



Glossary

Functions and Terms

CNC controls	13/2 13/16	SINUMERIK 802S/802C/802D SINUMERIK 810D/840Di/840D
Positioning modules	13/68	FM 353, FM 354, FM 357-2
AC motors	13/72	
SIMODRIVE 611 converter systems	13/76	
SIMODRIVE POSMO converter systems		
Abbreviations	13/90	

Glossary

Functions and Terms

SINUMERIK 802S/802C/802D

Acceleration characteristic, bent

SINUMERIK 802S

When operating step drives, it is necessary to reduce the acceleration beginning at a certain speed (velocity) to guarantee optimum utilization of the stepper motor characteristic. Path and single-axis interpolations can thus be executed by step drives taking into account the specific acceleration characteristic. It is possible to select either a linear or hyperbolic acceleration reduction. Utilization of the steeper characteristic in the lower speed range permits optimum start/stop performance of the stepper motor at these low speeds.

The axis dynamics are also optimized for look ahead. This utilization of the stepper motor characteristic leads to a reduction in non-productive times in conjunction with rapid traverse movements.

The bent acceleration characteristic is also effective in the set-up mode, for override movement, and for tapping.

Acceleration with jerk limitation

SINUMERIK 802D

In order to couple optimum acceleration with reduced wear on the machine's mechanical parts, you can select SOFT in the part

program to ensure continuous, "jerk-free acceleration". When you select jerk-free acceleration, the velocity curve is generated as a sinusoidal-shaped curve.

Access protection

Protection level	Type	User	Access to (examples)
0	Password	Siemens	All functions, programs, data
1	Password	Machine manufacturer: development engineer	Defined functions, programs and data (options)
2	Password	Machine manufacturer: commissioning engineer	Defined functions, programs and data (machine data)
3	Password	End user: Service	Assigned functions, programs and data
4	Key red Switch position 3	End user: programmer, set-up engineer	< protection level 0-3 Machine manufacturer/end user
5	Key green Switch position 2	End user: qualified operator who does not program	< protection level 0-3 End user
6	Key black Switch position 1	End user: Professionally trained operator who does not program	Program selection only, tool wear input and input of zero offsets
7	Switch position 0	End user: Semi-skilled operator	No input and program selection possible, only the machine control panel can be operated

Access to programs, data and functions is protected in a user-oriented hierarchical system of eight access levels.

These are subdivided into four password levels (protection level 0 to 3) for Siemens, machine manufacturers and end users.

This gives you a multistage concept for regulating access rights to SINUMERIK controls. Protection level 0 is the highest, protection level 7 the lowest access privilege. A higher protection level automatically includes lower protection levels.

Access privileges for protection levels 1 to 3 are standardized (Siemens defaults).

Access privileges for protection levels 4 to 7 can be assigned by the machine manufacturer or end user.

Actual-value system for workpiece

The term "actual-value system for workpiece" identifies functions which allow the SINUMERIK user to do the following:

- To go to a workpiece coordinate system defined via machine data in JOG or AUTOMATIC mode without any additional manipulation after starting the control.
- To retain the settings valid at the end of the part program for the following part program as regards active level, frames

(G54–G59), kinematic transformations, and active tool offset.

- To change back and forth between the WCS workpiece coordinate system and the MCS machine coordinate system.
- To change the workpiece coordinate system (e.g. change the frames or the tool offset).

Alarms and messages

All "messages and alarms" are output on the operator panel in plain text with a symbol indicating the clear criterion. The alarm texts are stored on the control.

"Alarms and messages" from the machine can be displayed in plain text from the PLC program. A distinction is made between status messages and fault messages. While status messages are immediately cleared when the associated condition are im-

mediately deleted, fault messages must always be acknowledged.

The control's response to alarms or messages is configurable, and the texts are stored on the control.

"Alarms and messages" in the part program
Messages can be programmed to give the operator information on the current processing status while the program is executing.

Analog spindles

Unipolar or bipolar 10 V interfaces can be used.

Auxiliary function output

Auxiliary function output informs the PLC when the part program wants the PLC to handle certain machine operations. This is accomplished by forwarding the relevant auxiliary functions, with their parameters, over the PLC interface. The PLC program must process the forwarded values and signals. The following functions can be transferred to the PLC:

- Tool selection T
- Tool compensation
- Feed F/FA

- Spindle speed S
- H functions (802D only)
- M functions

Auxiliary function output may be carried out either with speed reduction and PLC acknowledgement up to the next block, or before and during travel without speed reduction and without block change delay. Subsequent blocks are then traversed without acknowledgement delay.

Axes/spindles

→ Spindle functions

- Axes
The number of interpolating path axes is limited to no more than three.
- Spindles
Spindle drives can be speed-controlled or position-controlled.

SINUMERIK 802S/802C

On the SINUMERIK 802S/802C, the encoder evaluations are integrated in the control module.

SINUMERIK 802D

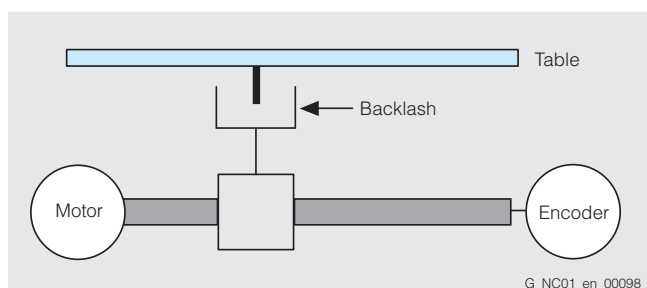
On the SINUMERIK 802D, the encoder evaluation is located in the SIMODRIVE 611 universal E's digital drive modules.

Backlash compensation

During power transmission between a moving machine part and its drive (e.g. ball screw), there is normally a small amount of backlash because setting mechanical parts so that they are completely free of backlash would result in too much wear and tear on the machine. In the case of axes/spindles with indirect measuring systems, mechanical backlash results in corruption of the traverse path, causing an axis, for example, to travel too much or too little by the amount of the backlash when the direction of movement is reversed.

To compensate for backlash, the axis-specific actual value is corrected by the amount of the backlash every time the axis/spindle reverses its direction of movement.

Following reference point approach, backlash compensation is always active in all modes.



With positive backlash (normal case) the actual encoder value is ahead of the true actual value (table): table does not travel far enough.

Glossary

Functions and Terms

SINUMERIK 802S/802C/802D

Block search

For testing part programs or following interruption of machining, it is possible to select any point in the part program using the block search function in order to start or resume at this point.

You have a choice of three different search options:

- Block search with calculation at the contour line: during the block search, the same calculations are executed as during normal program operation. The destination block is then traversed true-to-contour until the end position is reached. With this function, you can reapproach the contour from any position.
- Block search with calculation at the block end position: this function allows you to approach a target position (such as tool change position).

Once again, all calculations are executed as during normal program operation. The approach targets the end position of the destination block or the next programmed position using the method of interpolation valid in the destination block.

- Block search without calculation:
This method is used for high-speed searches in the main program. No calculations are carried out during the search. The internal control values remain the same as before the block search.
In order to execute an NC program without faults, the target block must be included as relevant block information.

You can define the target of the search by

- Directly positioning the cursor to the destination block, or
- By specifying a block number, a label, a string, a program name or a line number.

Circle via center point and end point

Circular interpolation causes the tool to move along a circular path in a clockwise or counter-clockwise direction.

The required circle is described by:

- Starting point of circle
(actual position in the block before the circle)
- Direction of rotation of circle
- End point of circle (target defined in circle block)
- Circle center point

The circle center point can be programmed as an absolute value with reference to the current coordinate zero or as an incremental value with reference to the circle starting point. If the opening angle is evident from the drawing, it can be directly programmed. In many cases, the dimensioning of a drawing is chosen so as to make it easier to program the radius to define the circular path. For arcs exceeding 180 degrees, the radius must be given a negative sign.

Circle via intermediate point and end point

If a circle is to be programmed which does not lie in a paraxial plane but obliquely in space, an intermediate point can be used to program it instead of the circle center. Three points are

required to program the circle. They are the starting point, the intermediate point, and the end point.

Clamping monitor

SINUMERIK 802D

The "clamping monitor" is one of the many extensive monitoring mechanisms for axes. When an axis is to be clamped following conclusion of the positioning procedure, you can activate the clamping monitor with the PLC interface signal "clamping in progress". This may become necessary because it is possible for the axis to be pushed beyond the zero-speed tolerance from the setpoint position during the clamping procedure.

→ Position monitoring, standstill monitoring

The amount of deviation from the setpoint position is set via the machine data. During the clamping procedure, the clamping monitor replaces the zero-speed monitor, and is effective for linear axes, rotary axes, and position-controlled spindles. The clamping monitor is not active in follow-up mode. When the monitor responds, its reactions are the same as those of the zero-speed monitor.

CNC high-level language

To meet the various technological demands of modern machine tools, a high-level CNC language has been developed for the SINUMERIK 802D and SINUMERIK 802S/802C which provides the highest possible flexibility.

Indirect programming (SINUMERIK 802D)

One option for universal use of a program is indirect programming, in which the addresses of axes, spindles, R parameters, etc., are not directly programmed but are referenced by a variable in which the required address is entered.

Program jumps

The inclusion of program jumps allows extremely flexible control of the machining process. Conditional and absolute jumps are available as well as program branches that depend on a current value. Labels that are written at the beginning of the blocks are used as destinations. The destination can be located before or after the block containing the jump.

Arithmetic and trigonometric functions

User variables and arithmetic variables make it possible to implement extensive arithmetic functions. In addition to the four basic arithmetic operations, there are functions for:

- Sine, cosine, tangent
- Arc sine, arc cosine, arc tangent (802D only)
- Square root

- Absolute value
- Power of 2 (squaring) (802D only)
- Integer component
- Rounding to whole number
- Natural logarithm (802D only)
- Exponential function (802D only)
- Translating (802D only)
- Rotation (802D only)
- Scaling (802D only)
- Mirroring (802D only)

Comparison operations and logic operations

Comparison operations with variables can be used to formulate branch conditions. The comparison functions that can be used are:

- Equal to, not equal to
- Greater than, less than
- Greater than or equal to
- Less than or equal to
- String concatenation (802D only)

The following logic operations are also available (802D only): AND, OR, NOT, EXOR

These logic operations can also be executed by bit.

CNC user memory

All programs and data, such as part programs, subroutines, comments, tool compensations, zero offsets and program user

data can be stored in the shared, battery-backed CNC user memory.

CNC program messages

→ Alarms and messages

Contour definition programming

"Contour definition programming" enables fast input of simple contours.

You can program 1-point, 2-point or 3-point definitions with the transition elements Chamfer and Fillet with the aid of help displays in the Editor program by specifying Cartesian coordinates and/or angles.

Contour monitoring

SINUMERIK 802C/802D

The following error is monitored for a defined tolerance band to ensure machining precision. An impermissibly high following error might be caused by a drive overload, for example. If an

error occurs, the axes/spindles are stopped. "Contour monitoring" is always on when the channel is active and in position-controlled mode. If the channel is interrupted or is in the Reset state, contour monitoring does not take place.

Cycle support

→ Technological cycles

Data backup

The following data backup methods are available for your system software and user data:

- Integrated FEPRM
- Serial interface RS 232C (V.24)
- PC card (802D only)

Glossary

Functions and Terms

SINUMERIK 802S/802C/802D

Diagnostic functions

A self-diagnostics program and test aids for service have been integrated in the controls. The status is displayed for:

- Interface signals between the CNC and the PLC and between the PLC and the machine
- Variables
- PLC memory bits, timers and counters
- PLC inputs and outputs

For test purposes, the user can set combinations of output signals, input signals, and memory bits. Alarms and messages also provide valuable diagnostic information.

In the "service display" menu it is possible to call up important information about the axis and spindle drives such as:

- Absolute actual position (802C/802D only)
- Setpoint position
- Following error (802C/802D only)
- Setpoint speed
- Actual speed

Dimensional notation metric and inch

Depending on the measuring system used in the production drawing, you can program workpiece-related geometrical data in either metric measure (G71) or inches (G70). The control can be set to a basic system regardless of the programmed measuring system. You can have the control convert the following geometrical data into the opposite system, and thus enter them directly (examples):

- Path information X, Y, Z ...
- Interpolation parameters I, J, K and circle radius CR
- Thread pitch

SINUMERIK 802D

- Programmable zero offset (TRANS)
- Polar radius RP

With the G700/G710 programming expansion, all feedrates are also interpreted in the programmed measuring system (inch/min or mm/min). In the "machine" control area, you can also switch back and forth between inch and metric notation using a softkey.

Display functions

All current information can be displayed on the control panel screen, such as:

- Block currently being executed
- Previous and following block
- Actual position, setpoint/actual distance to go (802C/802D only)
- Current feedrate
- Spindle speed
- G functions
- Auxiliary functions
- Workpiece name
- Main program name

- Subroutine name
- All data entered, such as part programs, user data and machine data
- Help texts

Important status information is displayed in plain text, for example:

- Alarms and messages
- Position not yet reached (802C/802D only)
- Feed hold
- Program in progress

Drives

SINUMERIK 802S

To control power circuits for stepper motors, the SINUMERIK 802S is equipped with an interface for frequency and directional signals.

The interface can control up to three stepper motor power circuits.

SINUMERIK 802C

The SINUMERIK 802C is equipped with a ± 10 V interface to the SIMODRIVE 611 analog and SIMODRIVE 611 universal converter systems.

SINUMERIK 802D

The SINUMERIK 802D is equipped with a PROFIBUS interface to the SIMODRIVE 611 universal E converter system.

Electronic handwheels

Using electronic handwheels it is possible to move selected axes simultaneously in manual mode. The meaning of the lines on the handwheels is defined by increment weighting. If coordinate offset or coordinate rotation is selected, it is also possible

to move the axes manually in the transformed workpiece coordinate system. The maximum input frequency of the handwheel inputs is 500 kHz.

Execution of large CNC programs

Part programs that are too large for CNC memory can be read in via the RS 232C (V.24) interface and executed while the read-in is in progress.

The CNC executes the program from cyclic storage. Part programs are automatically reloaded into cyclic storage as soon as free space becomes available.

