

The workpieces can be measured as follows:

- edge finder, dial gage, reference tool
- 3D switching probe

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

- Point measurement for edges
- Orienting the edge (angle)
- Inner/outer corner (3 or 4 points)
- Orienting the edge by means of 2 boreholes/spigots
- Inner / outer rectangle / circle / spigots
- Orienting the plane with three points





- Time saving due to user-friendly determination of the workpiece's clamping position instead of orienting the workpiece by hand
- Fast zero point definition by scratching at the opposite edges of the workpiece and automatically calculating the center

3.2 Work offsets

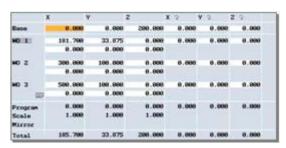
☑ 810D ☑ 840D

☑ PCU 20 ☑ PCU 50

☑ Basic configuration

The following settable zero offsets are available in ShopMill:

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



Highlights



- Flexible machining due to great number of settable zero offsets
- Plus the unlimited possibilities of programmable zero offsets

3.3 Measure tool

☑ 810D ☑ 840D

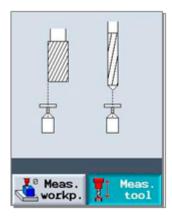
☑ PCU 20 ☑ PCU 50

☑ 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 TSM universal cycle

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ 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 zero offsets



Highlight



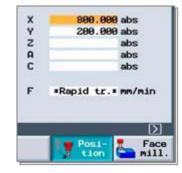
• Take over and change in tools directly from the tool table

3.5 Positioning cycle

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Basic configuration

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

- Linear axes
- · Rotary axes
- Feedrate/rapid traverse



Highlight



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

3.6 Face milling cycle

☑ 810D ☑ 840D

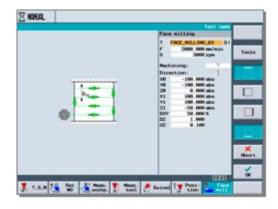
☑ PCU 20 ☑ PCU 50

☑ Basic configuration

The blank can be milled over before machining in the setup.

The following parameters can be specified:

- · Machining strategy and direction
- Machining limitations



Highlight



• Preparation of workpiece without having to create a part program

3.7 Swiveling in setup mode

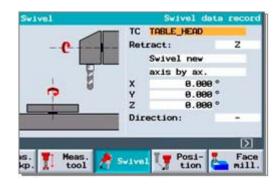
☑ 810D ☑ 840D

☑ PCU 20 ☑ PCU 50

☑ Basic configuration

The machining plane can be swiveled any way in the setup mode:

- · Machining inclined surfaces
- · Measure with inclined tool or table
- · Swivel axis by axis or direct



Highlight

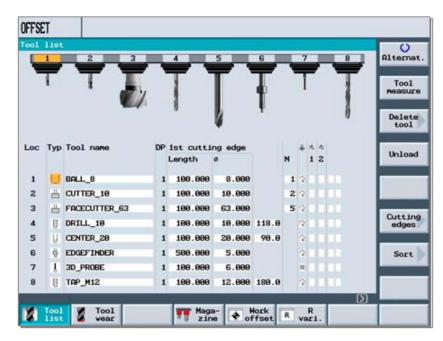


• Swivel the machining plane in setup mode by dialog

Tool management

4.1 Tool table

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Basic configuration



Tools with their complete information may be managed in the tool list. Each tool is presented on scale with its diameter in a type-specific display.

(The maximum number of tools is defined by the machine manufacturer.)

Tools are assigned to the desired magazine locations with the load function.

The following data can be stored for each tool:

- Tool type: Milling cutter, 3D milling cutter, drill, face milling cutter, threading tap, angle head mill, centerer, edge finder, 3D probe
- Clear tool name in plaintext, example: MESSERKOPF_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

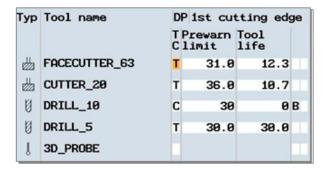


- All tool data at a glance
- Simple and secure handling via unmistakable tool names

4.2 Tool monitoring, sister tools

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Basic configuration

ShopMill offers efficient tool management for the activation of replacement tools.



