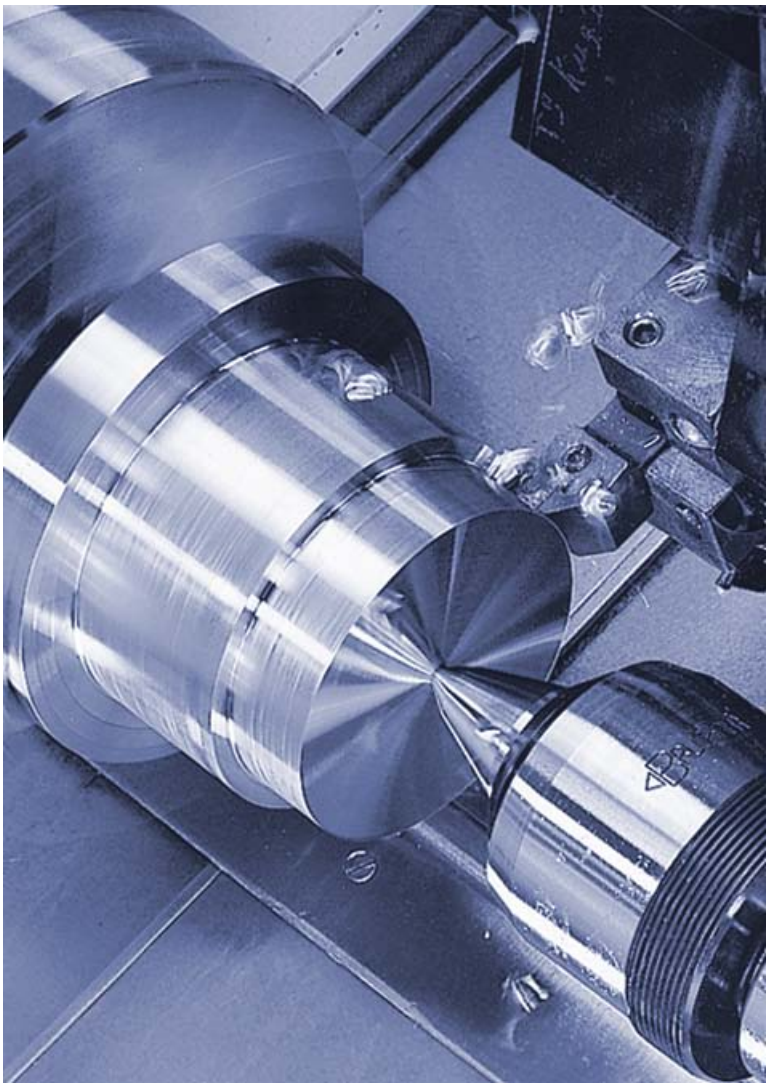
SIEMENS

SINUMERIK 802D sl plus

Turning

Control system overview
for machine tools' sales people



03/2011

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

℗ 03/2011

Foreword

Scope of validity

This document provides you with an overview of the range of functions included in the operator panel control **SINUMERIK 802D solution line plus** Version 1.4 for turning machines.

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

Structure of the information

- From the wide variety of functions of the SINUMERIK products, only those that are of direct significance to the machine user are listed.
- All functions contained in the machine's basic configuration are identified as follows: ☑ Basic configuration
- All functions not contained in the machine's basic configuration are identified as follows: ☑ Option: ...
- A summary of the unique selling points of the SINUMERIK 802D sl plus in comparison with competitors, may be found in the chapter "Summary of unique selling points".
- For information on marketing options through the machine manufacturer, please see the technical description of the particular machine.

We reserve the right to make technical changes

Contact person at the machine manufacturer

Sales

Phone:	+49 xxx xxx
Fax:	+49 xxx xxx
Email:	xxx@machinemanufacturer.com
Web:	http://www.machinemanufacturer.com

Service

Phone:	+49 xxx xxx
Fax:	+49 xxx xxx
Email:	xxx@machinemanufacturer.com
Web:	http://www.machinemanufacturer.com

Homepage:

http://www.machinemanufacturer.com

Contact person at Siemens

Sales / 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

Introduction 1

1.1 Application

The SINUMERIK 802D sl plus is a customized operator panel controller for standard CNC turning machines..

The SINUMERIK 802D sl plus allows you to easily operate machine tools by providing graphical support for all operator control actions.
The functions in JOG (manual) mode enable you to quickly set the machine up for machining, in accordance with your requirements. More specifically, this consists of calculating the workpiece position in the machine, as well as maintaining and measuring the tools in use.

For programming purposes, the SINUMERIK 802D sl plus provides you with a DIN/ISO editor that is easy to operate and has a complete G-code in accordance with DIN66025 and ISO dialect. During programming, graphical support is provided for technology canned cycles and contours.

The SINUMERIK 802D sl plus is an efficient complete system, covering all required fields of application without additional setup and training costs:

- Easy-to-use interface for all machine functions
- Graphical interfaces for technology canned cycles and contour editor
- DIN/ISO programming on the machine
- DIN/ISO programming offline via CAD/CAM system
- Automatic tool measuring during setup

1.2 Machine spectrum

The SINUMERIK 802D sl plus is particularly recommended for the following machine types:

1. Single-slide turning machines with X and Z axes
   - Turning
   - Centric drilling on the end face
2. Like 1.) with rotating tools (C axis mode)

System overview

2

2.1 SINUMERIK 802D sl

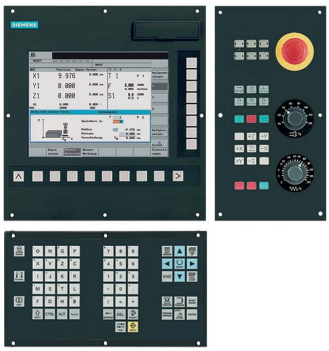
The SINUMERIK 802D sl operator panel controller is the ideal package for use with standardized turning and milling machines. Used in conjunction with the new, compact and reliable SINAMICS S120 drive system, the SINUMERIK 802D sl is a complete package for machine tool applications. These range from low-volume production to medium-volume production to production of more advanced workpieces with any type of hole machining and milling on end face and outer surface.

- Digital drive technology via DRIVE-CLiQ
- Up to 4 interpolating axes and one spindle
- Identical hardware and software for turning and milling
- Powerful PLC based on SIMATIC S7-200 with "ladder logic" programming
- Large, easy-to-read color display

See catalog NC 61 for additional information

Highlights

- **Powerful and reliable machine package from Siemens**
- **Extremely dynamic drives and motors**
- **Matched, complete package**



SINUMERIK 802D sl
with 10.4 inch TFT color display,
CNC keyboard and machine control panel

Modular
SINAMICS
Drive System

1PH7 spindle motor,
1FK6 servo motors

2.2 Operator panel

The operator panel front consists of an operator panel with a 10.4 inch color TFT display, 8 + 2 horizontal and 8 vertical softkeys, and a CNC keyboard (horizontal or vertical options available). This facilitates clear and user-friendly operation of the machine functions. We also offer the coordinated machine control panel MCP 802D sl with feed and spindle override.