Feedforward control

SINUMERIK 802D

Feedforward control allows you to reduce axial following errors almost to zero. For this reason, feedforward control is often referred to as "following error compensation".

Particularly during acceleration in contour curvatures, e.g. circles and corners, this following error leads to undesirable, speed-dependent contour errors.

For compensation of contour violations, the SINUMERIK 802D features speed-dependent feedforward velocity control.

Feedrate override

The current velocity setting is overridden by the programmed velocity via machine control panel or by the PLC.

0 to 200% on the 802D, and
0 to 120% on the 802S/802C.

Follow-up mode

SINUMERIK 802D/802C

An axis/spindle which is in follow-up mode can be moved externally, and the actual value can still be recorded. The traverse paths are updated in the display. Standstill, clamping and posi-

tioning monitoring functions are not effective in follow-up mode. A new reference point procedure for the axes is not required when follow-up mode is cancelled.

FRAME concept

SINUMERIK 802D

With the Frame concept, it is possible to transform perpendicular coordinate systems very simply by translating, rotating, scaling and mirroring using the following instructions:

- TRANS Programmable zero offset
- ROT Rotation in space or in a plane
- SCALE Scaling (scale factor)
- MIRROR Mirroring

The instructions can also be used several times within one program. Existing translations can either be overwritten or added. Additive Frame instructions:

- ATRANS
- AROT
- ASCALE
- AMIRROR

If swivelling tools or workpieces are available, machining can be extremely flexible.

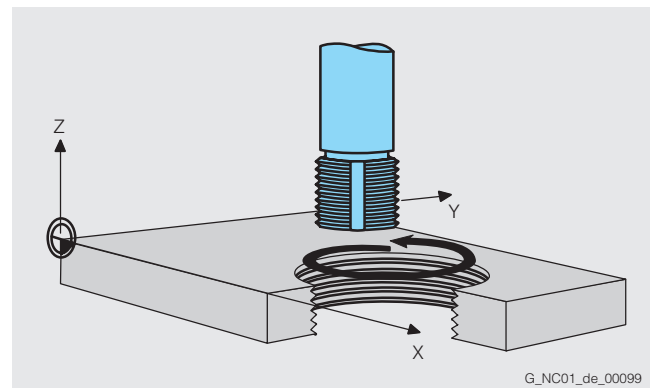
Helical interpolation

Helical interpolation is especially suitable for machining inside or outside threads with profiling cutters and for milling lubrication grooves. The helix consists of two motions:

- A circular movement in one plane
- A linear movement perpendicular to this plane

The programmed feed F either refers only to the circular motion or to the total path velocity of the three CNC axes involved.

In addition to the two CNC axes performing circular interpolation, other linear motions can be performed. The programmed feed F refers to the axes specially selected in the program.



Helical interpolation: Thread cutting with profiling cutter

Intermediate blocks for tool radius compensation

Traversing movements with tool offset selected can be interrupted by a limited number of intermediate blocks (blocks without axis movements in the compensating plane).

Glossary

Functions and Terms

SINUMERIK 802S/802C/802D

I/O interfacing via PROFIBUS-DP

SINUMERIK 802D

PROFIBUS-DP represents the protocol profile for the distributed I/O. It enables high-speed cyclic communication at 12 Mbit/s for smaller volumes of data. PROFIBUS-DP offers the same

advantages when joint transfer of data for I/O and drives is involved: high availability, data security, and standard message frame structure.

Languages/extended languages

The graphical user interface for the SINUMERIK controls is available in practically any desired language.

The interface can be switched back and forth online between two different languages.

Leadscrew error/measuring system error compensation

The principle of "indirect measuring" on CNC-controlled machines is based on the assumption that the leadscrew pitch is constant at every point within the traversing range so that the actual position of the axis can be derived from the position of the drive spindle (ideal situation). Manufacturing tolerances in leadscrews, however, result in more or less considerable dimensional deviations (so-called leadscrew errors). Added to this are the dimensional deviations caused by the measuring system as well as its installation tolerances on the machine (so-called measur-

ing system errors), plus any machine-dependent error sources.

Because these dimensional deviations directly affect the precision of workpiece machining, they must be compensated for by the respective position-dependent offsets.

The offsets are determined on the basis of the measured error characteristic, and entered in the control on start-up in the form of so-called compensation tables.

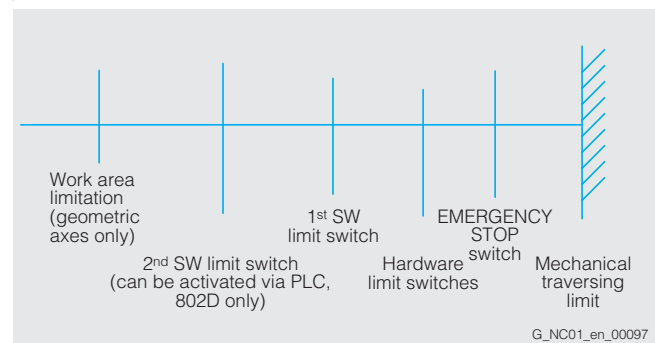
Limit switch monitor

Hardware limit switches preceding the EMERGENCY STOP switch, hardware limit switches in the form of digital inputs limit the traversing range of the machine axes via the PLC interface. Deceleration is carried out either in form of emergency braking with setpoint zero or in accordance with a braking characteristic. The axes must be retracted in the reverse direction in JOG mode.

Software limit switches precede the hardware limit switches, are not overrun, and do not become active until after the approach to reference point.

SINUMERIK 802D

A second pair of plus/minus software limit switches can be activated via the PLC.



Overview of travel limits

Linear interpolation

Up to 3 axes can interpolate linearly.

Look Ahead

During the machining of complex contours, most of the program blocks describe very short paths with sharp changes in direction. If a contour of this type is processed with a fixed programmed path velocity, an optimum result cannot be obtained.

In traverse blocks with tangential transitions the drives cannot attain the required final velocity because of the short paths. Corners are cut off.

With the "look ahead" function, an optimum machining velocity is attained. On tangential block transitions, the axis is accelerated and decelerated beyond block boundaries so that no drops in velocity occur. On sharp changes of direction, bevelling of the contour is reduced to a programmable dimension.

Measuring system error compensation

→ Leadscrew error/measuring system error compensation

Modes

In the "machine" control area, you have a choice of three modes:

- JOG
JOG mode (jogging/inching) is intended for the manual movement of axes and spindles as well as for setting up the machine. The setting-up functions are approach to reference point, repositioning, handwheel control or incremental mode, and redefinition of control zero (preset/set actual value).
- MDA
In MDA (Manual Data Automatic/Manual Input) mode, you can enter program blocks or sequences of blocks, then execute them immediately via CNC Start. The tested blocks can then be saved in part programs.
- AUTO
In AUTO mode, your part programs are executed automatically once they have been selected from the workpiece, part program and subroutine directories (normal operation).

While AUTOMATIC mode is in progress, the user can write or edit another part program.

In the TEACH IN submode (802S/802C only), you can take over sequences of motions by traveling to and storing positioning in the AUTO programs.

In MDA and AUTO mode, you can modify the sequence of a program with the following "program-affecting" functions:

- SKP Skip block
- DRY Dry run feed rate
- ROV Rapid traverse override
- SBL1 Single block with stop after machine function blocks
- SBL2 Single block with stop after every block
- SBL3 Halt in cycle (802D only)
- M01 Programmed stop
- PRT Program test

Monitoring functions

The controls contain permanently active watchdogs (monitors) which detect malfunctions in the CNC, the PLC and the machine early enough to prevent damage to workpiece, tool or machine. When a problem occurs, the machining sequence is interrupted and the drives stopped. The reason for the problem is recorded and displayed as alarm. At the same time, the PLC is informed that a CNC alarm is pending.

There are monitors for the following:

- Read
- Format
- Position encoder and drive
- Contour
- Position
- Zero speed
- Clamping
- Setpoint speed
- Actual speed
- Enable signals
- Voltage
- Temperatures
- Microprocessors
- Serial interfaces
- Transfers between CNC and PLC
- System memory and user memory

Monitoring of tool life and count (option)

SINUMERIK 802D

This function permits monitoring of tool life and/or count. If the monitoring time of a cutting edge expires during machining, an alarm is output, and a VDI signal set. The life of the active edge

of the loaded tool is monitored. Monitoring of the count covers all tool cutting edges which are used to manufacture a workpiece.

Online ISO dialect interpreter

SINUMERIK 802D

With the online ISO dialect interpreter, part programs in other ISO dialects such as G codes from other manufacturers can be read into, edited and processed in the SINUMERIK 802D.

Parts programs can also be written in the normal manner. G290/G291 can be used to also swap between the two programming languages within a parts program.

Part program management

Part program management can be organized according to workpieces. This permits clear assignment of programs and data to workpieces.

Glossary

Functions and Terms

SINUMERIK 802S/802C/802D

PLC status

In its "diagnostics" area, the operator interface allows you to check and/or change PLC status signals.

This allows you to do the following on site without a programming unit:

- Check the input and output signals from the PLC's I/O
- Troubleshoot
- Check the interface signals for diagnostic purposes

The status of the following data can be displayed on the operator panel:

- Interface signals from/to the machine control panel
- NCK/PLC and MMC/PLC interface signals
- Data blocks, memory bits, timers, counters, inputs and outputs

For test purposes, the status of the above-mentioned signals can also be changed. Signal combinations are possible, and up to 10 operands can be modified simultaneously.

PLC user memory

PLC user memory is used to store the PLC user program, the user data, and the PLC basic program.

PLC remote diagnostics

SINUMERIK 802D

Using this function you are able to monitor your PLC program online, or also transmit the PLC program from the control to a PC

or vice versa. A modem-to-modem connection is used via the stationary network or mobile radio network.

Polar coordinates

SINUMERIK 802D

When programming in polar coordinates, it is possible to define positions with reference to a defined center by specifying the

radius and angle. The center point can be defined as an absolute or incremental dimension.

Position monitor

To protect the machine, SINUMERIK controls provide extensive monitoring mechanisms for axis monitoring:

- Motion monitoring: contour monitor, positioning monitor, zero-speed monitor, clamping monitor, setpoint speed monitor, actual speed monitor, encoder monitor
- Static limitations monitoring: limit switch monitor, working area limitation (802D only)

The positioning monitor is always activated following "setpoint-related" termination of motion blocks (802C/802D only). In order to ensure that an axis is positioned within the specified amount of time, a timer, which can be programmed in the machine data,

is started following termination of a motion block. When the timer has run down, a check is made to make sure that the following error did not exceed the limit value (machine data). When the specified "fine exact stop limit" has been reached or following output of a new, non-zero position setpoint (e.g. when positioning to "coarse exact stop limit" with subsequent block change), the positioning monitor is deactivated and replaced by the zero-speed monitor.

The positioning monitor is effective for linear and rotary axes (802D only) as well as for positioned-controlled spindles.

Programmable acceleration

SINUMERIK 802D

With the function "programmable acceleration" it is possible, for example, to modify the axis acceleration in the program in order to limit mechanical vibration in critical program sections.

The path or positioning axis is then accelerated at the programmed value.

The maximum acceleration value stored in the control is not exceeded. This limitation is active in AUTOMATIC mode and in all interpolation types. As part of intelligent motion control, the function provides a more precise workpiece surface.

Programming language

The CNC programming language is based on DIN 66025.

Reference point approach

When using a machine axis in program-controlled operation, it is important to ensure that the actual values supplied by the measuring system agree with the machine coordinate values.

Reference point approach (limit switch) is performed separately for each axis in a sequence that can be defined in the machine data at a defined velocity using the direction keys or automatically via program command G74.

Reference point approach for an axis with absolute value encoders is carried out automatically when the control is switched on (without axis motion) if the corresponding axis is recognized as being calibrated.

Repos

Following a program interruption in AUTOMATIC mode (e.g. to take a measurement on the workpiece and correct the tool wear values or because of tool breakage), manual repositioning of the tool is possible after changing to JOG mode. In this case, the control stores the breakpoint coordinates and displays the differential travel of the axes in the actual-value window as Repos (repositioning) offset.

Repositioning can also be performed in JOG mode using the axis- and direction keys.

It is not possible to overshoot the breakpoint; the feedrate override switch is effective.

Rotary axis, endlessly-turning

SINUMERIK 802D

Depending on the application, the traversing range of a rotary axis can be limited via software switch (e.g. operating range

between 0° and 60°) or to a corresponding number of rotations (e.g. 1000°), or unlimited (endlessly turning in both directions).

Serial interface (RS 232C)

A serial interface is provided for data input/data output. This interface can be used to load and archive programs and data. The

interface can be initialized and operated with menu assistance via the operator panel.

Skip blocks

CNC blocks that are not to be executed in every program pass, for example can be skipped.

The blocks to be skipped are marked with "/" character preceding the block number. The statements in the skip blocks are not executed, and the program resumes with the block that follows the skip block.

Spindle speed functions

Spindle speed

- analog (± 10 V)
- digital (802D only)

Spindle override 0 % to 200 % (120 % on the 802S/802C); gear stages selectable

- via the part program (commands M41 to M45) or
- automatically via programmed spindle speed (M40) or oriented spindle stop (positioning mode) with SPOS ¹⁾

Spindle monitoring with the functions ¹⁾

- axis/spindle not moving ($n < n_{\min}$)
- spindle in setpoint range
- max. spindle speed
- programmable lower (G25) and upper (G26) spindle speed limitation
- min./max. speed of gear stage
- max. encoder operating frequency
- target position monitoring for SPOS

Constant cutting speed with G96 (in m/min or inch/min) at the tool tip for uniform turning images and thus better surface quality.

Thread cutting with constant pitch ¹⁾:

The following types of thread can be produced with G33:

Cylindrical, conical or transversal, single-start or multiple-start, right-hand or left-hand.

In addition, it is possible to produce multi-block threads by chaining thread blocks.

Tapping with compensating chuck/rigid tapping:

When tapping with compensating chuck (G63), the compensating chuck takes up differences between spindle movement and drilling axis. Prerequisite for rigid tapping (G331/G332) is a position-controlled spindle with position encoding system.

The traversing range of the drilling axis is thus not restricted. By using the method whereby the spindle as a rotary axis and the drilling axis interpolate, threads can be cut to a precise depth (e.g. blind hole thread).