- Monitor cutting time (T) or number of exchanges (C)
- Prewarning limit for timely preparation of new tools
- Automatic exchange of sister tools for automatic operation possible (duplo number DP)



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

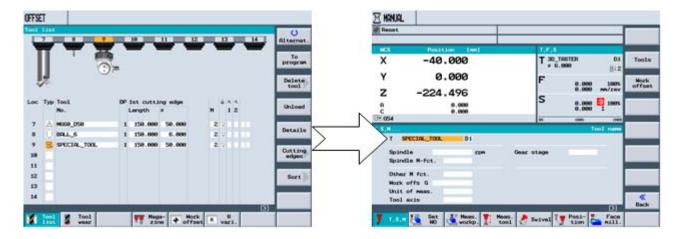
4.3 Manual tool change

☑ 810D ☑ 840D

☑ PCU 20 ☑ PCU 50

☑ Basic configuration

Ultraprecision tools, cable-bound probes or adapter heads for an additional spindle can be manually changed in the spindle. ShopMill provides the necessary operator dialog.



Highlight

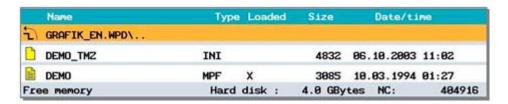


• Easy handling of special and manual tools

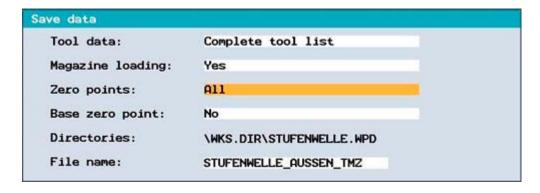
5.1 Program manager

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Basic configuration

The program manager supports plaintext file names up to 24 characters.



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





- Part programs can be created, retrieved and saved within a very short time
- User-friendly data handling in typical PC style with copying, pasting, renaming, etc.

5.2 User memory and data handling

5.2.1 Buffered CNC work memory

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50

SINUMERIK 810D powerline Basic configuration: 0.5 MB,

expandable to 2.5 MB

SINUMERIK 840D powerline Basic configuration: 3 MB, NCU 571.5, NCU572.5, NCU 573.5 expandable to 6 MB

Highlight



• Exceptionally large storage space in basic configuration already

5.2.2 USB memory stick

☑ 810D ☑ 840D □ PCU 20 ☑ PCU 50 ☑ Basic configuration

(USB memory stick required)

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

User data stored on the PC can be quickly transferred to the internal CNC memory. For security reasons executing from the USB memory stick is not possible. In this regard use the possibility of execution from the hard disk.





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

5.2.3 CompactFlash card

☑ 810D ☑ 840D ☑ PCU 20 ☐ PCU 50 ☑ Option: Management of network and disk drives

A CompactFlash card can be used to expand the CNC memory of the PCU20.

The amount of additional memory capacity depends on the CompactFlash Card used (maximum 1 GB).

The CompactFlash Card is not included in the PCU20 scope of delivery!

Highlight

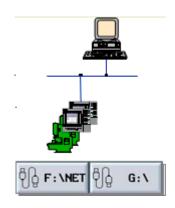


• All of the relevant user data is available immediately

5.2.4 Ethernet networking

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Option: Management of network and disk drive

The PCU20 and PCU50 come ready for Ethernet (RJ45 connection). The data transfer rate is 10 / 100 Mbit/s. Access to the network drive is available directly from the ShopMill program manager. No additional software is required on the server.



Highlight



 Easy and economical connection via Ethernet (TCP/IP) to Windows PCs or Unix workstations

5.2.5 Hard disk drive

☑ 810D ☑ 840D □ PCU 20 ☑ PCU 50 ☑ Basic configuration

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

Highlights



Hard disk can still be used as a data carrier

5.2.6 Disk drive

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Option: Management of network and disk drive

PCU 20 and PCU 50 are prepared for connection to a USB disk drive as standard. Access to the disk drive is available directly from the ShopMill program manager.

The disk drive is not supplied with the product.



Highlight



• Diskette can still be used as a data carrier

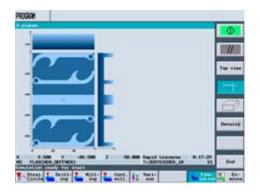
Simulation

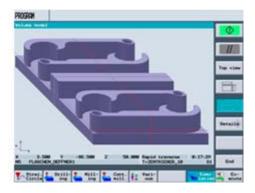
6.1 3D simulation

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Basic configuration

The simulation integrated in ShopMill offers an optimum process reliability. This is guaranteed as it is calculating with exactly the same tool geometry as for the actual machining process. The simulation can be controlled by start, stop and reset softkeys. For critical machining, the simulation can be operated in single block mode and controlled by the feed override. Possible forms of display:

- Top view, side view
- Dynamic 3D volume model with sectional planes





Simulation (3-side view) Simulation (3D volume model)



- Zoom into Details possible without another simulation
- Automatic calculation of machining time

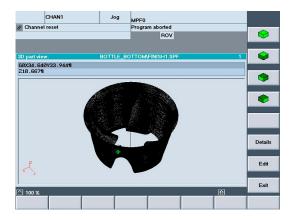
6.2 Quick view

 ☑ PCU 50

☑ Basic configuration

The quick view is available for particularly large part programs.