Highlights

- All relevant functions at a glance, thanks to horizontal and vertical softkeys
- Brilliant color display, balanced and high-quality design of operator components
- Easy data handling thanks to the easily accessible Compact Flash card slot on the front

Setup functions

3

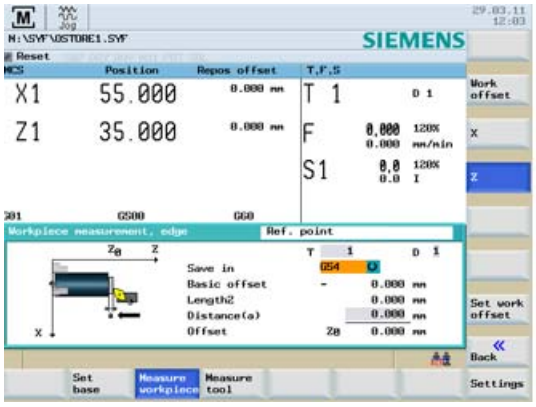
3.1 Work offsets

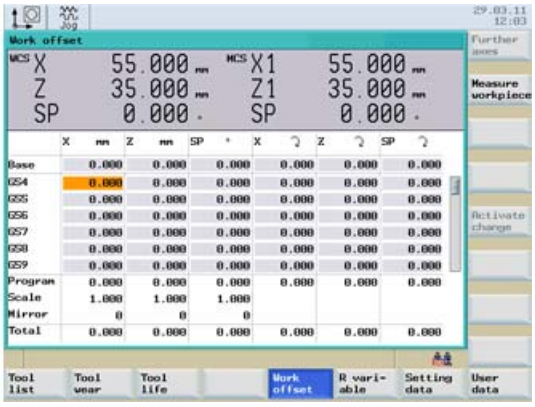
☑ Basic configuration

Within the SINUMERIK 802D sl plus, the following adjustable work offsets are available:

- A permanently effective basic offset (G500)
- Other work offsets (G54-G59)

Graphical interface support is provided for the purpose of adjusting workpiece zeros. This means that you can switch directly between the Measure workpiece screens and Work offset list.





Highlights

- **Easy setup of different workpieces using graphical guidance**
- **Clear overview of all work offsets**

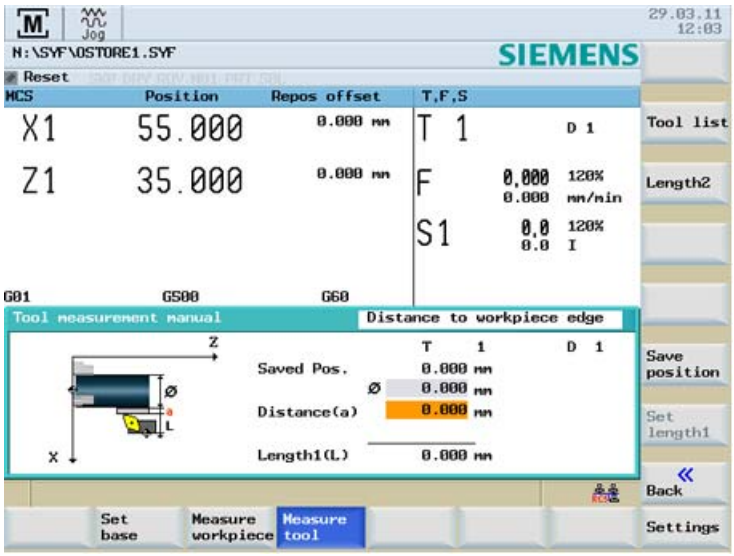
3.2 ToolMeasuring

3.2.1 Measure tools manually in JOG

☑ Basic configuration

The tool compensation values can be directly determined during machine setup. To this end the SINUMERIK 802D sl plus offers graphical support for measuring tool length and diameter:

- Graphic support for measuring tools in the X- (length 1) and Z-direction (length 2)
  - Either enter the workpiece diameter in the Ø field or the workpiece length in the Z0 field.
  - After selecting the **SetLength1** or **SetLength2** softkey, the control system will determine the searched length 1 or length 2 according to the preselected axis.
  - The determined tool offset value will be saved and transferred into the tool list.
- Button for saving the position, to retract the tool from the workpiece, to stop the spindle, and to measure the workpiece prior to transferring the results



Highlight

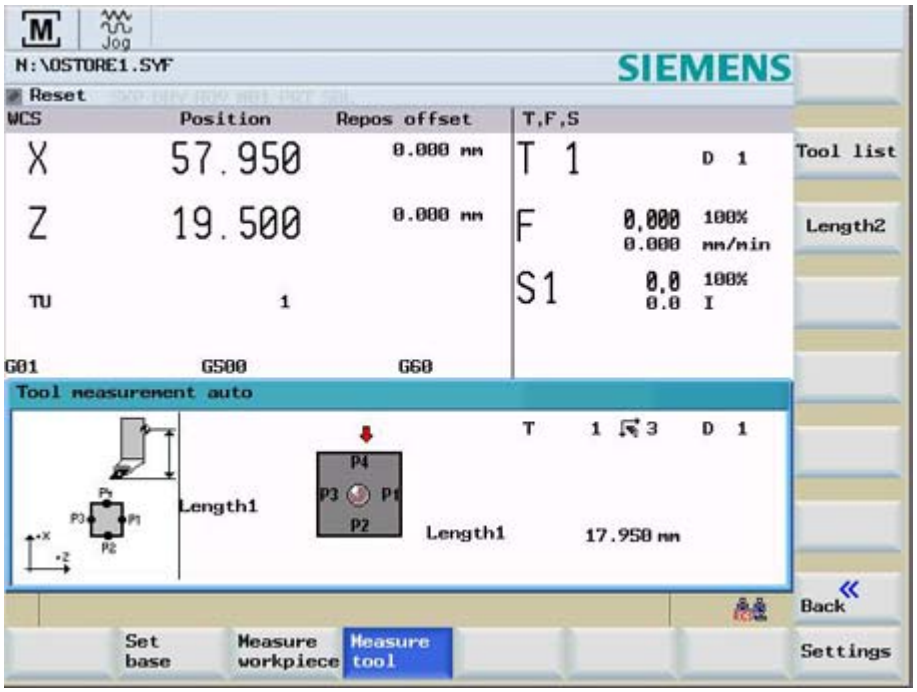
- **Save time during tool setup by seeing exactly what you are doing.**

3.2.2 Measure tools automatically in JOG

☑ Basic configuration

In JOG mode, the machine can automatically determine the tool compensation values for length 1 (X-direction) or length 2 (Z-direction). To this end the SINUMERIK 802D sl plus offers graphical support for automatic measuring and for calibrating the tool probe.