1) Prerequisite: Actual position encoder (measuring system) with corresponding resolution.

Glossary

Functions and Terms

SINUMERIK 802S/802C/802D

Spindle speed limitation

→ Spindle functions

Standard system start-up

In order to transfer a specific configuration as easily as possible to other controls on the same type of machine, you can create so-called standard system start-up files.

Standard system start-up is then extremely easy and user-friendly, and can even be accomplished without a programming unit by using an IBM-compatible PC.

SINUMERIK 802S/802C

Simply link two ECUs via the RS 232C (V.24) interface and make a direct transfer of all data (MD, MPF, SSK, etc.) from one control to the other in order to obtain an exactly equivalent control.

SINUMERIK 802D

Store a standard system start-up file on PC card in the control, plug the PC card into the next control, and start the standard system there.

Subroutines

Machining sequences which are often repeated are best written and stored in the form of a subroutine. Such a subroutine is called from a main program (number of passes ≤ 9999).

The SINUMERIK 802D allows seven, the SINUMERIK 802S/802C four subroutine levels in a main program. Subroutines can be completely protected against unauthorized readouts and displays (cycles). A main program can also be called from another main program or subroutine.

Tapping with compensating chuck/rigid tapping

→ Spindle functions

Teach-in

SINUMERIK 802S/802C

"Teach-in" is generally taken to mean the transfer of current positions to the CNC program.

When teach-in is used in AUTOMATIC mode, it is possible not only to transfer the program but also to test and correct it immediately.

The program is stopped and the axes are moved into the desired position with the JOG keys on the MCP or handwheel. This position is transferred to the program as a traversing block and it can then be started again at any point. A reset is not required. Positions already taught in the program can be corrected and new positions can be inserted.

Technology cycles

Technology cycles (standard cycles) for drilling/milling and turning are available for frequently repeated machining tasks. You can store these technology cycles together with your user cycles

in the control as protected subroutine. The parameters are initialized via graphically supported input screen forms in plain text.

Thread cutting

→ Spindle functions

Tool change via T number

In chain, rotary-plate and box magazines, a tool change takes place in two stages. The T command searches for the tool in the magazine, and the M command places the new tool in the spindle. In the case of turret magazines on turning machines, the tool

change, that is to say, the search and the change itself, is executed with only a T command. You can preselect the type of tool change in the machine data.

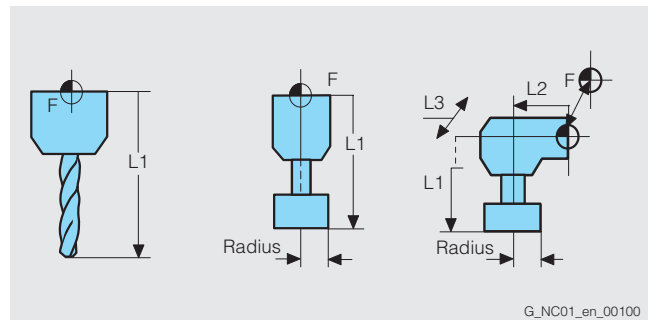
Tool offsets

When writing programs, you do not need to take tool dimensions such as cutter diameter, cutting edge length or tool length into account. You program the workpiece dimensions directly, e.g. according to the production drawing. When a workpiece goes into production, the tool paths are controlled in dependence on the relevant tool geometry in such a way that the programmed contour can be produced with any tool used.

You enter the tool data separately in the control's tool table, then simply call the required tool, with its offset data, in the program. While the program is executing, the control fetches the offset data it needs from the tool files and automatically corrects the tool path for different tools.

A tool is selected by programming a T function (a 5-digit integer on the 802D, a 2-digit integer on the 802S/802C) in the block.

Each T number can be assigned a corresponding axis (D addresses). The number of tools to be managed in the control is specified during configuring. A tool offset block comprises 25 parameters, for example:



Tool offsets

- Tool type
- Up to 3 length offsets
- Radius compensation
- Wear dimension for length and radius
- Base dimension

The wear and the base dimension are added to the relevant offset.

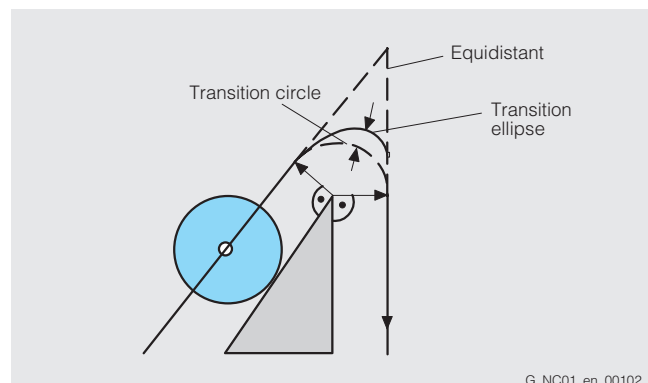
Tool radius compensation

When tool radius compensation is enabled, the control automatically computes the equidistant tool paths for different tools. To do this, it needs the tool number (T), the tool offset number (D) (with cutter number), the machining direction (G41/G42), and the relevant working plane (G17 to G19).

The path is offset in two axes in dependence on the selected tool radius.

The control can also automatically insert a circle or a straight line in the block with the tool radius compensation when no intersection with the preceding block is possible.

The process of tool radius compensation may be interrupted only by a certain number of successive blocks or M commands containing no traversing command or path specifications in the compensating plane.



Traveling around the external angles with transition circle/ellipse

Tool types

The tool type determines which geometric data are needed for TO memory and how they are to be used.

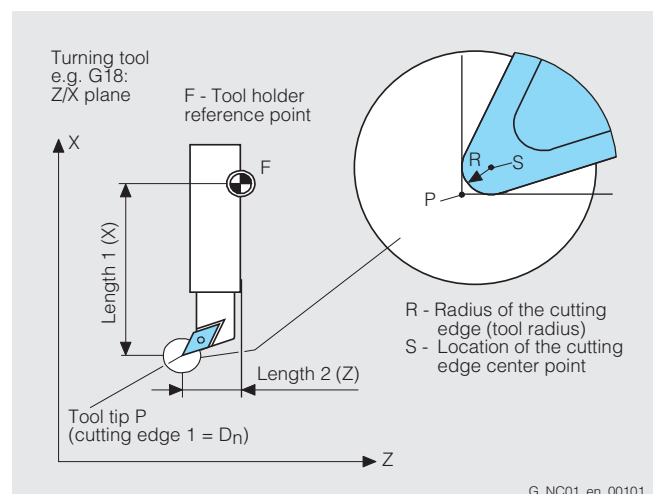
The control combines these geometric data into a result value (e.g. total length, total radius).

The calculated result value goes into force when the offset memory is activated. The use of these values in the axis is determined by the tool type and by the current processing level (G17, G18 or G19).

The following tool types may be parameterized:

- Group 1xy: Cutting tools (from ballhead cutter to bevel cutter)
- Group 2xy: Drills (from twist drill to reamer)
- Group 5xy: Turning tools (from roughing tool to threading tool)

The storing of all tool offsets is supported by input screenforms.



Example: geometry of turning tool

Glossary

Functions and Terms

SINUMERIK 802S/802C/802D

Travel to fixed stop (option)

SINUMERIK 802D

You can use this function to move e.g. tailstocks or quills to a fixed stop in order to clamp workpieces. The contact pressure can be defined in the parts program. »Travel to fixed stop« is possible simultaneously for several axes, and parallel to the movement of other axes.

SINUMERIK 802C

(see function: SIMODRIVE 611)

Traversing range

The range of values for the traversing ranges depends on the specified computational resolution. The default value for "computational resolution for linear or angle positions" in the machine data (1000 increments per mm or per degree) can be used to program the following value ranges (see Table).

The traversing range can be limited by software limit switches and operating ranges.

	G70 [inches, degrees]	G71 [inches, degrees]
Linear axes X, Y, Z, ...	± 399,999.999	± 999,999.999
Rotary axes A, B, C, ... (802D only)	± 999,999.999	± 999,999.999
Interpolation parameters I, J, K	± 399,999.999	± 999,999.999

User interface

SINUMERIK 802S/802C

The user interface is divided into five operating areas:

- Machine
- Parameters
- Program
- Services
- Diagnostics

This makes it possible, for example, to write a new part program while parts production is in progress. When a switch is made from one operating area to another, the menu that was last active is saved. Five horizontal softkeys and window techniques ensure easy, user-friendly machine operation.

SINUMERIK 802D

The user interface is divided into six operating areas:

- Machine
- Offset/parameters
- Program manager
- Program editor
- System
- Alarms

This makes it possible, for example, to write a new part program while parts production is in progress.

When a switch is made from one operating area to another, the menu that was last active is saved. "Hot keys" are provided for switching from one operating area to another.

Eight horizontal and eight vertical keys as well as windows techniques ensure easy, user-friendly machine operation.

User machine data

Machine data are provided by the NCK to configure the PLC user program.

These data make it possible to activate specific machine configurations, machine expansions, and user "options".

Velocity

The maximum path velocity, axis velocity and spindle speed are affected by the dynamic response of machine and drive and by the limit frequency of actual-value acquisition.

The minimum velocity must not fall below 10^{-3} units/IPO cycle.

The maximum velocity of the axis is generally limited by the mechanics or by the limit frequency of the encoder.

Working area limitation

→ Zero offsets

SINUMERIK 802D

In addition to the limit switches, "work area limitations" limit the traversing range of the axes. They set up protected zones in the machine's work area in which tool movement is prohibited and which protect surrounding equipment, such as tool turrets, measuring stations, etc., from damage.

The limitations refer to the basic coordinate system.

Monitoring consists of ascertaining whether the tool tip penetrates the protected space also taking into account the tool radius.

Zero offsets

You can define »Zero offsets« which can then be called in the parts programs.

Zero-speed monitoring

SINUMERIK 802C/802D

The zero-speed monitor checks to see whether the axis moves further out of its position than the value specified as zero-speed tolerance in the machine data. The "zero-speed monitor" is always active when the "zero-speed monitor delay time" has expired or upon reaching "fine exact stop" as long as no new traversing command is pending.

When the monitor responds, an alarm is generated and the relevant axis/spindle is brought to a standstill with rapid stop via a speed setpoint ramp. The zero-speed monitor is effective for linear and rotary axes as well as for position-controlled spindles. It is not active in follow-up mode.

Glossary

Functions and Terms

SINUMERIK 810D/840Di/840D

3D tool compensation (option)

Inclined surfaces can be machined with 3D tool compensation or tool compensation in space. The 3D tool compensation function enables contour milling and face milling with a defined path. The inclined tool clamping position on the machine can be entered and compensated. The control computes the resulting positions and movement automatically. The radius of a cylindrical





cal milling cutter at the tool insertion point is included in the calculation. The plunge depth of a cylindrical milling cutter can be programmed. The milling cutter can be turned not only in the X, Y and Z planes, but also by the angle of lead and the lateral angle.

Acceleration with jerk limitation

To couple optimum acceleration with reduced wear on the machine's mechanical parts, you can select SOFT in the part program to ensure continuous, "jerk-free acceleration". When you

select jerk-free acceleration, the velocity characteristic over the path is generated as a sinusoidal-shaped curve.

Access protection

Protection level	Type	PLC DB10 DBB 56 Bit ...	User	Access to (examples)
0	Password	–	Siemens	All functions, programs, data
1	Password	–	Machine manufacturers: Development	Defined functions, programs and data (options)
2	Password	–	Machine manufacturers: Commissioning engineers	Defined functions, programs and data (machine data)
3	Password	–	End user: Service	Assigned functions, programs and data
4	Key red Switch position 3 	7	End user: Programmers, Machine-setters	< Protection level 0-3 Machine manufacturers/end users
5	Key green Switch position 2 	6	End user: Qualified operators who do not program	< Protection level 0-3 End users
6	Key black Switch position 1 	5	End user: Trained operators who do not program	Program selection only, tool wear entries, and zero offset entries
7	Switch position 0 	4	End user: Semi-skilled operator	No input and program selection possible, only the machine control panel can be operated

Access to programs, data and functions is protected in a user-oriented hierarchical system of eight access levels. These are subdivided into:

- 4 password levels (protection levels 0 to 3) for Siemens, machine manufacturers, and end users, and
- 4 keyswitch positions (protection levels 4 to 7) for end users (keyswitch positions can also be evaluated via PLC).

SINUMERIK controls thus provide a multistage concept for controlling access privileges.

Protection level 0 has the highest, protection level 7 the lowest access privileges. A higher protection level automatically includes all protection levels below it. Access privileges for protection levels 0 to 3 are preprogrammed by Siemens as standard defaults.

A password takes precedence over a keyswitch position, and machine manufacturers or end users can change protection levels 4 through 7.

Subroutines can only be protected in their entirety against unauthorized reading and displaying.

Advanced Processing 1 and 2 (Option)

The function »Advanced Processing 1« permits reduction of the interpolation cycle down to 4 ms with the SINUMERIK 840Di/840DiE system software Basic, Universal and Plus.

The function "Advanced Processing 2" only applies to the SINUMERIK 840Di/840DiE software Plus, and permits reduction of the interpolation cycle down to 2 ms.

Alarms and messages

- Alarms and messages:
All messages and alarms are output on the operator panel in plain text with the date and time and a symbol indicating the acknowledgment criterion. The alarm texts are saved either on the hard disk (PCU 50/PCU 70/MMC 103) or on the flash card (PCU 20/MMC 100.2). All alarms are entered in a configurable alarm log.
- Alarms and messages in the part program:
Messages can be programmed to give the operator information on the current machining situation during the program run. Message texts may be up to 124 characters long, and are displayed in two lines (of 62 characters each). The contents of variables may also be displayed in message texts.

Example 1:

N10 G1 F2000 B=33.333

N15 MSG ("Rotary table position: «\$AA_IW[B]«" degrees")

Display on message line following traversed block N10:

Rotary table position: 33.333 degrees

Example 2:

N20 MSG ("X-position" »\$AA_IW[X]« "Check!")

Display: X-position ... Check!