The simulation automatically changes between standard and quick view - depending on the size of the program.



Highlights



• More safety when handling mold making programs

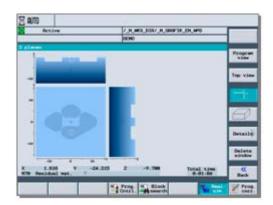
6.3 Simultaneous recording

☑ 810D ☑ 840D

☑ Option: Simultaneous recording (simulation of current machining)

During machining, the tool paths can be simultaneously recorded on the display of the controller in three-side view or in 3D view.

 Workpiece graphics and views correspond to the graphic simulation



Highlight



• Machining can also be monitored in a complex machine room

Mold making

7.1 ShopMill applicability

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Basic configuration

The ShopMill technology package enables the machining of mold making applications. Convenient setup functions such as workpiece or tool measurement can also be used for this application.

Mold making programs are usually generated by CAD/CAM systems as DIN/ISO programs. For this reason, the programs are edited with the DIN/ISO editor in ShopMill. Work plan programming is not typically used for mold making programs.

Depending on the size (e.g. 100MB), mold making programs are mostly archived on a network PC for external execution, on the Compact Flash Card of the PCU20 or on the PCU 50 hard disk.

Highlights



• One operator interface for all workshop applications

7.2 High speed settings

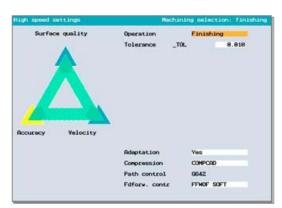
☑ 810D ☑ 840D

☑ PCU 20 ☑ PCU 50

☑ 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 started in the DIN/ISO editor
- For simplification, the selection is limited to the roughing, prefinishing, finishing machining types (or deselection) and the tolerance band from the CAD/CAM system.
- All other values such as compression, feedforward control and jerk limitation are preprogrammed by the machine manufacturer and password-protected.



Highlight



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

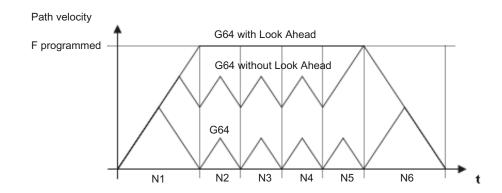
7.3 Look Ahead

☑ 810D ☑ 840D

☑ PCU 20 ☑ PCU 50

☑ Basic configuration

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



Highlight

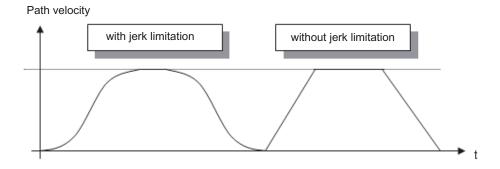


Shorter machining times through optimum velocity control

7.4 Jerk limitation

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ 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.





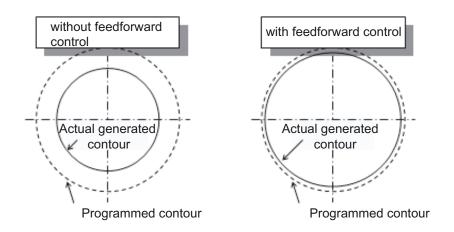
- Longer machine lifespan through protection of the mechanical components
- Higher path accuracy through softer acceleration

7.5 Dynamic feedforward control

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ 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



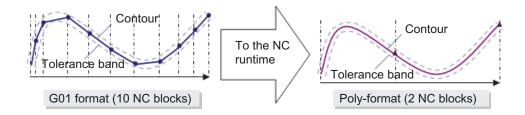
Higher path accuracy through compensation of contouring errors

7.6 Online compressor

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Option: Spline interpolation (compressor)

Freeform programs in G01 format are converted to the internal control spline format during NC runtime. The number of traversing blocks is thereby drastically reduced. With an active compressor, free-form surfaces can be machined at a higher speed and without reaching the block change limit.