- To measure the tool length, simply move the tool near the probe.
- Click "Start". When the probe detects the tool, the controller automatically calculates the tool geometry.



Highlight

- **Speedy and precise tool measuring is standard**
- **Pre-installed graphic support for measuring with tool probe**

3.3 Tool management

3.3.1 Tool list

☑ Basic configuration

For managing tools, the SINUMERIK 802D sl plus provides you with an easy-to-use tool list, which displays all relevant tool data and wear.

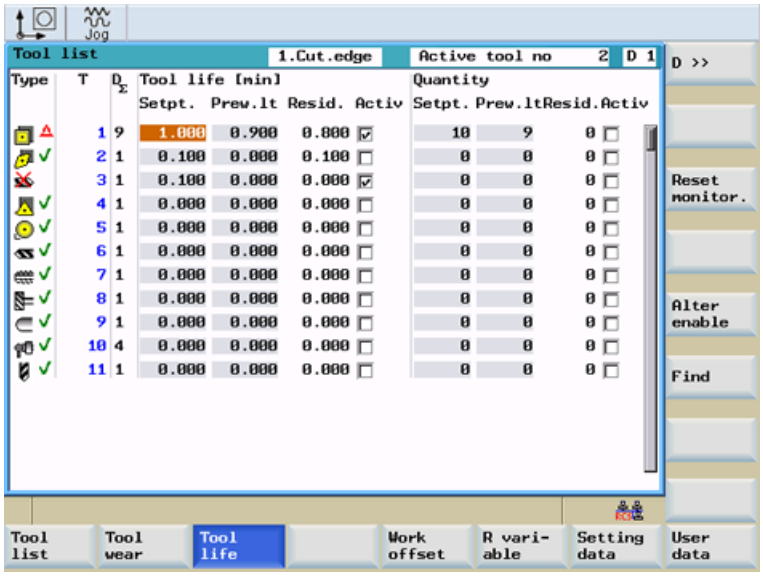
- In the tool list, you can create and delete tools using softkeys.
- For each tool, you can store the following data:
  - Special symbol for the individual tool type with direction of tool orientation (miller or drill)
  - Tools are displayed in the list with a number, e.g. T1.
  - Number of the offset set for the tool cutting edge, e.g. D1
  - Tool offset data in the X/Z direction.
  - Radius for drilling and milling tools, or plate radius for turning tools
  - Values for geometry and wear in a single table
  - Display of the cutting edge position for turning tools
- Using individual password protection, you can specify the maximum permissible input values for tool wear to avoid collisions, for example. This can be done using display machine data MD 208, MD 209 and MD 374.

Highlight

- **All tool data at a glance**
- **Higher degree of reliability when managing tool data**

3.3.2 Monitoring of tool life and quantity of workpieces

☑ Basic configuration



The SINUMERIK 802D sl plus offers automatic tool monitoring.

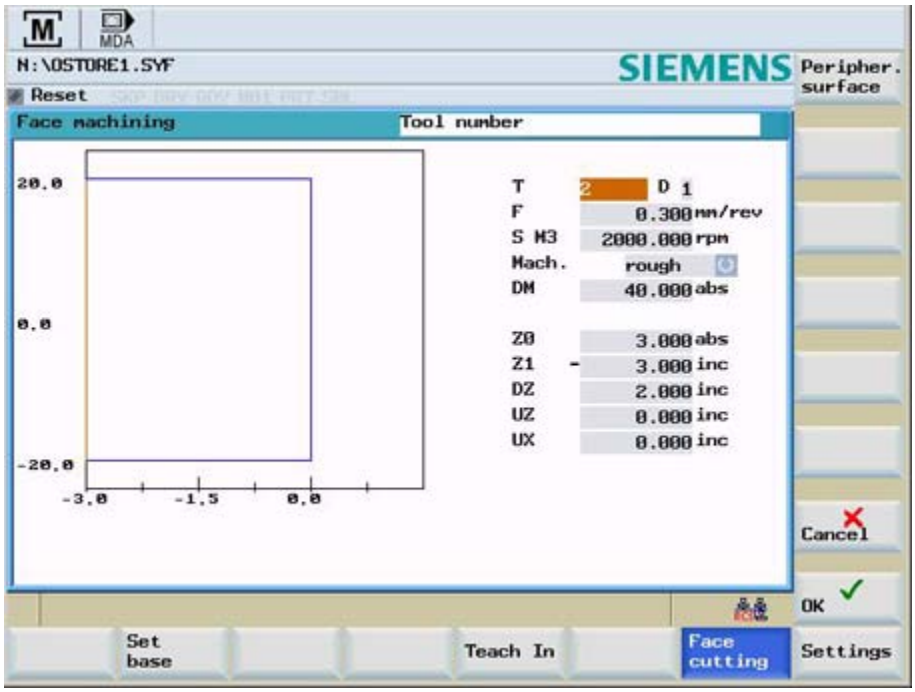
- You can monitor tool wear by observing tool life and/or workpiece quantity. If a tool reaches its wear limit, an alarm is given automatically and the tool is suspended from further machining.
- You can specify the following data in the tool monitoring:
  - Tool life, specified as a setpoint, and prewarning limit for tool monitoring. The time remaining before the tool is deactivated is calculated and displayed.
  - Workpiece quantity, specified as a setpoint, and prewarning limit for tool monitoring. The workpiece quantity remaining before the tool is deactivated is calculated and displayed.
  - Tool monitoring can be activated for tool life and/or quantity of workpieces.
- When tool life monitoring is activated, tool life is monitored during the tool's operation time (G1, G2, G3). Workpiece quantity is monitored using a program command at the end of the part program, usually Setpiece(1).

Highlight

- **Efficient monitoring of tool life and workpiece quantity is standard**

3.4 Face turning in MDA mode

☑ Basic configuration



When preparing a blank workpiece, SINUMERIK 802D sl plus supports you in the following ways:

- Input of machining parameters by dialog, complete with tool, work offset, feedrate, and spindle speed
  - Position the axes on the starting point and enter the values in the mask
  - After entering the machining parameters press the OK button, the program will be created automatically; i.e. the programmer does not have to write the part programs himself.
- The MDI blocks are also retained after NC Start and can be saved as a part program.