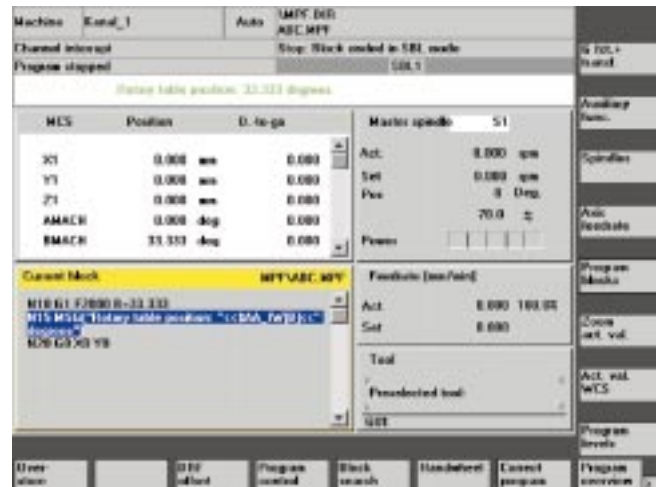
In addition to programming messages, you can also set alarms in an NC program. An alarm always goes hand in hand with a response from the control as per the alarm category.

You will find a list of responses to the various alarms in the Start-up Guide. The alarm text must be configured. Alarm numbers 65000 to 67999 are reserved for the user.

Example 3:

N100 SETAL (65001)

Effect: Display CNC start inhibit,
delete: with Reset



Programming and displaying message texts

- Alarms and messages from the PLC:
Machine-specific alarms and messages from the PLC program can be displayed as plain text. Messages comprise operational messages and error messages. Whereas the display of an operational message is immediately deleted when the condition is no longer active, error messages must always be acknowledged. Application-specific alarm numbers in the range from 40000 to 89999 can be assigned to general, channel-specific, axis-specific and spindle-specific application alarms and messages. The reaction of the control to alarms or messages can be configured. The configured alarm and message texts are saved in application-specific text files.
- Specific evaluation of alarms:
A channel-specific signal can be used to decide whether other channels may continue to be used when an alarm is issued.

Analog axis (option)

This function is intended for individual motors on machines which cannot be controlled with digital drives, such as large spindle motors or motors for tool changers. An analog axis can be used very much like a digital axis. It can be programmed like a digital interpolating path axis or spindle.

Pure functions of the SIMODRIVE 611 drive control system are, of course, not possible for external drive units linked via an analog speed setpoint interface. This involves functionalities which fall back on internal axis feedback and communication via the drive bus, such as torque feedforward control, filters for damping mechanical resonance, "Safety Integrated", and so on. Separate EMC measures must be taken for external drive units where applicable.

Analog axes can be implemented in two different ways:

- With the "analog axis" option, which is available for each axis, you can control with SINUMERIK 840D, software version 4.3 and higher, and depending on the NCU system software used

12/31 axes (on Technology PC card) and up to 3 or 8 of the CNC axes available per NCU via a speed setpoint interface ± 10 V with analog drives (e.g. SIMODRIVE 611 analog). The setpoint output to the analog drive amplifier is handled by a DMP compact "analog output" module, which is operated on an NCU terminal block on the SIMODRIVE 611 digital's drive bus. The actual axis or actual spindle value is directed by an unconditioned signal generator on the motor to a free actual-value input for direct measuring systems on the SIMODRIVE 611 digital.

- Beginning with software version 5.3 for the SINUMERIK 840D, up to two analog axes can be operated via a closed-loop control plug-in module with digital setpoint interface for HLA (HLA submodule) hydraulic linear drives:
 - Velocity setpoint output ± 10 V
 - Positioning-measuring system evaluation for voltage signals
 A technology PC card is no longer required starting with this software release since the functionality is already included in the NCU system software.

Glossary

Functions and Terms

SINUMERIK 810D/840Di/840D

Analog value control

With the system variable \$A_OUTA(n), values from up to eight possible analog outputs can be preset directly in the part program. A submodule "DMP Compact 1 A analog" (see Section 5) for analog outputs is required in the NCU terminal block (on the

SINUMERIK 840Di PROFIBUS-DP and S7 300 output modules). Prior to being output to the hardware, the value preset by the NCK can be modified by the PLC in DB10. The values are written to the hardware output in the interpolation cycle.

Asynchronous subroutines

An asynchronous subroutine is a CNC program which can be started based on an external event (e.g. a digital input) or from the PLC. Inputs are allocated to subroutines and activated by programming SETINT. If the relevant event occurs, the CNC block currently being processed is immediately interrupted. The CNC program can later be continued at the point of interruption.

→ Interrupt routines with high-speed retraction from the contour

Multiple asynchronous subroutines must be assigned different priorities (PRIO) so that they can be processed in a certain order. Asynchronous subroutines can be disabled and re-enabled in the CNC program (DISABLE/ENABLE).

Auxiliary function output

With "auxiliary function output", the PLC is informed when the part program wants it to carry out certain operations. This is accomplished by forwarding the appropriate auxiliary functions, with their parameters, to the PLC interface. The values and signals that are forwarded must be processed by the PLC user program. The following functions can be forwarded to the PLC:

- Tool selection T
- Tool compensation D/DL

- Feed F/FA
- Spindle speed S
- H functions
- M functions

Auxiliary function output may be carried out either with speed reduction and PLC acknowledgment up to the next block or before and during the movement without speed reduction and without block change delay. Following blocks are then traversed

Axes/spindles or positioning axes/auxiliary spindles

→ Spindle functions

Axes

In accordance with their functions, the axes are subdivided into:

- Interpolating path axes:

An additional interpolating axis/spindle extends the number of axes/spindles in the basic configuration.

- Positioning axes:

Non-interpolating feed and positioning axes with axis-specific feed; axis movements beyond block boundaries are possible. Positioning axes need not participate in the actual machining process, e.g. workpiece/tool feeder, tool magazine.

Positioning axes can move in parallel with the machining process without reserving an additional machining channel (concurrent positioning axes). Parallel movements of this type can considerably reduce non-productive machining time.

Spindles

Spindle drives can be speed-controlled or position-controlled.

Auxiliary spindles

Auxiliary spindles are speed-controlled spindle drives without actual position encoder, e.g. for power tools.

Axes, trail

When an axis defined as a leading axis moves, the coupled axes (following axes) assigned to it travel the paths derived from the leading axis, taking into account a coupling factor (setpoint coupling). Together, the leading axis and the following axes form a coupled-axis grouping. Definition and activation take place simultaneously when the modal-like instruction TRAILON is encountered. A coupled-axis grouping may consist of any com-

bination of linear and rotary axes. A coupled axis can be assigned up to two leading axes (in different coupled-axis groupings). A simulated axis can also be defined as leading axis, in which case the real axis actually does the travelling, taking into account the coupling factor. Another application for coupled axes is the use of two coupled-axis groupings to machine the two sides of a workpiece.

Axis container (option)

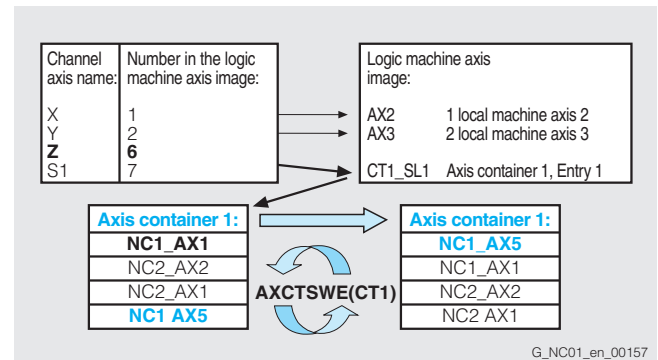
→ Link axis

In rotary indexing machines/multi-spindle machines, the axes holding the workpiece move from one machining unit to the next. Since the machining units are subject to different NCU channels, the axes holding the workpiece must be dynamically reassigned to the corresponding NCU channel if there is a change in station/position. Axis containers are used for this purpose. Only one workpiece clamping axis/spindle is active on the local machining unit at a time. The axis container combines the possible connections to all clamping axes/spindles, of which only one is active at a time for the machining unit.

The following can be assigned using axis containers:

- Local axes and/or
- Link axes

Changing of the usable axes defined by an axis container is achieved by shifting the entries in the axis container. Shifting can be triggered by the parts program.



Example of axis container: following rotation of the axis container by 1, the channel axis Z is assigned to axis AX5 on NCU 1 instead of axis AX1.

Axial coupling in the MCS machine coordinate system (option)

This option is required in order to be able to use coupled axes implemented in the basic coordinate system in transformations. A coupling is carried out 1:1 in the machine coordinate system.

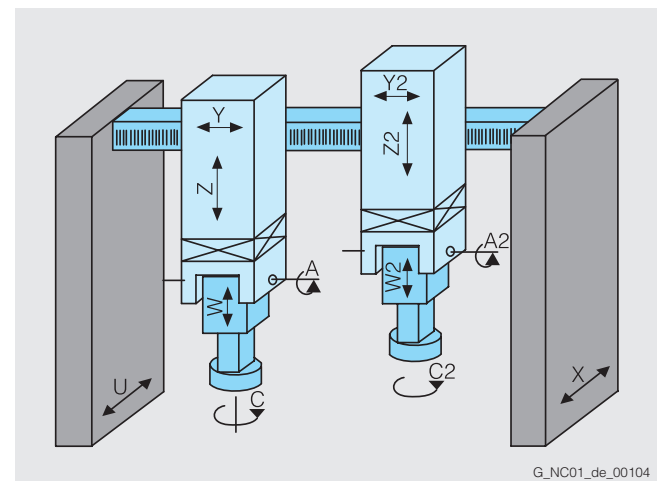
The participating axes can be reconfigured following Reset.

On machine tools with separately movable heads on which a transformation must be activated, the orientation axes cannot be coupled using the standard coupling methods (COPON, TRAILON).

The axes participating in the coupling are determined via an axial machine datum that is updated with RESET. This makes it possible to reassign pairs of axes during operation and enable and disable them via CNC language commands.

There are master and slave axes. A master axis can have more than one slave axis, but a slave axis cannot at the same time be master axis (no cascading).

To protect the heads from collisions, collision protection can be set and activated via either machine datum or VDI interface.



Glossary

Functions and Terms

SINUMERIK 810D/840D/840D

Axis limitation from the PLC

→ Protection zones

The preactivation of protection areas with specification of a position offset is programmed in the part program. You can put the preactivated protection areas into effect in the PLC program via the PLC interface. As a result, the relevant protection area is activated, for example, before a tool probe is swivelled into position in the work area to see whether the tool or a workpiece is in the path of the swivelling probe.

Another axis limitation by the PLC can be put into effect by activating the second software limit switch via a PLC interface signal. This reduction of the work area may become necessary, for example, when a tailstock is swivelled into position. The change is immediately effective, and the first software limit switch is no longer valid.

Axis/spindle exchange

An axis/a spindle is permanently assigned to a specific channel via machine data. With the "axis/spindle exchange" function, it is possible to release an axis/a spindle (program command

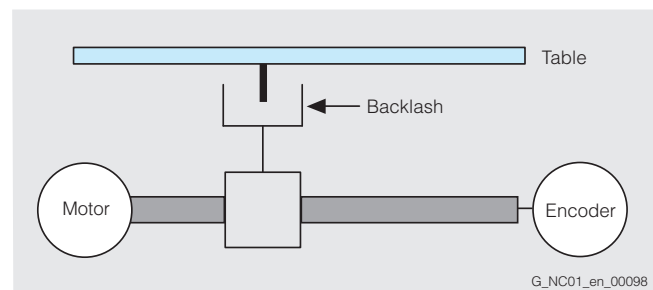
RELEASE) and to assign it to another channel (command GET), i.e. to exchange the axis/spindle. The relevant axis/spindles are determined via machine data.

Backlash compensation

During power transmission between a moving machine part and its drive (e.g. ball screw), there is normally a small amount of backlash because setting mechanical parts so that they are completely free of backlash would result in too much wear and tear on the machine. In the case of axes/spindles with indirect measuring systems, mechanical backlash results in corruption of the traverse path, causing an axis, for example, to travel too much or too little by the amount of the backlash when the direction of movement is reversed.

To compensate for backlash, the axis-specific actual value is corrected by the amount of the backlash every time the axis/spindle reverses its direction of movement.

If a second measuring system is available, the relevant backlash on reversal must be entered for each of the two measuring systems. Backlash compensation is always active in all modes following reference point approach.



Positive backlash (normal case)
The actual encoder value is ahead of the true actual value (table):
The table does not travel far enough

Basic offsets in the WCS workpiece coordinate system

→ Zero offsets

With HMI-Advanced/MMC 103 system software, you can define up to 16 channel-specific and 16 global basic frames which are then effective for all part programs.

Block search

For testing part programs or following interruption of machining, it is possible to select any point in the part program using the block search function in order to start or resume at this point.

You have a choice of 4 different search options:

- Block search with calculation at the contour line: during the block search, the same calculations are executed as during normal program operation. The destination block is then traversed true-to-contour until the end position is reached. Using this function it is possible to approach the contour again from any situation.
- Block search with calculation at the block end position: this function allows you to approach a target position (such as tool change position). All calculations are also executed here as during normal program operation. The approach targets the end position of the destination block or the next programmed position using the method of interpolation valid in the destination block.

- Block search without calculation:

This method is used for high-speed searches in the main program. No calculation are carried out during the search. The internal control values remain the same as before the block search.

- External block search without calculation:

In the menus »Search position« and »Search pointer«, you can use the softkey »External without calc.« to start an accelerated block search for programs which are executed by an external device (local hard disk or network drive).

You can specify the target of the search by

- Directly positioning the cursor to the destination block, or
- By specifying a block number, a label, a string, a program name or a line number.
- A cascaded block search is also possible starting with software release 6.2.

Cartesian PTP travel

For handling and robot-related tasks, two types of movement are required: either in the Cartesian coordinate system (continuous path, CP), or point-to-point (PTP) travel. With PTP, the shortest way to reach the target point is with activated (!) transformation TRAORI. PTP generates a linear interpolation in the axis space of the machine axis. By trueing from PTP to CP movement, it is possible to switch with optimum timing from fast feed to a mounting or positioning movement.

PTP travel does not result in an axis overload when traveling through a singularity (such as the changing of an arm position during handling).