Example:





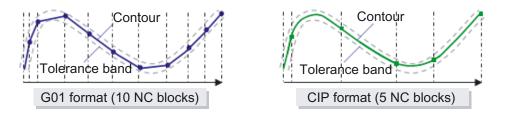
- Higher machining speed through quasi-shortening of block change times
- Better surface smoothing through splines instead of linear blocks
- Compressor now comes in a cost-efficient three-axis solution, in addition to the five-axis solution

7.7 CIP interpolation

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Basic configuration

The CNC offers the possibility of programming circles with start, intermediate and end points (Circle Intermediate Point). If the intermediate and end points lie outside of the machining plane, then this results in spacious circular paths (free-form surfaces). Special "afterburner" software solutions on a separate PC can transform G01 programs into CIP format. The number of NC blocks is greatly reduced for consistent path accuracy.

Example:



Highlight



Higher machining speed through shortening of block change times

7.8 Block change times

The classic block cycle time plays a secondary role in the SINUMERIK 810D and/or SINUMERIK 840D 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 computational block change time.

This is demonstrated in a concrete test situation: A SINUMERIK 840D NCU 573.5 with an active online compressor processes 10,000 G01 blocks in approx. three seconds. This corresponds to a block change time of approx. 0.3 msec.

The maximum machining velocity stems directly from processing the polynomials. The CNC block change time is no longer meaningful, as the machining velocity is itself limited by the machine's mechanical properties and the technical boundary conditions of the chip removal process.

Highlight



Internal shortening of block change times by reducing amount of data

7.9 Block search

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ 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



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

7.10 Program control

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Basic configuration

Single block

Single block mode may be activated when running-in a 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).

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 machining steps. 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.



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

DIN/ISO programming

8.1 DIN/ISO scope of language

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Basic configuration

The CNC interpreter of the SINUMERIK controller 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 Functions

G0, G1, G2, G71 ...

• Language commands (expanded G functions)

CIP, SOFT, BRISK, FFWON ...

• Frame operations (programmable zero offsets)

The workpiece coordinate system can be shifted, scaled, mirrored or rotated with the commands TRANS, SCALE, MIRROR, ROT.

• R parameters (arithmetic parameters)

100 predefined R parameters are available as arithmetic parameters (floating comma 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 zero offsets, tool offsets, axis positions, measurement values, control conditions etc.

Arithmetic operations

Arithmetic operations are available to combine the variables:

- Computational 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 ...



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

8.2 DIN/ISO editor / G code editor

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Basic configuration

ShopMill can accept direct input of DIN/ISO programs.

To this end, a line-oriented DIN/ISO editor is available.

The DIN/ISO editor enables one to directly enter or edit CNC language commands. Thereby, the complete range of CNC functions are available for the most complex machining.

The following possibilities arise:

- Creation of DIN/ISO programs at the control
- Edit externally rendered DIN/ISO programs (Ex.: mold making programs)



The following functions are included in the DIN/ISO 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
- Renumber program
- Execute directly from any NC program block (block search)
- Jump to program start or program end

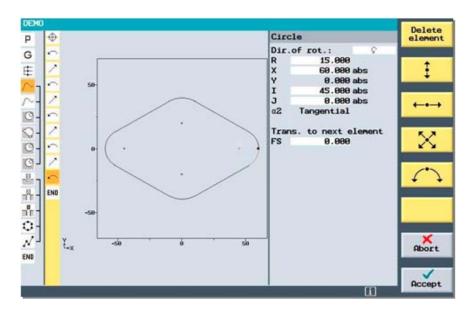


- Save time by programming with efficient DIN/ISO editor
- Even large part programs allow extremely fast editing in MB size

8.3 Contour calculator (DIN/ISO)

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Basic configuration

A contour calculator is available in the DIN/ISO editor as well as in the work plan editor.



The following contours may be generated with this geometry calculator:

- Contours for path milling
- Pocket and island contours
- Groove contours for peripheral surface machining

The contour calculator has the following characteristics:

- Up to 250 geometry elements programmable
- Input of Cartesian coordinates; polar coordinates also possible with graphical programming
- Automatic calculation of unknown elements ("drawing dot to dot")
- Chamfer or radius programmable in contour transitions
- · Separate feedrate for individual contour elements; chamfer and radius programmable
- Conversion of workpiece drawings in DXF format with CAD Reader for PC and further processing with the contour calculator in the control.