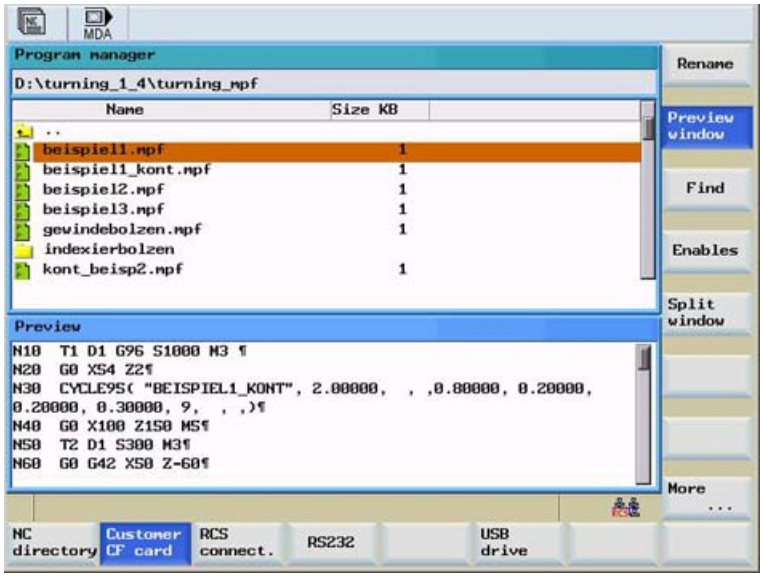
Highlight

- **Speedy preparation of the blank workpiece**

Program management and user memory 4

4.1 Program Manager

☑ Basic configuration



With the SINUMERIK 802D sl plus Program Manager, you can easily manage your part programs.

- PC-like functions, e.g. Mark, Copy, Paste, and Rename
- File names for part programs can be entered in clear text, making them easy to identify (max. 25 characters).
- Clear structures with subdirectories on several levels
- Quick search function based on entry of the 1st letter of the program name. The controller automatically positions the cursor on a program with the initial letter matching that which was entered.
- Preview of the first seven lines of the part program before editing
- All part programs available on the machine, thanks to the 1 MB user memory
- Access to shared network drives and sharing of directories for remote access via Ethernet networking

Highlights

- **Better overview with clear-text file names**
- **User-friendly data handling in typical PC style with copy, paste, rename, etc.**

4.2 User memory and data management

4.2.1 Buffered CNC user memory

☑ Basic configuration

SINUMERIK 802D sl plus 1 MB

Management of up to 100 part programs. For larger quantities, we recommend that part programs be managed using the CF card.

Highlight

- **Large memory space included in scope of delivery**

4.2.2 Compact Flash card

☑ Basic configuration, only CompactFlash card required

A CompactFlash card slot is located directly at the operator panel front of the SINUMERIK 802D sl.

- Card can be inserted or removed during operation, i.e., the machine does not have to be restarted in order for the CompactFlash card to be recognized.
- Cover can be closed while the card is inserted in order to protect the unit from dust.
- Load and execute part programs from the CompactFlash card
- No loss in speed during execution of part programs from the CompactFlash card (DNC operation)
- No special software necessary for reading/writing CompactFlash cards via PC

Part programs on the CompactFlash card are not edited on the control, but instead, at the PC.

Highlight

- **Powerful and reliable solution for handling a large volume of user data**

4.2.3 Serial data transfer

☑ Basic configuration, installation of RCS 802 tool on PC (included on Toolbox CD as standard)

The SINUMERIK 802D sl plus facilitates easy data transfer to and from the PC via the RS 232 interface. To do this, install the RCS 802 tool on your PC.

- Backing up of machine data
- Archive/series startup file
- Backing up of part program data

Note: If you have not received the Toolbox CD, please contact your machine OEM.

Highlight

- **Easy and well proven data transfer**

Programming 5

5.1 DIN/ISO language

☑ Basic configuration

For DIN/ISO programming purposes, the SINUMERIK 802D sl plus offers a large pool of commands which are oriented to the task at hand:

- G-code according to DIN66025 and in ISO dialect mode (mixed programming possible with G290/G291 commands)
- G-functions and extended G-functions

  Powerful commands, e.g. CIP for circular interpolation via intermediate point
- Unlimited number of programmable work offsets

  Using the commands TRANS, SCALE, MIRROR, ROT, you can shift, scale, mirror, and rotate the workpiece coordinate system as required.
- Calculation operations and logic operations of variables

  These calculation operations include, e.g.: +, -, *, /, sin, cos, exp, ==, <>
- User data

  You can freely define variables in the part program using names (clear text) and type (LUDs, no GUDs).
- R parameters (calculation parameters)

  300 predefined R parameters are available as flexible calculation variables (floating point format).
- System variables

  Access from the program to, for example, tool offsets, axis positions and measuring values
- Program control structures

  Language commands such as IF and GOTO are available for programming with conditions and loops.

Highlight

- **Unbeatable pool of commands for flexible and time-optimized part programs**

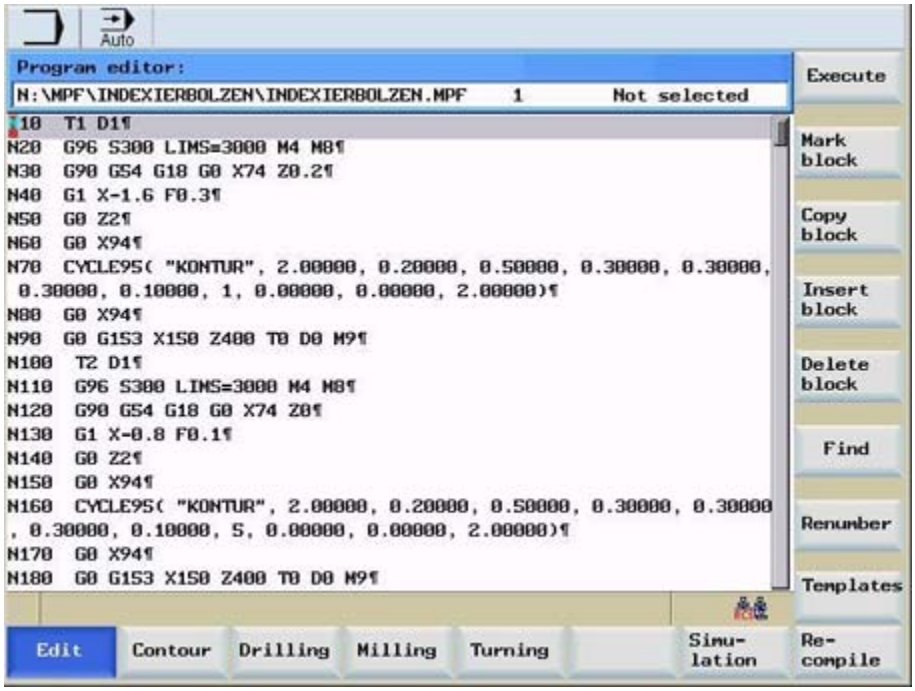
5.2 DIN/ISO editor

☑ Basic configuration

For programming purposes, the SINUMERIK 802D sl plus has a text-based DIN/ISO editor. This allows you to directly enter or change CNC language commands, meaning that you have access to the entire range of CNC functions.