PTP travel is also possible in JOG mode, and does not require conversion of Cartesian coordinates (e.g. from CAD systems) into machine axis values. The Cartesian PTP travel is also used for cylindrical grinding machines with inclined axis: with active transformation, the infeed axis can be moved either according to Cartesian coordinates or at the angle of the inclined axis.

Circle via center point and end point

Circular interpolation causes the tool to move along a circular path in a clockwise or counter-clockwise direction. The required circle is described by:

- Starting point of circle (actual position in the block before the circle)
- Direction of rotation of circle
- End point of circle (target defined in circle block)
- Circle center point

The circle center point can be programmed as an absolute value with reference to the current coordinate zero or as an incremental value with reference to the circle starting point.

If the aperture angle is apparent from the drawing, then it can be directly programmed.

In many cases, the dimensioning of a drawing is chosen so that it is more convenient to program the radius to define the circular path. In the case of an arc of more than 180 degrees, the radius specification is given a negative sign.

Circle via intermediate point and end point

If a circle is to be programmed which does not lie in a paraxial plane but obliquely in space, an intermediate point can be used to program it instead of the circle center. Three points are

required to program the circle. They are the starting point, the intermediate point, and the end point.

Glossary

Functions and Terms

SINUMERIK 810D/840D/840D

Clamping monitor

The "clamping monitor" is one of SINUMERIK's many extensive monitoring mechanisms for axes.

When an axis is to be clamped following conclusion of the positioning procedure, you can activate the clamping monitor with the PLC interface signal "clamping in progress".

This may become necessary because it is possible for the axis to be pushed beyond the zero-speed tolerance from the setpoint

→ Positioning monitoring, standstill monitoring

position during the clamping procedure. The amount of deviation from the setpoint position is set via the machine data.

During the clamping procedure, the clamping monitor replaces the zero-speed monitor, and is effective for linear axes, rotary axes, and position-controlled spindles. The clamping monitor is not active in follow-up mode. When the monitor responds, its reactions are the same as those of the zero-speed monitor.

Clearance control

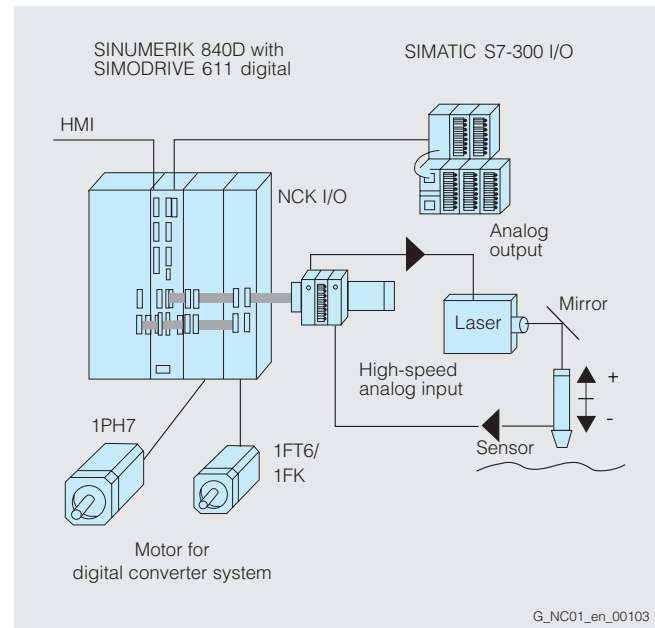
Clearance control makes it possible for sensor signals, for instance, to be evaluated via the NCK I/O's high-speed analog input (A/D conversion: 75 µs).

The "clearance control 1D in the IPO cycle" is used to compute a position offset \$AA_OFF for an axis via synchronous action.

The "clearance control 1D/3D in the (LR) position control cycle" (which includes the IPO cycle) controls three machine axes as well as a gantry axis and makes it possible to automatically maintain the constant clearance that is technologically required for the machining process.

The most important applications for this are water jet cutting and laser cutting, for example the radial cutting of rods with non-circular cross-sections.

»Limited functionality with SINUMERIK 840DE: only clearance control 1D in the position control cycle«.



Components for setting up for laser machining with SINUMERIK 840D

CNC program messages

→ Alarms and messages

All messages programmed in the part program and all alarms recognized by the system are displayed on the operator panel in plain text. The display is divided into alarms and messages.

You can program messages in order to provide the operator with the latest information on the current machining situation during the program run.

CNC user memory

All program and data, such as part programs, subroutines, comments, tool compensations, and zero offsets/frames, as well as channel- and program user data can be stored in shared CNC user memory. CNC user memory is battery-backed.

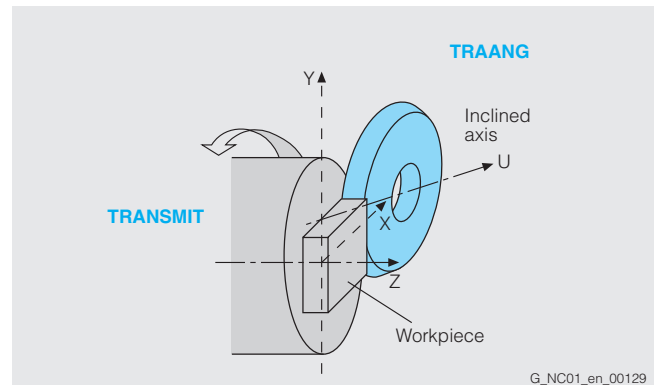
13

Concatenated transformations

With the TRACON, two transformations can be concatenated: TRAANG (inclined axis), as base transformation, can be linked with TRAORI (5-axis transformation), TRANSMIT (end face machining of turned parts), or TRACYL (cylinder surface transformation).

Applications:

- Rotary milling with mechanically non-orthogonal Y axis to X, Z (inclined-bed rotary milling machine)
- Grinding of contours programmed with TRACYL (cylinder processing)
- Dressing of a distorted contour created with TRANSMIT



Grinding a TRANSMIT contour with inclined axis

Connection for SIMATIC HMI via PLC

All PLC variables (inputs, outputs, memory bits, data values, timers, counters, and so on) can be displayed on the SIMATIC HMI operator panel.

It is only currently possible to access the CNC variables from the OP7/17. Further versions will be available soon.

Continue machining at the contour (retrace support) (option)

When using 2D flat bed cutting procedures, e.g. laser, oxygen or water jet cutting, the machine operator can return to the program continuation point (damage point) following an interruption in machining without exact knowledge of the parts program in order to continue machining the workpiece from there.

The functionality »Retrace support« contains a ring buffer for the geometric information of the executed blocks.

A new parts program is generated from this for the reverse traverse. Retracing is used e.g. when the machine operator only notices the failure or interruption a few blocks after the actual interruption. The head has usually already progressed further in the machining, and must therefore be appropriately returned for continuation of machining.

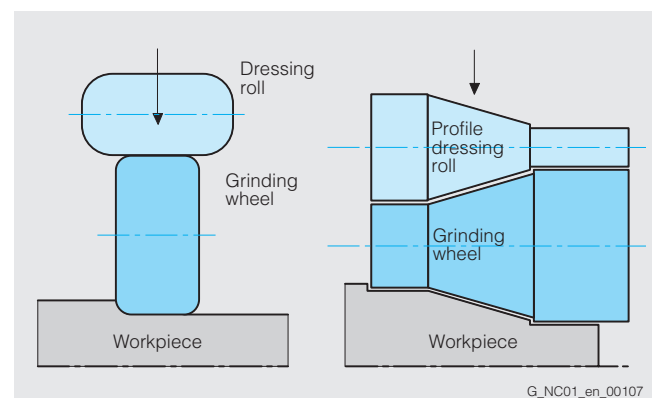
Continuous dressing

With this function, the form of the grinding wheel can be dressed in parallel with the machining process. The grinding wheel compensation resulting from dressing the wheel takes immediate effect as length compensation.

When the tool radius compensation is programmed to machine the contour and the tool radius changes because of the dressing of the grinding wheel, the CNC computes the dressing amount online as true tool radius compensation.

Functionality restrictions on the SINUMERIK 810DE/840DiE/840DE:

Only one measured variable (e.g. actual axis value, analog input) can be evaluated and subsequently only one correction made (e.g. axis correction during dressing of the grinding wheel).



Continuous dressing

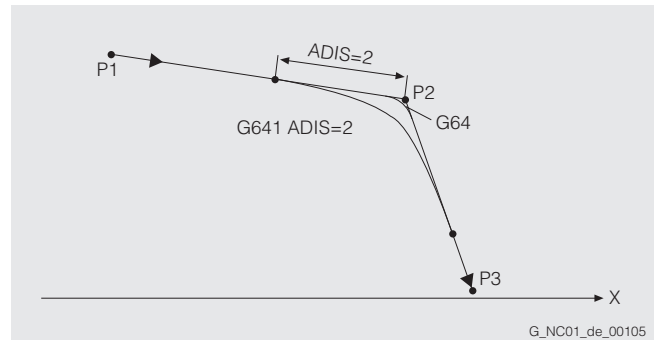
Glossary

Functions and Terms

SINUMERIK 810D/840D/840D

Continuous-path mode with programmable rounding clearance

The aim of continuous-path mode is to avoid excessive deceleration at the block boundaries and to achieve as constant a speed as possible during tangential transitions from one block to the next. Because the tool does not stop at block boundaries, no milling marks are made on the workpiece. In continuous-path mode (G64), speed reduction takes place and corners are rounded on non-tangential transitions. A softer contour transition without a jump in acceleration can be programmed with G641 ADIS=...



Continuous-path mode with programmable rounding clearance

Contour definition programming

Contour definition programming allows you to quickly input simple contours.

With the aid of help displays in the editor, you can program 1-point, 2-point or 3-point definitions with the transition elements

chamfer and fillet quickly and easily by entering Cartesian coordinates and/or angles.

Contour handwheel

When the "contour handwheel" function is activated, the handwheel has a velocity-generating effect in the AUTOMATIC and MDA modes on all programmed traversing movements of the path and synchronous axes.

A feedrate specified via the CNC program becomes ineffective and a programmed velocity profile is no longer valid. The feedrate is given in mm/min from the pulses of the handwheel on the basis of the pulse evaluation (machine data) and the active

→ Feedrate interpolation

increment. The direction of rotation of the handwheel determines the direction of travel:

- Clockwise:
In the programmed direction of travel (even beyond block boundaries)
- Counter-clockwise:
Against the programmed direction of travel (continuation beyond the start of the block is prevented).

Contour monitoring

The following error is monitored for a defined tolerance band to ensure machining precision. An impermissibly high following error might be caused by a drive overload, for example. If an error occurs, the axes/spindles are stopped.

"Contour monitoring" is always enabled when a channel is active and in position-controlled mode.

→ Travel to fixed stop

If the channel is interrupted or in the reset state, the contour monitor is not active.

Contour monitoring is also deactivated during execution of the "travel to fixed stop" function.

Contour monitoring with tunnel function (option)

With the function "contour monitoring with tunnel function", the absolute movement of the tool tip in space can be monitored in 5-axis machining or when complex workpieces are being machined. This function provides optimum protection of expensive workpieces. A cylindrical tunnel (tolerance field) with a definable diameter is placed around the programmed path.

If during machining, the deviation from the path is greater than the defined tunnel diameter, the axes are brought to a standstill immediately.

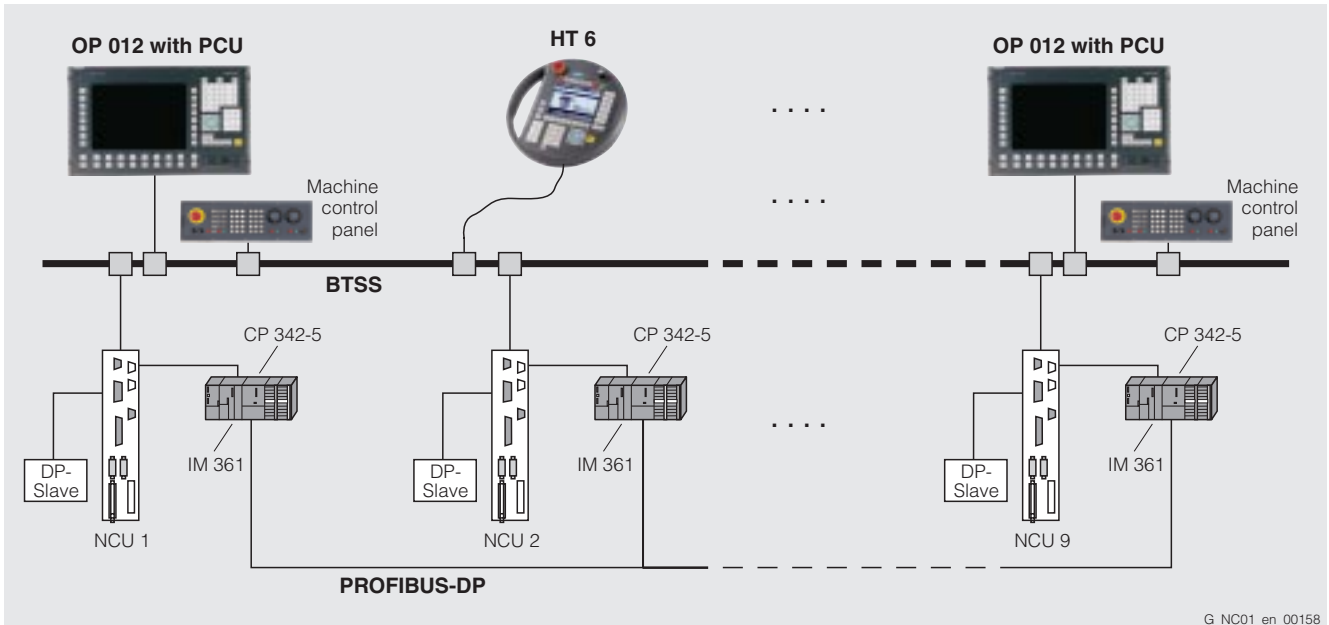
The deviation from the path can be written simultaneously to an analog output.

13

Control unit management (option)

In SINUMERIK control systems, several control units (M) can be allocated to multiple CNC controls (N) via a shared bus (BTSS/ MPI) with the M:N link.

In the basic configuration, up to 8 NCUs can be controlled by one PCU/MMC. The "control unit management" option makes it possible to operate up to 9 NCUs on up to 9 PCUs (MMCs) via active, passive and displacer mechanisms.