- Easy input of workpiece geometry: Drawing dot to dot
- Determination of contours with many unknown intermediate values is possible
- Speedy and certain from drawing to finished workpiece

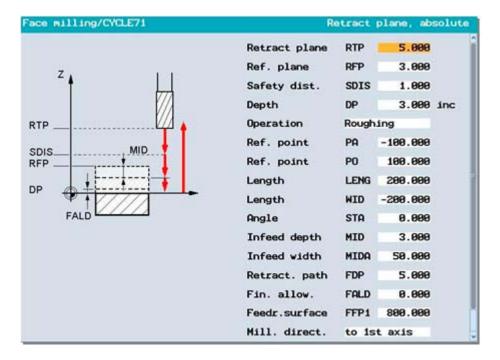
8.4 Machining cycles

☑ 810D ☑ 840D

☑ PCU 20 ☑ PCU 50

☑ Basic configuration

Machining cycles for standard machining, turning, milling and drilling may be called up through the DIN/ISO editor. Input masks with dynamic help displays are available for the convenient input of machining parameters.



The following machining cycles are available:

Milling

Face milling, path milling, circular and rectangular pockets, circular and rectangular spigots, grooving, thread milling, engraving

- Drilling
 - Centering, drilling, boring, reaming, deep-hole drilling, tapping
- Drill patterns
 - Line, grid, circle
- Swivel plane
- High-speed settings

Highlight



 Graphical support of cycles as an expansion of the highly flexible DIN/ISO programing Graphical program input

9

9.1 Overview

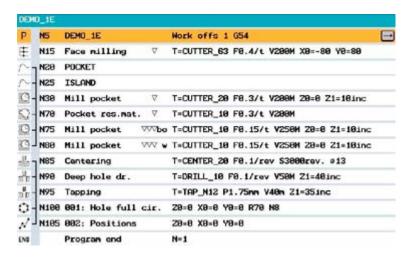
☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Option: Machining step programming

The graphical programming is executed via a graphic, interactive machining step editor. Each program line represents a technological machining step (ex.: face milling, centering, drilling, tapping) or geometric data required for the machining steps (pattern positions or contours). Machining steps belonging together are linked to each another. The interlinked machining steps are executed consecutively at the appropriate contours or position patterns.

Graphical programming offers, in comparison to DIN/ISO programming, a compact and comprehensible program overview.

Entering individual machining steps requires no knowledge of DIN/ISO. All required technical and geometric parameters are entered in screen forms.

Simple, intuitive programming with machining steps can always be expanded very flexibly by inputting DIN/ISO blocks and control functions.





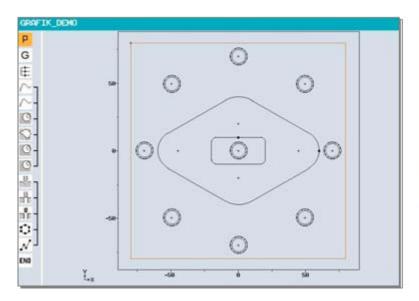
- Intuitive program input, without knowledge of DIN/ISO or Operating Manual
- Compact, clearly arranged machining program
- Reduction of programming time through graphical input screens, copying / inserting and linking machining steps

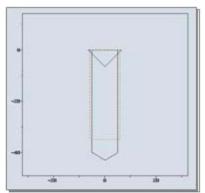
9.2 Dynamic broken-line graphics

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Option: Machining step programming

During the full programming period, the previously entered machining steps will be continuously displayed to scale. A simulation is not required for this.

- Plan view of workpiece
- Front view for linked drilling operations





Highlights

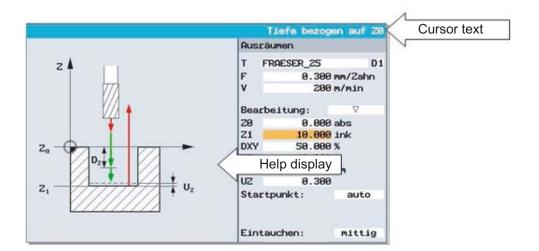


• More certainty during program input thanks to quick review of the resulting contour

9.3 Online help

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Option: Machining step programming

All input parameters are clarified with dynamic help displays. In addition, explanatory text is displayed. This enables creation of programs on the machine without a programmer handbook.





With the Help key you can toggle between the work plan and programming graphics as well as between the parameterization screen form with programming graphics and the parameterization screen form with the help display.