The editor offers the following range of functions:

- Easy-to-use program entry with Copy, Paste, Search/Replace, Numbering, etc.
- Graphics editor for creating workpiece contours entered in the program as G-code
- Standard machining cycles for turning, drilling, and milling
- Simulation of the program created
- Recompilation of program cycles for further editing in the graphical interface
- Direct execution from any NC program block (block search)



Highlight

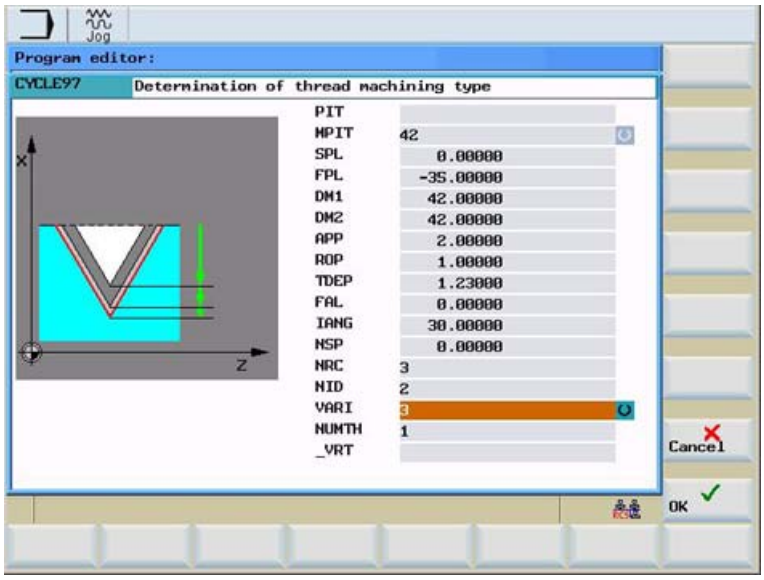
- **Save time by programming with efficient DIN/ISO editor**

5.3 Machining Cycles

☑ Basic configuration

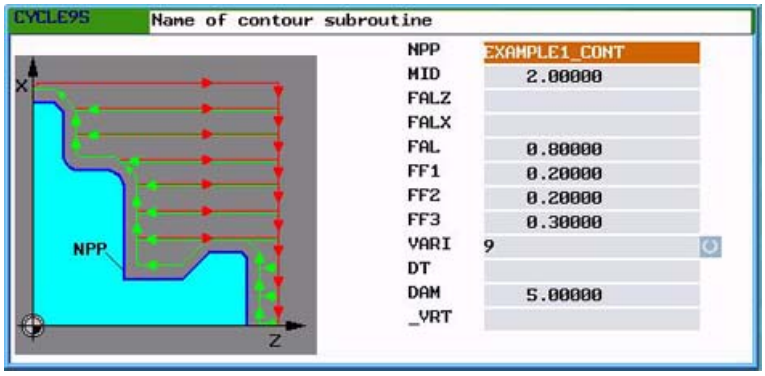
For recurring machining operations, the SINUMERIK 802D sl plus provides you with graphical support for the following technology canned cycles. You can parameterize these and assemble them for the program in any way you wish.

- Turning
  - Face turning
  - Stock removal
  - Groove, undercut
  - Thread cutting, thread
- Drilling
  - Centering, drilling, counter-boring, reaming, deep-hole drilling, tapping
  - Repetition of hole machining using hole patterns row/circle (MCALL)
- Milling
  - Face milling
  - Contour milling
  - Rectangular pocket and spigot, circular pocket and spigot
  - Elongated holes on a circle, slots on a circle, circumferential slots
  - Thread milling (inside and outside)



Comprehensive functional and graphical support is provided:

- For parameterization support is provided by the clear screens and infotexts accompanying the parameters, e.g. machining type of the thread.
- Tapping without compensating chuck is including in the basic scope, including thread interpolation (CYCLE84).
- You have extensive selection possibilities for high-performance machining, e.g. feed interruption for stock removal. The parameter DAM is used for this.



Highlight

- **Graphical cycle support helps you create your part program faster**
- **Highly flexible G-code programming extended by graphical cycles**

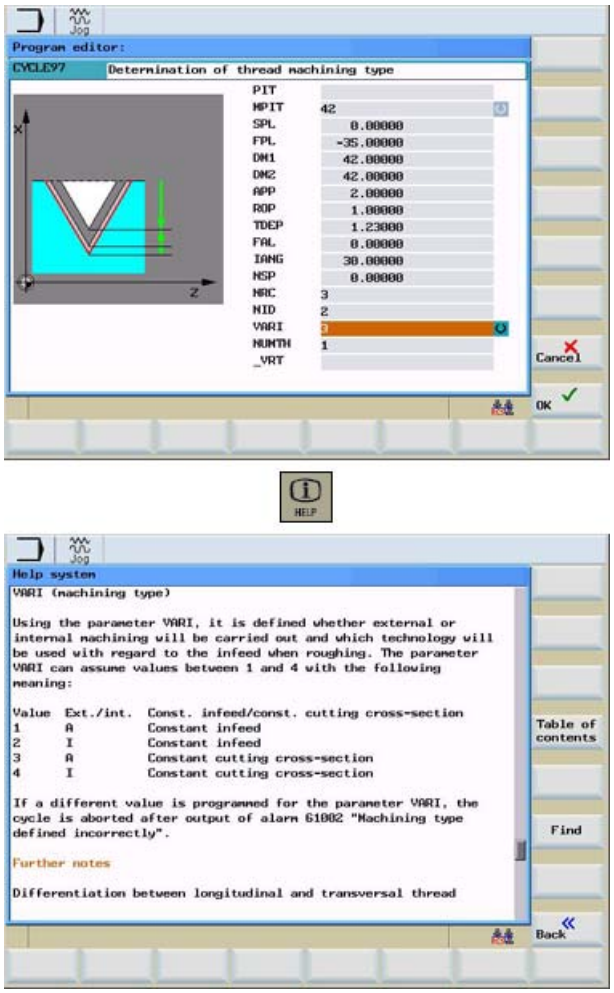
5.4 On-board user manual

☑ Basic configuration

The on-board user manual provides descriptions of all important operator functions. In addition, it provides a complete description of NC commands, cycle programming and drive alarms (as can be found in the paper documentation).