M:N link with SINUMERIK 840D

G_NC01_en_00158

Cross-mode actions (option)

Asynchronous subroutines make it possible to respond immediately to high-priority events not only during program execution, but in all modes and program states.

In the case of such an interrupt, it is also possible to start an asynchronous subroutine in manual mode.

→ Interrupt routines with fast retraction from the contour

The subroutine can be used, for example, to bring the grinding wheel to a safe position.

This option also enables statically effective synchronous actions IDS which are active in all operating modes.

Cycle storage separate from CNC user memory (option)

Files which have not been modified online (e.g. Siemens and machine manufacturer cycles) can be relocated using this function from the SRAM into a DRAM file system for cycle storage. More space is then available in the SRAM for parts programs.

The function can only be used in combination with HMI-Advanced.

Cycle support

The technology cycles for drilling, milling and turning and the measuring cycles are supported by cycle screen forms.

→ Expand operator interface

Similar input screen forms are also available for geometric contour programming. You can, however, also define a number of softkeys, input fields and displays yourself using the functionality of "expand operator interface".

Glossary

Functions and Terms

SINUMERIK 810D/840Di/840D

Data backup

Software is delivered on diskette or CD-ROM, or installed on the hardware.

Diskettes or CD-ROMs (for large data quantities, e.g. PCU 50 with hard disk) are used as the medium for data backup. The following data backup methods are available.

Data management software:

ADDM – Automation and Drives Data Management™ for PG or PC, including server link.

- 810D/840Di/840D/PCU 20 (MMC 100.2):

- System software and user data via V.24 serial interface with SinuCom PCIN or PCIN to PC/PG (from PC/PG via CD writer to CD-R or with SINUCOPY to PC card for PCU 20/ MMC 100.2, CCU or NCU)

- 810D/840Di/840D/PCU 50 (MMC 103):

- System software and user data via parallel interface (Centronics) with Ghost to PC/PG (from PC/PG via CD writer to CD-R or with SINUCOPY to PC card for PCU 20 (MMC 100.2), CCU or NCU)
- System software and user data via V.24 serial interface with SinuCom PCIN or PCIN to PC/PG
- Ethernet for PCU 50
- Data via floppy drive to diskette
- Removable hard disk

Data interchange between machining channels

→ High-level CNC language

In the "program coordination" function, variables shared by the channels (NCK-specific global variables) can be used for data

interchange between the program. The program message itself is separate for each channel.

Diagnostic functions

A self-diagnostics program and test aids for service have been integrated in the controls. The status of the following can be displayed on the operator panel:

- Interface signals between the CNC and the PLC and between the PLC and the machine
- Data blocks
- PLC memory bits, timers and counters
- PLC inputs and outputs

For testing purposes, signal combinations can be set for the output signals, input signals, and memory bits.

All alarms and messages are displayed in plain text on the operator panel with the corresponding acknowledgment criterion. Alarms and messages are displayed separately.

In the "service display" menu it is possible to call up important information about the axis and spindle drives, such as:

- Absolute actual position
- Setpoint position
- Following error
- Setpoint speed
- Actual speed
- Trace of CNC and drive variables

Differential resolver function (DRF)

→ Handwheel override

The differential resolver function generates an additional incremental zero offset in AUTOMATIC mode via the electronic hand-

wheel. This function can be used, for example, to correct tool wear within a programmed block.

Dimensional notation in metric and inch

Depending on the measuring system used in the production drawing, you can program workpiece-related geometrical data in either metric measure (G71) or inches (G70). The control can be set to a basic system regardless of the programmed measuring system. You can have the control convert the following geometrical data into the opposite system, and thus enter them directly (examples):

- Path information X, Y, Z ...
- Interpolation parameters I, J, K and circle radius CR

- Thread pitch
- Programmable zero offset (TRANS)
- Polar radius RP

With the G700/G710 programming expansion, all feedrates are also interpreted in the programmed measuring system (inch/min or mm/min). In the "machine" control area, you can also switch back and forth between inch and metric notation using a softkey.

Display functions

All current information can be displayed on the operator panel's screen, such as:

- Block currently being executed
- Previous and following block
- Actual position, distance to go
- Current feedrate
- Spindle speed (option)
- G functions
- Auxiliary functions
- Workpiece name
- Main program name
- Subroutine name

- All data entered, such as part programs, user data and machine data
- Help texts

Important operating states are displayed in plain text, for example

- Alarms and messages
- Position not yet reached
- Feed hold
- Program in progress
- Data input/output in progress

Dynamic block buffer (FIFO)

The traversing blocks are readied prior to execution and stored in a dynamic block buffer (FIFO = first in/first out) of specifiable size. In contour sections that are machined at high speed with short path lengths, blocks can be executed from this buffer at very high speed. The dynamic block buffer is constantly reloaded during execution.

Block execution can be interrupted with the STARTFIFO command until the block buffer has been filled, or STOPFIFO (start high-speed machining section) or STOPRE (stop buffering) can be programmed.

Electronic gear (option)

The "electronic gear" function allows highly accurate kinematic coupling of axes with programmable gear ratio. Couplings can be specified and selected for any CNC axis via program or operator panel.

The "electronic gear" function makes it possible to control the movement of a following axis in dependence on up to five leading axes.

The connection between the leading axes and the following axis is defined for each leading axis by a fixed gear ratio (numerator/denominator) or as a linear or non-linear coupling using a curve

table. The following axis can be a leading axis for another gear system (cascading). Real as well as simulated linear or rotary axes can be used as the leading and following axes. Master input values can be setpoints generated by the interpolator (setpoint linkage) or actual values delivered by the measuring system (actual-value linkage). The electronic gears with non-linear linking available starting with software release 6 of the SINUMERIK 840D also permit e.g. compensation of non-linear properties of the process in addition to the manufacture of convex teeth when machining gear wheels.

Electronic handwheels (accessory)

Using electronic handwheels it is possible to move selected axes simultaneously in manual mode. The meaning of the lines on the handwheels is defined by increment weighting. If coordinate offset or coordinate rotation is selected, it is also possible to move the axes manually in the transformed workpiece coordinate system. The input frequency of the handwheel inputs is 100 kHz.

A third handwheel can also be operated over the actual-value input of the SIMODRIVE 611 digital's closed-loop control plug-in units or the CCU unit.

The function »Contour handwheel« permits use of a handwheel on conventional turning machines (applications for ManualTurn and ShopTurn) and also during grinding for traversing on a contour.

Once the "contour handwheel" function has been activated, the handwheel has a velocity-generating effect in AUTOMATIC and MDA mode, that is, a feedrate specified via the CNC program is no longer effective, and a programmed velocity profile is no longer valid. The feedrate, in mm/min, results from the handwheel pulses as based on pulse evaluation (via machine data) and the active increment (INC1, INC10, etc.).

The handwheel's direction of rotation determines the direction of travel: clockwise in the programmed direction, even over block boundaries, and counter-clockwise up to the block start.

Electronic transfer (option)

In presses with step tools as well as in large-parts transfer presses, a modern electronic transfer system handles part transport. Positioning drives are controlled in step with the press's main motions. The "electronic transfer" option makes it possible to control sequences of motions in transfer systems (such as gripper or suction lines, etc.) in dependence on a control value which corresponds to the current plunger position of the press. The "electronic transfer" option contains suboptions for "position signals/cam group", "polynomial interpolation", "master value coupling and curve table interpolation", "cross-mode actions", "I/O interfacing via PROFIBUS-DP", "synchronous actions stage 2", as well as two "pairs of synchronized axes (gantry axes)" and one "additional positioning axis".

Combinations of these operations satisfy any and all requirements for highly dynamic and accurate transfer control.

When using the "electronic transfer" option, the "spindle" and "tool compensation" functions cannot be activated.

Functionality limitations on the SINUMERIK 840DE/840DiE: see the functional limitations for each of the above-mentioned functions.

Glossary

Functions and Terms

SINUMERIK 810D/840DI/840D

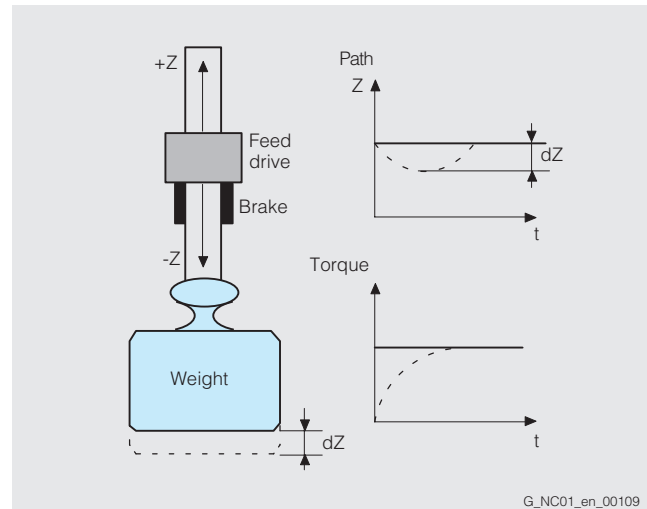
Electronic weight compensation (option)

With weight-loaded axes without mechanical or hydraulic weight compensation, the vertical axis drops when the brake is released and the servo enable is switched on.

The undesired drop (dZ) of the axis can be compensated by activating electronic weight compensation. After releasing the brake, the constant weight compensation torque maintains the position of the vertical axis.

Sequence:

1. Brake holds Z axis
2. Brake is released; servo enable on; pulse enable on
3. Z axis does not drop, holding its position



Electronic weight compensation

— With electronic weight compensation

- - - Without electronic weight compensation

Evaluation of internal drive variables (option)

With the "evaluation of internal drive variables" function, a second process variable (such as a path-specific or axis-specific feedrate) can be controlled (adaptive control) in dependence on a measured process variable (such as spindle current).

This permits, *inter alia*, the cutting volume to be kept constant when grinding, or faster covering of the grinding gap when scratching (»first touch«). Evaluation of these drive variables also permits machines and tools to be protected from overloading, as well as achievement of shorter machining times and an improved surface quality for the workpieces.

The »Evaluation of internal drive variables« is the prerequisite for implementation of adaptive control. Adaptive control can be parameterized within the parts program in the following manner:

- Additive influencing: the programmed value (F word) is corrected by adding.

- Multiplicative influencing: the F word is multiplied by a factor (override).

The following real-time variables can be evaluated as internal drive variables:

\$AA_LOAD Drive capacity utilization in %

\$AA_POWER True drive power in W

\$AA_TORQUE Driving torque setpoint in Nm
(actual power value in N only with SIMODRIVE 611 digital/ with hydraulic linear drives HLA)

\$AA_CURR Actual axis/spindle current in A

Functionality restrictions on the SINUMERIK 810DE/840DE: Only one measured variable (e.g. spindle current) can be evaluated at a time.

Execution from hard disk

Extremely long part programs, or programs which no longer fit in the CNC program memory, can be saved on the hard disk and also executed from there.

This can also be carried out in several channels.

You can use the »EXTCALL« command to also call programs from the hard disk for cascading. This »Execution from hard disk« has an effect beyond a reset or the end of a part program, and is only terminated by selection of a program which is

located in the CNC program memory. To process the subroutines from hard disk, a FIFO buffer (first in/first out) whose size can be adjusted using machine data is organized on the CNC.

Note concerning all above-mentioned forms of this external execution:

If a part program is executed more rapidly than further data can be provided externally (e.g. via V.24 interface), the CNC waits for further data without sending an alarm.

→ PC card as additional program memory

Execution from network drive or PC card

Execution of extremely long part programs is possible via a network server.

PCU 20, PCU 50 and PCU 70 already have the Ethernet interface onboard.

With the PCU 20, you require the option »Administration of network/disk drives for PCU 20«, and with the PCU 50 and PCU 70 the optional SINUMERIK software SinDNC (see Section 4).

The PC card plug-in present with the PCU 20 can also be used as an additional program memory together with a PC card.

Execution via the V.24 interface

Part programs that are too large for the CNC program memory can be processed using the HMI-Embedded/MMC 100.2 sys-

tem software via the V.24 interface in punched-tape format and simultaneously executed.

Expand user interface

"Expand user interface" functionality allows the SINUMERIK user to design his own user environments, expand the environments provided by machine manufacturers or end users, or simply design his own screen form layouts.

User environments configured by Siemens or non-Siemens machine manufacturers can be modified or replaced. The function is implemented via an integrated interpreter and via configuring files containing the description of the user environment.

The interpreter is available for HMI-Advanced/MMC 103 system software, HMI-Embedded/MMC 100.2 system software, Manual-Turn, ShopMill, ShopTurn and HT 6.

The screen forms can be designed right on the control itself. A graphic tool is required to create graphics and pictures. Part programs can be processed with newly created operator environments.

Configuring examples for new screen forms, which can also be used as the basis for the user's own new screen forms, can be found in the accompanying toolbox.

You can implement the following functions with "expand user interface":

- Display screen forms and softkeys, variables, tables, texts, help texts, graphics, and help displays.

- Start actions when screen forms are displayed and exited, actuate softkeys, and enter values (variables).
- Dynamically restructure screen forms, including changing softkeys, designing arrays, and displaying, replacing and deleting texts and graphics.
- Read and write variables, combine with mathematical, comparative or logical operators.
- Execute subroutines, file functions, program instance services (PI services) or external functions (HMI-Advanced/MMC 103 system software).
- Enable data interchanges between screen forms.
- "Expand operator interface" is configured using ASCII files that can be stored on the PCU/MMC. Files are interpreted which contain ASCII descriptions for the layout of screen forms, softkey functions, and texts and graphics. These configuring files are created with the ASCII editor, taking into account certain special rules of syntax.

With the integral editor, the basic version of the user interface can already be expanded at predefined softkeys by up to 20 pictures (more than 20 pictures with OA copy license, see Section 4).

Extended stop and retract (ESR), drive-independent (incl. generator operation) (option)

A safe position is assumed from the machining level without any collision between tool and workpiece.

As an extension to the independent drive stop/retract function possible with SW release 5, SW release 6 now offers the functionality »CNC-controlled stop/retract«.

To permit gentle interpolated retraction on the path or contour, the path interpolation can be processed further for a definable period following the triggering event. The retraction axes are subsequently driven in synchronism to an absolute or incremental position as programmed.