- Programming on the machine without a handbook
- Help key to toggle between the help displays

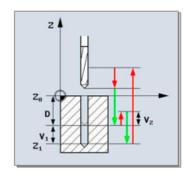
9.4 Standard machining steps

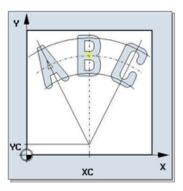
☑ 810D ☑ PCU 20 ☑ PCU 50 ☑ Option: Machining step

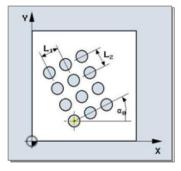
programming

A number of different machining operations are available in the machining step editor:

- **Drilling operations**
 - Centering / drilling / reaming
 - Deep-hole drilling
 - Boring
 - Tapping with and without chip breaking
 - Thread milling / drill and thread milling
- Milling operations
 - Face milling
 - Rectangular pockets and spigots
 - Circular pockets and spigots
 - Longitudinal / circular groove
 - Contour milling, path milling
 - Chamfering of any milling geometry
- Engraving cycle
 - Any text with special characters
 - Date, time, workpiece count, variable
- Position pattern (milling plane or peripheral surface)
 - Position list
 - Line / grid / frame (with shear angles)
 - Pitch circle/full circle
 - Hide individual positions
- Basic elements
 - Straight / line / helix
 - Cartesian / polar
- Miscellaneous
 - Marks/repetitions
 - Swivel plane
 - Peripheral surface transformation







Drilling and milling operations may be linked any way with position patterns.



- Gains in time and safety since contours or position patterns need only be entered or changed once when linking the machining steps
- Intuitive program input, as everything is at a glance complete with tool, feedrate and spindle speed or cutting rate

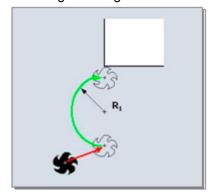
9.5 Path milling cycle

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Option: Machining step programming

A high-performant path milling cycle is available for path machining of milling contours:

The following operations are available:

- Milling with and without radius compensation
- Traversing in quarter circle, half circle, direct or straight
- · Reversing the machining direction
- Machining open or closed contours
- · Chamfering machining strategy



Highlight



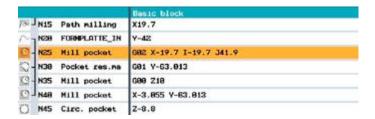
Machining of any milling contours with intelligent additional functions

9.6 Basic block display

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Basic configuration

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

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



Highlights



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

9.7 Contour calculator

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Basic configuration

A contour calculator is available for machining step programming, just as for DIN/ISO programming

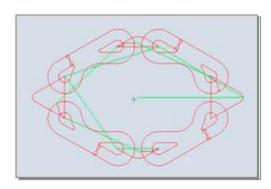
9.8 Contour pockets and spigots

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Option: Machining step programming

ShopMill offers a stock removal cycle for processing contour pockets and spigots. This cycle automatically calculates the required tool paths - even for complex pocket geometries. Up to 12 islands can be considered for each slot. The slot and island contours can be entered with the contour calculator directly into the control.

The following operations are available:

- · Stock removal / complete machining
- · Machining of contour spigots
- Edge/base finishing
- Direct or helical plunging
- Manual or automatic allocation of insertion point
- · Optional predrilling at plunging point



Highlight



Also program and machine complex contour pockets within a short time

9.9 Residual material detection

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Option: Residual material detection and machining

mit milling with large diameters are automatically

Contour ranges which do not permit milling with large diameters are automatically recognized in the cycle for contour pockets and contour pocket cycles. The operator can directly rework these ranges using a suitable smaller tool.

Highlight



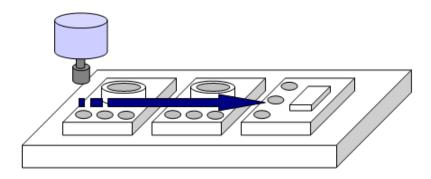
 Shorter machining times through the use of a large tool for the substantial part of the stock removal and a smaller tool for the remaining residual material

9.10 Multiple clamping

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Option: Machining step programming

Several identical workpieces can be clamped onto the machine table. With the "Multiple clamping" function, an entire program is generated from the required individual machining in the graphic program. The machining steps are sorted in this program so that the number of tool changes (and tthus he idle times) is reduced to a minimum. The number of possible workpieces corresponds to the number of available settable zero offsets.

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Option: Multiple clamping of various workpieces



This function allows similar and different workpieces to be finished on multiple vises or gripping yokes, while saving time.

Highlight



Reduction of secondary machining times through optimization of tool sequence

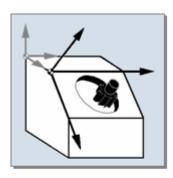
Complete machining 1

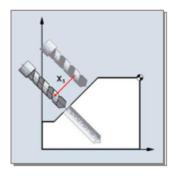
10.1 Swivel machining plane

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ 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 machining step programming and DIN/ISO programming.
- The planes can swivel axially or according to a specified projection angle.
- 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.