You can call up the Help menu in the following ways:

- Pressing the Help key on the CNC keyboard to call up the table of contents
- Pressing the Help key to call up the context-sensitive help system, e.g. when the cursor is at a cycle parameter. Help is immediately opened at the relevant place.

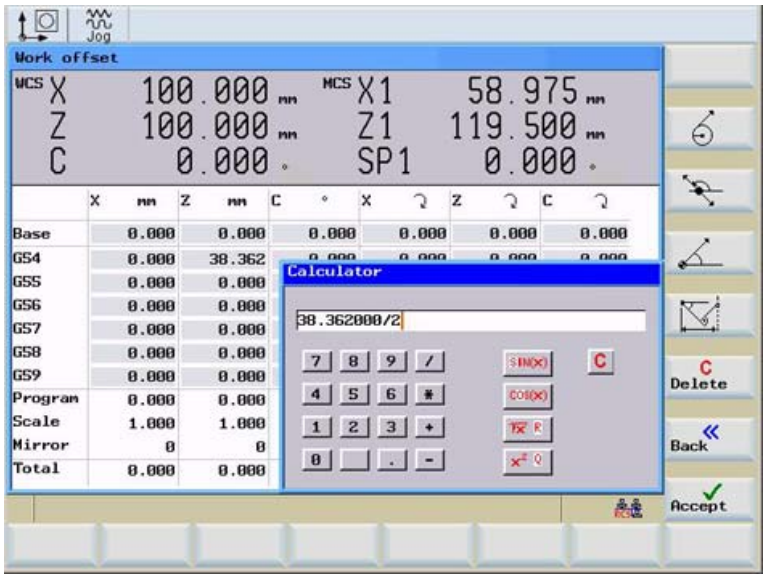


Highlight

- **Never lose time again because you don't have the user manual at hand**

5.5 On-board pocket calculator

☑ Basic configuration



The on-board pocket calculator offers the following range of functions:

- Callable from any operating area
- Take over a value from an input field and write back to it after calculation
- Four basic calculation operations, as well as sine, cosine, square, and square root functions
- Bracket function for calculating nested terms
- Functions for calculating construction points on a contour, e.g.:
  - Tangential transition between a circle sector and a straight line
  - Converting polar coordinates to Cartesian coordinates
- By pressing the Input key, you can see the result of a calculation before you confirm it with the Accept softkey.

Highlight

- **More certainty for operating and programming thanks to on-board pocket calculator - no more calculation or typing mistakes**

5.6 Free Contour Programming / Contour Calculator

☑ Basic configuration

The SINUMERIK 802D sl supports you in free programming, from simple to complex contours. Free contour programming is a support tool for the DIN/ISO editor. Programs can be decompiled in the program editor and can thus be revised in the contour calculator.

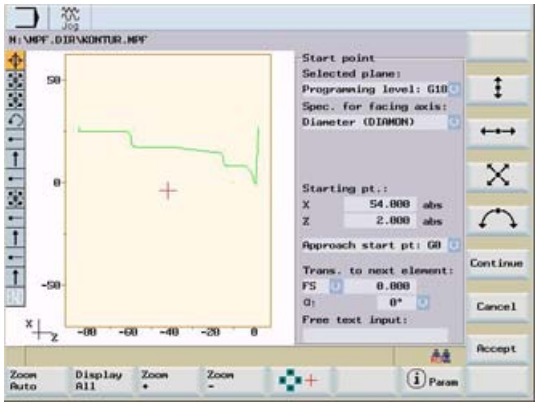
You can include the following contour elements and parameterize them in screen forms:

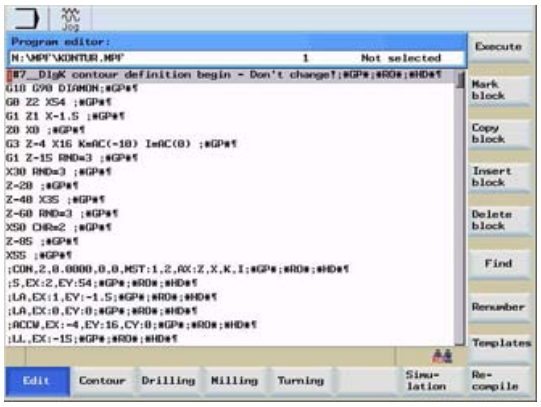
- Straight line in the vertical direction (X direction).
- Straight line in the horizontal direction (Z direction).
- Oblique line in the X/Z direction. You can enter the end point of the straight line using coordinates or an angle.
- Arc with any direction of rotation.

Additional screen forms enable you to determine the starting point and to close the contour.

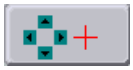
The contour calculator supports you in programming the following functions, among others:

- Calculation of only partly determined elements, as soon as the missing parameters can be derived from parameters already known, e.g. geometrical data missing in the parts drawing.
- Chaining of contour elements.
- Insert radius or chamfer between two contour transition elements.
- Transfer of the programmed contours to the edited part program.
- Toggle between radius/diameter programming
- Undercuts as transition elements between two axis-parallel straight lines: Form E, form F, thread undercuts, free undercuts

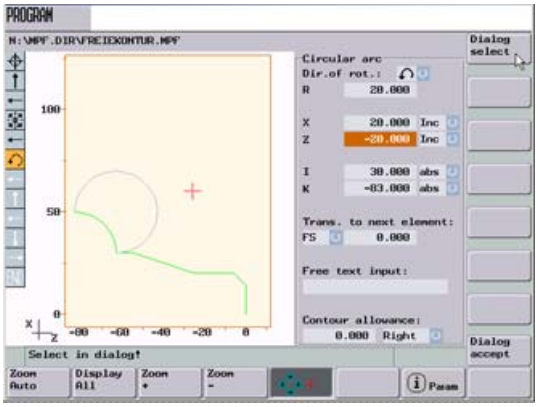




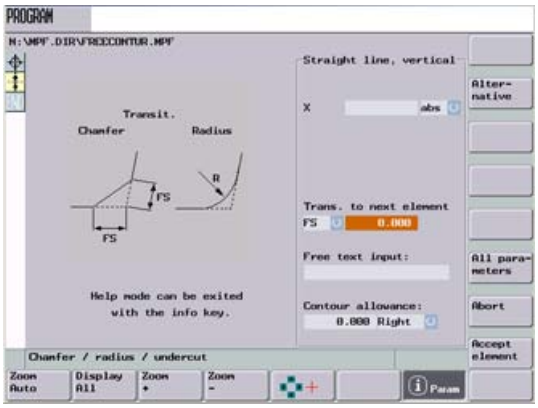
The following functions make it easier to work with the contour calculator:



When you select this softkey, you can use the cursor keys to determine a picture detail that is to be enlarged.