These functions are primarily used for gearing and grinding technologies.

Fast-IPO-Link (option)

Non-circular machining can be carried out for general workpiece contours using polynomial interpolation or, with sinusoidal inputs, using master value coupling and curve table interpolation.

In the case of very fast non-circular machining, »Fast-IPO-Link« permits transfer of the non-circular task (e.g. movement of X-axis) to a separate NCU with fast cycle. Speeds greater than 3000 rpm (for sinusoidal movements) can then be achieved.

Glossary

Functions and Terms

SINUMERIK 810D/840D/840D

Feedforward control, acceleration-dependent (option)

Using the function "feedforward control", you can reduce axial following errors almost to zero. Feedforward control is therefore sometimes called "following error compensation". Particularly during acceleration in contour curvatures, e.g. circles and corners, this following error leads to undesirable, speed-dependent contour errors.

The SINUMERIK controls therefore feature two different forms of feedforward control:

- Velocity-dependent speed feedforward speed:
In velocity-dependent feedforward control, the following error can be reduced almost to zero at constant speed.

- Acceleration-dependent torque feedforward control:
In order to achieve precise contours even when the demand for dynamics is at its highest, you can use torque feedforward control. If the settings are right, you can compensate the following error almost completely, even during acceleration. The result is extraordinary machining precision even at high tool path feed-rates.

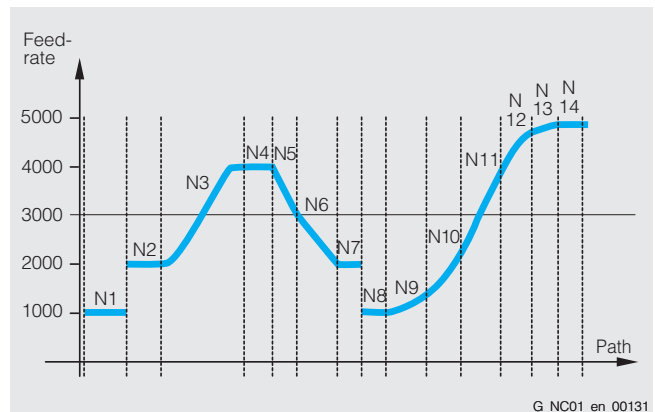
Feedrate interpolation (feed characteristic)

→ Polynomial interpolation

In accordance with DIN 66025, a constant feed-rate over the part program block can be defined via address F. For a more flexible definition of the feedrate profile, programming to DIN 66025 is extended by linear and cubic profiles over the path. The cubic profiles can be programmed directly or as interpolating spline.

This makes it possible, depending on the curvature of the workpiece to be machined, to program continually smooth speed profiles which in turn allow jerk-free acceleration changes and thus the production of uniform workpiece surfaces. You can program the following feed-rate profiles:

- FNORM:
Behavior as per DIN 66025 (default setting). A F value programmed in the CNC block is applied over the entire path of the block, and is subsequently regarded as modal value.
- FLIN:
An F value programmed in the block can be traversed linearly (rising or falling) over the path from the current value at the beginning of the block to the end of the block, and is subsequently regarded as modal value.
- FCUB:
The non-modally programmed F values, referred to the end of the block, are connected through a spline. The spline starts and ends tangentially to the previous or following feedrate setting.
- FPO:
You can also program the feedrate profile directly via a polynomial. The polynomial coefficient is specified analogous to polynomial interpolation.



Programming example for feedrate interpolation

- N1 Constant feedrate profile F1000 : FNORM
- N2 Abrupt setpoint speed change F2000 : FNORM
- N3 Feedrate profile via polynomial : $F = FPO(4000, 6000, -4000)$
- N4 Polynomial feedrate 4000 as modal value
- N5 Linear feedrate profile F3000 : FLIN
- N6 Linear feedrate 2000 as modal value
- N7 Linear feedrate, as modal value
- N8 Constant feed profile with abrupt acceleration change F1000 : FNORM
- N9 All subsequent F values are linked by splines F1400 : FCUB
- N13 Switch off spline profile
- N14 FNORM

Feedrate override

The current velocity is overlaid the programmed velocity via machine control panel or by the PLC (0% to 200%). In order for the cutting speed on the contour to be kept constant, the feedrate calculation is referred to the working point or tool end

point. The feedrate can also be corrected by a programmable percentage factor (1% to 200%) in the part program. This factor is overlaid (multiplication) on the setting made on the machine control panel. The velocity setting from the PLC is axis-specific.

Follow-up mode

If an axis/spindle is in follow-up mode, it can be moved externally, and the actual value can still be recorded.

The traverse paths are updated in the display. Standstill, clamping and positioning monitoring functions are not effective in follow-up mode. Following cancellation of follow-up mode, it is not necessary to carry out a reference point approach again.

Frame concept

Frame is the common term for a geometric expression describing an arithmetic operation, e.g. translation or rotation.

With SINUMERIK controls, the frame in the CNC program converts from one Cartesian coordinate system into another, and represents the spatial description of the workpiece coordinate system.

The following are possible:

- Basic frames:
Coordinate transformation from basic coordinate system (BCS) into basic zero system (BZS)
- Adjustable frames:
Work offsets using G54 to G57/G505 to G599
- Programmable frames:
Definition of workpiece coordinate system (WCS)

The frame concept makes it possible to transform perpendicular coordinate systems very simply by offsetting, rotating, scaling and mirroring.

The following instructions are used to program these options:

- TRANS Programmable zero offset
- ROT Rotation in space or in a plane
- ROTs Rotation referred to the solid angle projected into the planes
- SCALE Scaling (scale factor)

- MIRROR Mirroring
- TOFRAME Frame according to tool orientation
- TOROT Rotary component of programmed frame
- PAROT Frame for workpiece rotation (table rotation)
- MEAFRAME Frame calculation from 3 measuring points in the space (for measuring cycles)

The instructions can also be used several times within one and the same program. Existing transformations can either be overwritten or added.

Additive frame instructions:

- ATRANS Additive programmable zero offsets
- AROT Additive rotation in space or in a plane
- ASCALE Scale factor (multiplication)
- AMIRROR Repeated mirroring
- AROTS additive rotation based on the solid angle configured in the planes

If swivelling tools or workpieces are available, machining can be implemented very flexibly, for example

- by machining several sides of a workpiece by rotation and swivelling of the machining plane
- by machining of inclined surfaces using tool length and tool radius compensation

From software version 5 and higher NCK-global frames are also available for all channels of an NCU.

Generator operation (option)

With the "generator operation" function, brief power outages can be bridged or power provided for retraction. To make this possible, the energy stored during spindle rotation or axis movement

is fed back into the DC link, the same principle as that used by generators.

Generic transformation

The function »Generic transformation« is used to define the tool orientation as desired in the space with the initial setting of the axes, and not just according to the Z-direction.

It can then be used more flexibly and universally. It is then possible, for example, to also control machine kinematics by the

CNC where the orientation of the rotary axes is not exactly parallel to the linear axes.

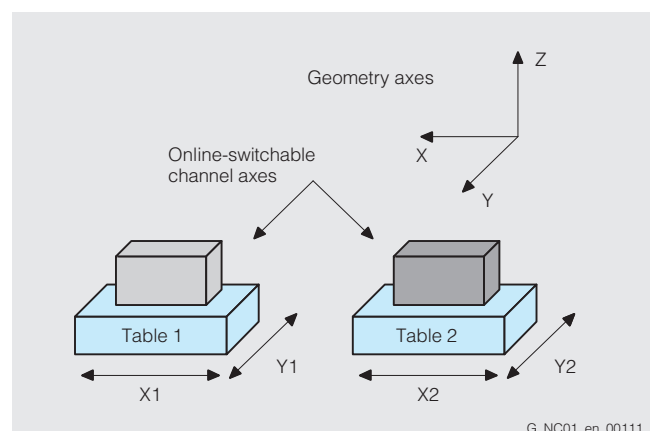
Starting with software release 6, extension of the generic 5-axis transformation to the 3-axis and/or 4-axis transformation is also possible for machines with only one rotary axis (rotatable tool or workpiece).

Geometry axes, switchable online in the CNC program

In the CNC, geometry axes form axis groupings per channel for the interpolation of path motions in the space.

Channel axes are assigned to geometry axes via machine data.

With the "switchable geometry axes" function, it is possible, from the part program, to assemble the geometry axis grouping from other channel axes. This makes problem-free operation of machine kinematics with parallel axes possible.



Geometry axes, switchable online

Glossary

Functions and Terms

SINUMERIK 810D/840D/840D

Grinding wheel surface speed, constant

Automatic conversion of the grinding wheel surface speed to a revolution speed as a function of the current grinding wheel diameter. This function can be active for several grinding wheels simultaneously in one CNC channel. The grinding wheel surface speed is monitored at the same time.

A constant grinding wheel surface speed is not only useful during processing of a part program in the AUTO and MDA modes, but can also be effective immediately after power-up of the control, on reset, and at the end of the part program, and remain in force beyond all mode changes (depending on the machine data).

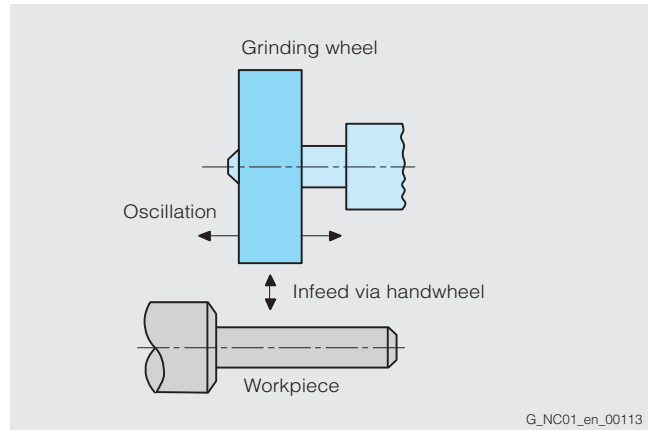
Handwheel override

With the function handwheel override an axis can be traversed or the speed of an axis modified non-modal.

The function is effective on a block basis.

At the same time, additional axes can be interpolated or traversed simultaneously. The actual-value display is continuously updated.

Used: grinding machines.



Handwheel override in AUTOMATIC mode

Helical interpolation

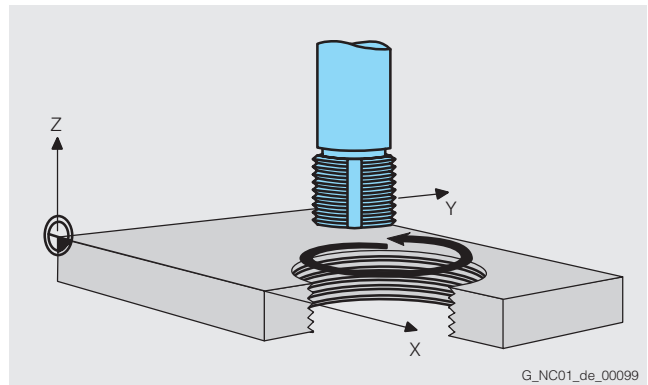
Helical interpolation is especially suitable for machining inside or outside threads with profiling cutters and for milling lubrication grooves. The helix consists of two motions:

- A circular movement in one plane
- A linear movement perpendicular to this plane

The programmed feed F either refers only to the circular motion or to the total path velocity of the three CNC axes involved.

In addition to the two CNC axes performing circular interpolation, other linear motions can be performed. The programmed feed F refers to the axes specially selected in the program.

Interpolation with more than 4 axes requires export approval.



Helical interpolation: Thread cutting with profiling cutter

High-level CNC language

To meet the various technological demands of modern machine tools, a CNC high-level language has been developed for SINUMERIK that provides a high degree of programming freedom.

System variables

The system variables can be processed in the CNC program (read, partially write). System variables allow access to machine data, setting data, tool management data, programmed values, and current values.

User variables

If a program is to be used flexibly, variables and parameters are used instead of constant values.

SINUMERIK gives you the option of executing all CNC functions and addresses as variables. The names of the variables can be freely defined by the user. Read and write protection can also be assigned using attributes. This means that part programs can be written in a clear and neutral fashion and then adapted to the machine, for example free selection of axis and spindle addresses.

A distinction is made between global user variables (GUDs) and local user variables (LUDs). LUDs can also be redefined via machine data to make them into program-global user variables (PUDs). They are displayed in the Parameter control area under the softkey user data, where they can also be changed.

Global user variables (GUDs) are CNC variables that are set by the machine manufacturer. They apply in all programs.

Local user variables (LUDs) are provided for initializing CNC program parameters. These variables can be redefined in every CNC program. These variables makes programming more user-friendly, and allows the user to create his own programming philosophy.

Indirect programming

Another option for the universal use of a program is indirect programming. Here, the addresses of axes, spindles, R parameters, etc., are not directly programmed, but are pointed to by a variable in which their address is entered.

Program jumps

The inclusion of program jumps allows extremely flexible control of the machining process. Conditional and absolute jumps are available as well as program branches that depend on a current value. Labels that are written at the beginning of the block are used as destinations. The destination can be before or after the block containing the jump.

Program coordination (in several channels)

Program coordination makes it possible to control the synchronism of part programs that are executing simultaneously in several CNC channels. Plain text commands allow programs to be loaded, started and stopped in several channels. Channels can be synchronized.

Arithmetic and trigonometric functions

Extensive arithmetic functions can be implemented with user variables and arithmetic variables. In addition to the four basic arithmetic operations, there are also:

- Sine, cosine, tangent
- Arc sine, arc cosine, arc tangent
- Square root
- Absolute value
- Power of 2 (squaring)
- Integer portion
- Rounding to a whole number
- Natural logarithm
- Exponential function
- Shift
- Rotate
- Scaling
- Mirroring

Comparison operations and logic operations

Comparison operations with variables can be used to formulate branch conditions. The comparison operations that can be used are:

- Equal to, not equal to
- Greater than, less than
- Greater than or equal to
- Less than or equal to
- Concatenation of strings

The following logic operations are also available: AND, OR, NOR, EXOR

These logic operations can also be performed bit by bit.