Highlight



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

10.2 Five-axis machining package (TRAORI)

□ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Option: Five-axis machining package

In contrast to static transformations (swiveling) in which the tool is perpendicular to the machining plane, the five-axis machining package TRAORI allows the dynamic coupled motion of a tool along the workpiece surface. It is used for five-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)





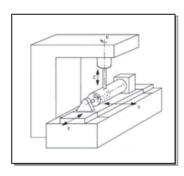
- Shorter machining time while hobbing with face cutters
- Machining the most complex contours, such as turbine blades, for example.

10.3 Peripheral surface machining (TRACYL)

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ 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

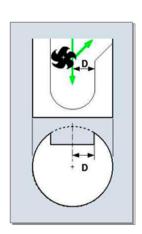
The axis behaves like a Y axis while programming in the run-off. All plane machining can also be executed in the run-off.

- 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.



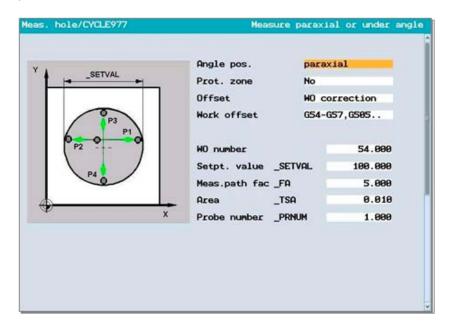


- Additional business through expansion of workpiece spectrum
- Reduction of set-up times by complete machining on one machine

In-process measuring

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Option: Measuring cycles

Measuring cycles are available in the DIN/ISO editor for measuring tasks in automatic mode. 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 zero point offsets, correction of tool geometry or only measuring
- Tool measurement: Correction of tool geometries
- Display measurement results
- Log measurement results

The following measuring variants are available:

- Borehole, spigot, corner, rectangle, groove, bar, edge, face
- Measure under any surface angle
- Measure in swiveled machining planes
- Orienting the plane across three points



- 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

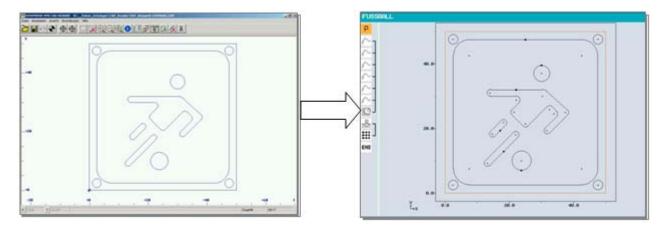
PC software 12

12.1 CAD reader for PC

☑ 810D ☑ 840D

☑ PCU 20 ☑ PCU 50 ☑ Option: 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



• Time saving by conversion of contours and position patterns from DXF files

12.2 ShopMill on PC, SinuTrain

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Option: SinuTrain ShopMill

PC system with identical controls for work preparation and CNC training

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



Highlight



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

12.3 ShopMill for self-study

☑ 810D ☑ 840D ☑ PCU 20 ☑ PCU 50 ☑ Option: ShopMill for self-study

Multimedia introduction to milling technology with ShopMill.

Programming exercises with guided examples

Highlight



• Graphically supported instruction software for beginners

Option list for the SINUMERIK package

13

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

Additional ShopMill functions	
Machining step programming	6FC5463-0FA21-0AA0
Residual material identification and machining for contour pockets	6FC5463-0FA01-0AA0
Multiple clamping of various workpieces	6FC5463-0FA04-0AA0
Simultaneous recording (real-time simulation of current machining)	6FC5463-0FA02-0AA0
Memory expansion and networking	
User memory expanded by 1MB each1)	6FC5251-0AD02-0AA0
Network drive management via Ethernet 1)	6FC5463-0FA03-0AA0
Hard disk PCU50 instead of PCU20 1), 2)	
Disk drive 1), 2)	
CNC additional functions	
Spline interpolation (compressor) for three-axis machining 1)	6FC5251-0AF14-0AA0
Spline interpolation (compressor) for five-axis machining 1)	6FC5251-0AA14-0AA0
Polynomial interpolation	6FC5251-0AA15-0AA0
Peripheral surface transformation ¹⁾	6FC5251-0AB01-0AA0
Drilling and milling measuring cycles for workpiece and tool measurement ¹⁾	6FC5250-0BX00-0AB0
Machining package five axes ^{1), 2)}	
PC software	
SinuTrain ShopMill, without CAD Reader (CD-ROM)	6FC5463-0FA40-0AG0
SinuTrain ShopMill, with CAD Reader (CD-ROM)	6FC5463-0FA41-0AG0
CAD Reader for PC (CD-ROM)	6FC5260-0AY00-0AG0
ShopMill for self-study (CD-ROM)	6FC5095-0AA71-0BG0
1) Notice: Start-up costs necessary.	