After selection of this softkey, graphical help screens will be displayed in addition to the relevant parameters.



Highlight

- **Quickly and reliably from the drawing to the finished workpiece**
- **Program decompilation for further processing in the contour calculator**
- **Easy input of the workpiece geometry: "Painting by numbers" just the same as for ShopTurn**

Simulation

6

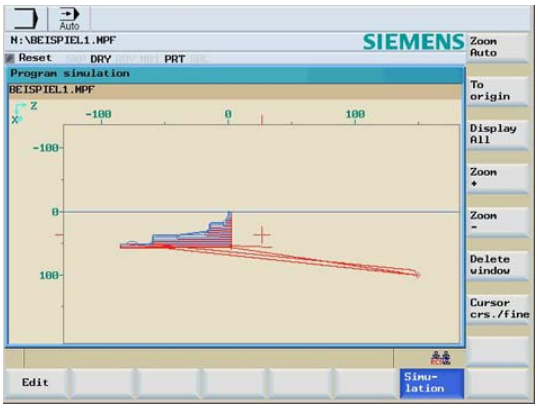
☑ Basic configuration

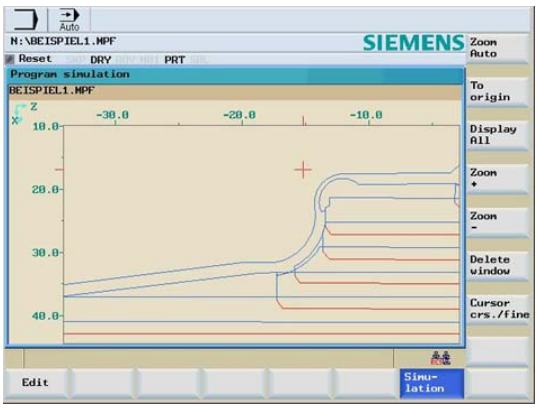
The SINUMERIK 802D sl plus offers you high process safety through integrated simulation, since it enables you to verify part programs before they are executed. By using broken-line graphics, you can trace the programmed tool path.

- Clear and transparent display by using different colors

  Rapid traverse = red

  Feedrate = blue
- Simulation results are quickly displayed by activating the dry-run feedrate (use instead of the programmed feedrate)
- Possibility of zooming into details at any time during and after simulation
- The complete workpiece is displayed using the Zoom Auto softkey





Highlight

- **Higher process reliability through powerful simulation**

Automatic mode

7

7.1 Program control

☑ Basic configuration

Single block

Single block mode can be activated for startup of the program. There is a program stop after each traversing block.

Program test

Programs can be checked before processing in a program test mode. The program is executed to completion with stationary axes.

Program editing

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

Repositioning to the contour

In machine status STOP, the machining axes can be moved to and away from the workpiece surface during machining using the handwheel or the direction keys.

Highlights

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

7.2 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, etc.) are available upon continuation of the program.

The following search variants are available:

- To the interruption point
- To any CNC block in the DIN/ISO programs
- To any subroutine levels in DIN/ISO programs

Highlights

- **Time-saving and secure re-start at any program point, as no editing of the part program is required**

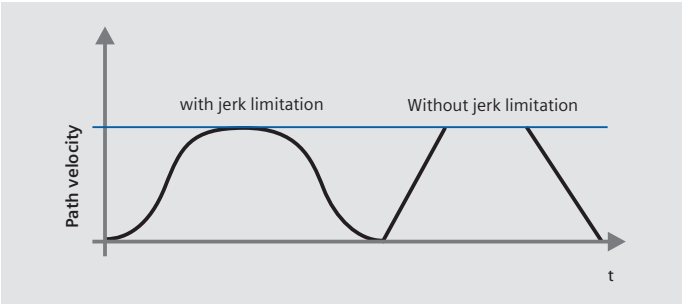
Velocity control

8

8.1 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

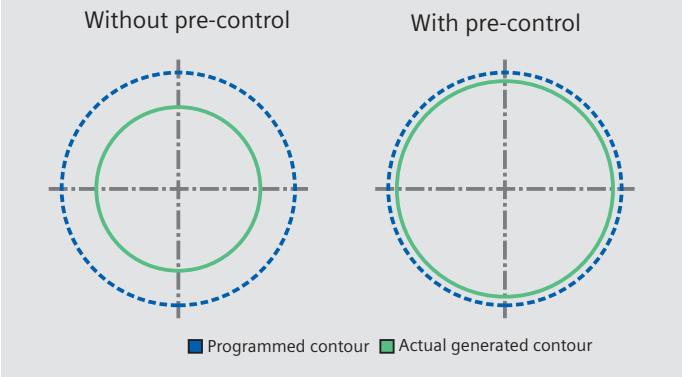
- **Longer machine lifespan through protection of the mechanical components**
- **Higher path accuracy through softer acceleration**

8.2 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**

C axis processing

9

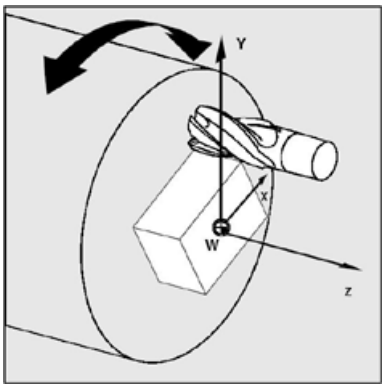
9.1 Front surface machining

☑ Basic configuration

With the SINUMERIK 802D sl plus, hole machining and milling can be executed on the front face of workpieces in the main spindle.

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

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



Machine without Y axis

- Machining with TRANSMIT

Highlights

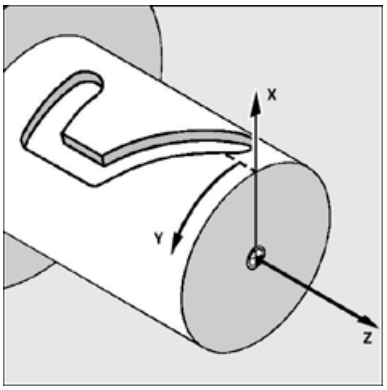
- **Full functional range for drilling and milling on the front end**
- **Reduction of set-up times by complete machining on one machine**

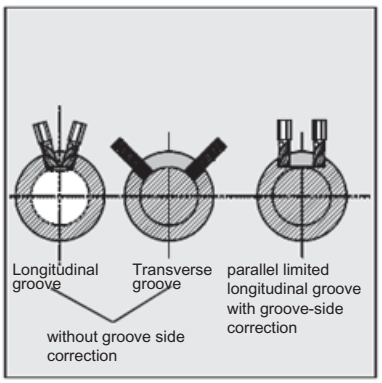
9.2 Peripheral surface machining

☑ Basic configuration

With the 802D sl plus and the outer surface transformation TRACYL drilling and milling can be performed on the outer surface of workpieces in the main spindle.

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





Highlights

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

Multilingual operator interface

10

☑ Basic configuration

On the SINUMERIK 802D sl, the following languages are available on the operator interfaces. You can switch between these using a softkey.

- Chinese (simplified and traditional)
- German
- English
- Finnish
- French
- Italian
- Korean
- Dutch
- Polish
- Portuguese
- Romanian
- Russian
- Swedish
- Spanish
- Czech
- Turkish
- Hungarian

Highlight

- **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**

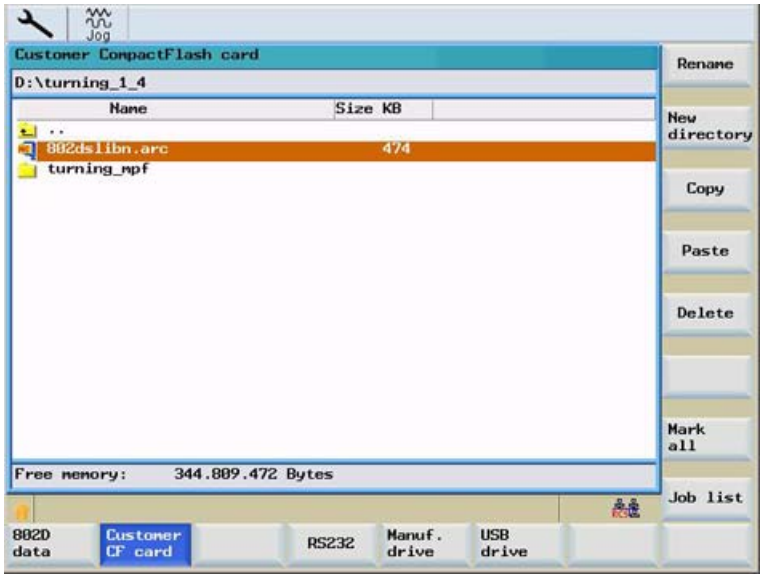
Maintenance and diagnostics 11

11.1 Maintenance-free operation

☑ Basic configuration

The SINUMERIK 802D sl offers maintenance-free operation:

- High reliability, because the SINUMERIK 802D sl has no hard disk, no battery and no fan
- Complete data backup on CF card, with all drive data



Highlight

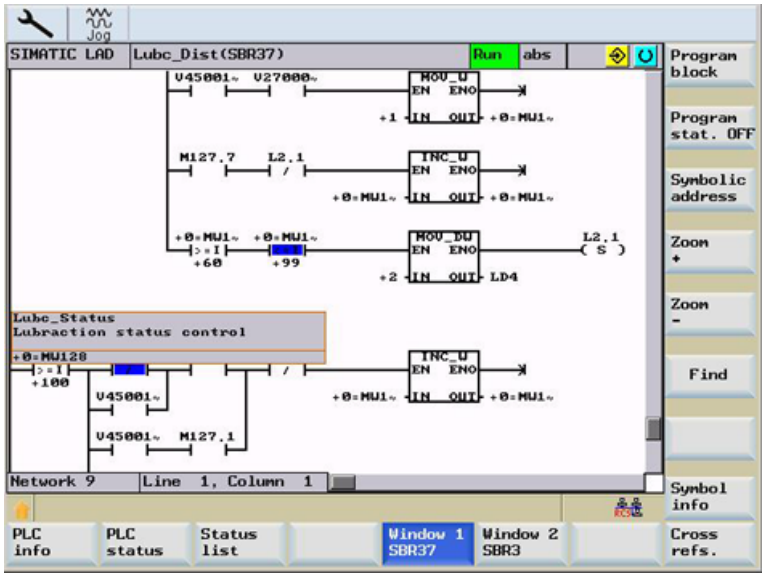
- **Highest machine availability thanks to reliable hardware**
- **Thanks to data backup via capacitor, regular battery changes are no longer required**

11.2 Diagnostics

☑ Basic configuration

The SINUMERIK 802D sl offers diagnostic functions which are easy to use:

- Diagnostic functions, such as ladder display, are available for finding causes of malfunctions or a PLC program error.
- You can switch between two windows in ladder display (e.g. for cross-references).
- You are provided with the same display as on a PC, with zoom, find, symbol info, and cross-reference functions.
- For reasons of safety, it is not possible to edit the PLC program at the machine.



Highlight

- **Highest machine availability thanks to modern diagnostic and troubleshooting tools**

Ordering data

12

The main information required for ordering is listed below:

SINUMERIK 802D sl Version T/M plus	6FC5370-0AA00-2AA1
Toolbox CD-ROM -> **Already included with each 802D sl**: including additional language files for operator interface, RCS 802 software and PLC library	6FC5810-0YC00-0YA8

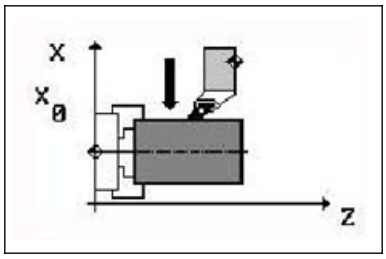
Summary of unique selling points

13

The SINUMERIK 802D sl operator panel control has the following outstanding features, which sets itself apart from the competition:

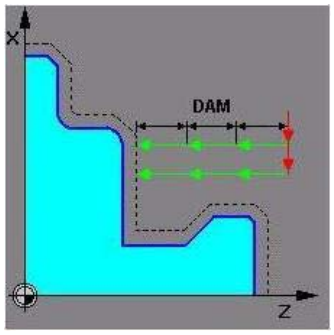
User-friendly operation

- Graphical support for setting up tools and workpiece zeros
- CF card for unlimited number of part programs
- Graphical program simulation with zoom



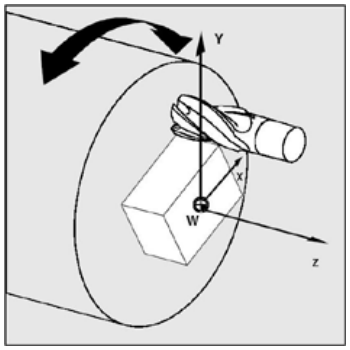
Time-saving programming

- Flexible G-code programming
- Graphical support for technological machining cycles, contour calculator as with ShopTurn
- Completely integrated user manual



Increased productivity

- C axis machining with driven tools
- Maintenance free operation and user-friendly diagnostics
- Programming and training software for the PC



Index