2) Please contact the machine manufacturer.

Summary of unique selling points

14

The control and drive package from Siemens with SINUMERIK 810D powerline / SINUMERIK 840D powerline and ShopMill has the following prominent characteristics:

Especially in comparison to European control providers:

• Better result for workpieces

- Programmable jerk limiting and velocity feedforward control provide for a better workpiece surface in conjunction with extremely dynamic drives.
- Reliable parts quality through process measurement with automatic correction of the tool data or zero offsets

More flexibility with DIN/ISO programming

 Unbeatable command range with G codes, variables and high-level language elements.

A universal technology package from simple three-axis machining to five-axis mold making

- Lower costs for training machine operators
- In addition to ShopMill for milling, ShopTurn for turning offers the same type of operation and programming.

Especially in comparison to Asian control providers:

• User-friendlier interface

- Plaintext descriptions for part programs and tools offer greater transparency.
- Fast setup of tools and workpieces over a graphically interactive dialog.
- Machining step programming complete with specifications for the tool, feedrate and spindle speed or cutting rate; no entries required in DIN/ISO.

• Greater guarantee for the future via intelligent control functions

- Thanks to the linking of machining steps, the position patterns and contours need only be entered or changed once.
- Residual material identification and engraving cycle save time for programming and execution.
- Tool monitoring and sister tools for automatic operation as standard.

PC software enabling the machine more time available for production

- Converting workpiece drawings from DXF format to finished contours and position patterns for use in the part program, using the CAD Reader for PC.
- Controller-identical PC software SinuTrain for training and work preparation

Index

Α

Application, 9 Application Memory, 28 Arithmetic operations, 43 Arithmetic parameters, 43

В

Basic block display, 51 Block change times, 39 Block cycle time, 39 Block search, 40 Broken-line graphics, 48

C

CAD Reader, 61
CIP interpolation, 39
CNC memory, 28
CompactFlash Card, 29
Complete machining, 55
Compressor, 38
Contour calculator (DIN/ISO), 45
Contour calculator (graphical programming), 52
Contour pocket cycle, 52
Contour spigots, 52
Control structures, 43

D

DIN/ISO editor, 44 DIN/ISO language, 43 DIN/ISO programming, 43 Disk drive, 30 DXF converter, 61 DXF import, 45

Ε

Engraving cycle, 50 Ethernet, 29

F

Face milling cycle, 22 Feedforward control, 37 Five-axis machining package, 56 Flashcard, 29 Floppy disk drive, 30 FRAMES, 43

G

G code editor, 44 G Functions, 43 Graphic programming, 47 Groove side offset, 57

Н

Hard disk, 30 High speed settings, 34

1

In-process measurement, 59

L

Look Ahead, 35

М

Machine spectrum, 9
Machining cycles, 46
Machining steps, 47
Manual tool change, 25
Measure tool, 20
Measuring cycles, 59
Mold making, 33
Multi-face machining, 55

Ν

Networking, 29

0

Online help, 49 Operator panel equipment, 17 Operator panel fronts, 13 Operator Panels, 13 Option list, 63

Ρ

Path Milling Cycle, 51 PC software, 61 PCU 20, 17 PCU 50, 17 Peripheral surface machining, 57 Program control, 41 Program editing, 41 Program manager, 27 Program test, 41 Programmable zero offsets, 43

R

Replacement tools, 24 Repositioning to the contour, 41 Residual material detection, 53 RTCP, 56

S

Setup swivel cycle, 22 ShopMill on PC, 62 Simulation, 31
Simulation of current machining, 32
Simultaneous recording, 32
Single block, 41
SINUMERIK 810D powerline, 11
SINUMERIK 840D powerline, 12
SinuTrain, 62
Standard cycles, 46
Swivel plane, 55
System variables, 43

T

TCP/IP, 29
Tool life monitoring, 24
Tool management, 23
TRACYL, 57
Training CD, 62
TRAORI, 56
TSM universal cycle, 21

U

Unique selling points, 65 User variables, 43

V

Velocity control, 36

Ζ

Zero offsets, 20