Macro techniques

Using macros, single commands of a programming language can be grouped together to form a complex command. This command sequence is given a freely definable name and can be called in the CNC program. The macro command is executed in the same way as the single commands.

Control structures

The control normally processes the CNC blocks in the order in which they are programmed.

Like program jumps, control structures allow the programmer to define alternatives and program loops. The commands make structured programming possible, and make the programs much easier to read:

- Choice of alternatives (IF-ELSE-ENDIF)
- Continuous loop (FOR)
- Program loop with start condition (WHILE)
- Program loop with end condition (REPEAT)

Glossary

Functions and Terms

SINUMERIK 810D/840D/840D

High-speed CNC inputs/outputs

The "high-speed CNC inputs/outputs" function makes it possible to read in or to output signals in the position control/interpolation cycle. The high-speed CNC inputs/outputs can be used for machines, such as those used for grinding and lasering, as well as in SINUMERIK Safety Integrated.

Input signals are possible for the following:

- Multiple feed values per block (calipers function)
The function allows modification of the feedrate through external signals. Six digital inputs can be combined with six different feedrates in a CNC block. There is no feed interruption in this case. An additional input can be used for infeed termination (starting a dwell), and yet another input can be used to start a immediate retraction movement.
Depending on the input, the retraction of the infeed axis (or axes) is initiated by a previously specified absolute value in the IPO cycle. The remaining distance-to-go is deleted.
- Multiple auxiliary functions in the block
Several auxiliary functions can be programmed in one CNC block. These functions are forwarded to the PLC in dependence on a comparison operation or in dependence on an external signal.

→ Position switching signals/cam controller

- Axis-specific deletion of the distance-to-go
The high-speed inputs affect a conditional stop and delete the distance-to-go for the path or positioning axes.
- Program branches
The high-speed inputs make program branches within a user program possible.
- Fast CNC start
Machining can be enabled conditionally in the CNC program in dependence on an external input.
- Analog calipers
Various feedrates, a dwell time, and a retraction path can be activated in dependence on an external analog input (specification of the threshold values via machine data).
- Safety-related signals such as EMERGENCY STOP

Output signals are possible for the following:

- Position signals
The position signals can be output with the function "position switching signals/cam controller".
- Programmable outputs
- Analog-value output
- Safety-related signals such as safety door interlock

High-speed data interchange between CNC and PLC

For fast, immediate information exchange between CNC and PLC, 1024 bytes are available in the communications buffer for bidirectional input/output.

Transfers are handled immediately. \$A variables are used for CNC access, and a function block with which the data in the

dual-port RAM (DPR) are immediately (rather than at the beginning of the PLC cycle) read or written is used for PLC access.

This allows you, for instance, to respond to I/O signals right away in the part program, without regard to the PLC cycle.

Higher Performance in Control & Drive

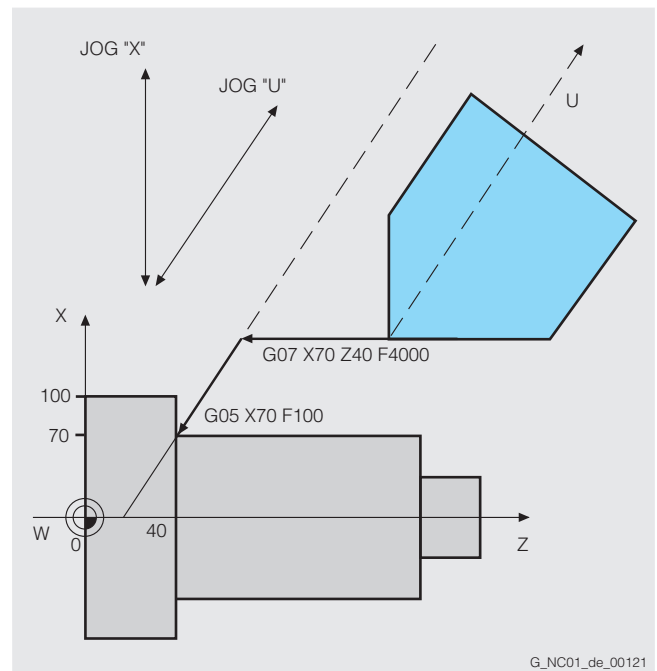
The »Higher Performance in Control & Drive« function can be used to increase the SINUMERIK 810D's performance even more. Parallel to faster interpolation cycles, the option permits improvement of drive dynamics by means of improved current and speed controller cycles.

Inclined axis (option)

The "inclined axis" function is used for fixed-angle interpolation using an oblique infeed axis (used primarily in conjunction with cylindrical grinding machines). The axes are programmed and displayed in the Cartesian coordinate system.

Tool offsets and zero offsets are also entered in the Cartesian system and transformed to the real machine axes.

For oblique plunge-cutting with G05 it is necessary to program the start position with G07. In JOG mode, the grinding wheel can be traversed either in the Cartesian coordinate system or in the direction of inclined axis U (for the number of the channel DB).

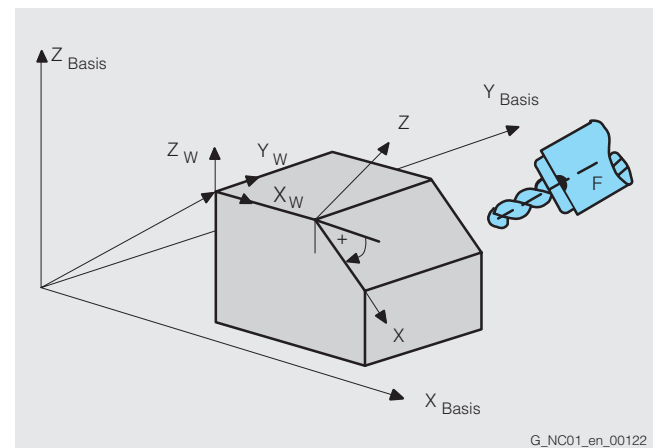


Oblique plunge-cut grinding; machine with non-Cartesian X axis (U)

Inclined-surface machining with frames

→ Frame concept

Drilling and milling operations on workpiece surfaces that do not lie in the coordinate planes of the machine can be performed easily using the function "inclined-surface machining". The position of the inclined surface in space can be defined by coordinate system rotation.



Inclined-surface machining with frames

Intermediate blocks for tool radius compensation

→ Tool radius compensation

Traversing movements with tool offset selected can be interrupted by a limited number of intermediate blocks (block without axis movements in the compensation plane).

The permissible number of intermediate blocks can be set in system parameters.

Glossary

Functions and Terms

SINUMERIK 810D/840Di/840D

Interrupt routines with fast retraction from the contour (option)

Interrupt routines are special subroutines which can be started on the basis of events (external signal) in the machining process. Any part program block currently in progress is interrupted. The positions of the axes at the time of interruption are automatically saved. It is also possible to save such things as the current states of G functions and the current offsets (SAVE mechanism), making it possible to resume the program at the point of interruption without difficulty.

Four additional program levels are available for interrupt routines, that is, an interrupt routine can be started in the 8th program level and lead as high as the 12th program level.

An interrupt (for example the switching of a high-speed CNC input) can trigger a movement via the special interrupt routine which allows fast retraction of the tool from the workpiece contour currently being machined. The angle of retraction and the absolute value of the path can also be parameterized. An interrupt routine can also be executed following the fast retraction of the tool.

Inverse-time feedrate

On the SINUMERIK, it is possible to program the time required to traverse the path of a block (rev/min) instead of programming the feedrate for the axis movement with G93.

If the path lengths from block to block are differ greatly, a new F value should be determined in every block when using G93.

When machining with rotary axes, the feedrate can also be specified in degrees/revolution.

Involute interpolation (option)

Using involute interpolation it is possible to program a spiral contour with the shape of a so-called circular involute using one CFC block instead of many approximated individual blocks.

The exact mathematical description of the contour enables a higher path velocity to be achieved, together with a reduction in machining time. Undesirable facets which could result from

coarse polygon definitions are thus avoided. Furthermore, it is unnecessary to define the end point for the involute interpolation exactly on the involute defined by the start point; it is possible to enter a maximum permissible deviation using machine data.

I/O interfacing via PROFIBUS-DP (option)

→ PLC area

PROFIBUS-DP represents the protocol profile for distributed I/O. It enables extremely high-speed cyclic communication. Due to generation of an optimum subset of the PROFIBUS message services and increasing of the data signalling rate to a maximum of 12 Mbit/s, the bus cycle times are virtually negligible. Despite all this, the many advantages of PROFIBUS, such as high availability, data integrity and standard message structure, remain unaffected.

All NCUs 561.2/571.3/572.3/573.3 in the SINUMERIK 840D and in the CCU 2/CCU 3 of the SINUMERIK 810D are equipped with a SIMATIC S7-300-compatible PLC 315-2DP. The NCUs 572.4 and 573.4 in the SINUMERIK 840D contain the PLC 314C-2 DP

compatible with the SIMATIC S7-300 with a higher performance and larger PLC user memory. Activation of the integrated PROFIBUS-DP interface connection for the NCUs/CCUs is available as separate option. The NCUs/CCUs can be operated as master or slave. Distributed I/O devices (such as the ET 200) are connected for communication purposes. Even if the interface connection integrated in the NCUs is not activated, I/O devices can be operated via a SIMATIC S7-300 equipped with an IM 361 interface module and a CP 342-5 communications processor.

In the case of SINUMERIK 840Di: the PROFIBUS-DP interface is available on the MCI board in the basic version (for I/O and drive).

Job list

You can use this function to create a job list (load list) for every workpiece to be machined.

The list contains directives for making the following preparations for executing part programs, even when multiple channels are involved:

- Parallel setup (LOAD/COPY):
Load or copy main programs and subroutines and associated data such as initialization programs (INI), R parameters (RPA), user data (GUD), zero offsets (UFR), tool/magazine data (TOA/TMA), setting data (SEA), protective areas (PRO), and sag/angularity (CEC) from the PCU's (MMC's) hard disk into the CNC's memory

- Preparations for CNC start (SELECT):
Select programs in different channels and make initial preparations for processing them
- Parallel clean-up (reverse LOAD/COPY):
Swap main programs and subroutines and associated data from CNC memory back out to the hard disk.

You can also save your own templates for job lists. Following loading and job list selection, CNC start initiates processing of all programs and data for workpiece production.

13

Languages/language expansions

Our control speaks your language! A user interface for the SINUMERIK controls is available in a number of different languages:

In the HMI-Embedded/MMC 100.2 system software and HMI-Advanced/MMC 103 system software, the basic languages for display texts and system messages and alarms are English and German.

The language expansion (German, English, French, Italian, and Spanish) is part of the HMI-Advanced/MMC 103 system software package, and can be installed in the HMI-Embedded/MMC 100.2 system software. The operator can switch back and forth between foreground and background language.

Laser switching signal, high-speed (option)

For fast lasering of such things as aperture plates, an automatic, high-speed, position-dependent signal for switching a laser on and off has been implemented in the controls. Under the prerequisite that all movements for which the laser must be switched off are made in rapid traverse mode G0, it is possible to logically combine the switching signal for the laser with the rising or falling edge of G0.

Furthermore, the laser switching signal can also be coupled to an adjustable G1 feed threshold value.

To achieve the fastest possible responses, the switching on and off of the digital laser signal is controlled by the position controller in dependence on the actual axis position.

No programming measures are required for switching the laser itself on and off, as these procedures are directly linked to the programmed G functions.

The overall procedure, however, requires programming of an Enable (at the beginning of the program) with CC_FASTON (DIFF1, DIFF2). Together with the Enable, the two offset values are entered which can offset the switching on and off of the laser by a specific path differential in relation to the setpoint position. A negative value means an offset before the setpoint position, a positive value means an offset after the setpoint position. If the programmed value is too high, that is, if the setpoint position had already been exceeded when the edge was detected, the signal is immediately switched.

Leadscrew error/measuring system error compensation

In the SINUMERIK controls, "interpolating compensation" is divided into two categories:

- Spindle error compensation (SSFK) or measuring system error compensation (MSFK) as axial compensation (basic axis and compensating axis are always identical) and
- Sag error and angularity error compensation as cross-axis compensation (basic axis affects other compensation axis)

The measuring principle of "indirect measurement" on CNC-controlled machines is based on the assumption that the lead of the ball screw is constant at every point within the traversing range, so that the actual position of the axis can be derived from the position of the drive spindle (ideal case). Tolerances in ball screw production, however, result in more or less large dimen-

sional deviations (referred to as leadscrew error). Added to this are the dimensional deviations occasioned by the measuring system used, as well as the assembly tolerances for that system on the machine (referred to as measuring system error) and any other machine-related error sources.

Because these dimensional deviations directly affect the accuracy of workpiece machining, they must be compensated for by the relevant position-dependent correction values.

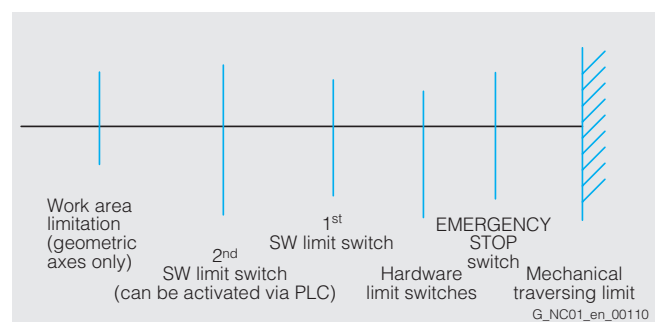
The correction values are computed on the basis of the measured error curve, and are entered in the control in the form of compensation tables on start-up. The relevant axis is compensated using linear interpolation between the intermediate points.

Limit switch monitor

Preceding the EMERGENCY STOP switch, hardware limit switches, which take the form of digital inputs controlled via the PLC interface, limit the traversing range of the machine axes. Deceleration is effected either as quick stop with setpoint zero or in accordance with a braking characteristic. The axes must be retracted in the opposite direction in JOG mode.

Software limit switches precede the hardware limit switches, are not overridden, and are not active until reference-point approach has been completed.

Following preset, software limit switches are no longer effective. A second pair of plus/minus software limit switches can be activated via the PLC.



Overview of travel limits

Glossary

Functions and Terms

SINUMERIK 810D/840Di/840D

Linear interpolation

"Linear-Interpolation" is understood to be the CNC-internal calculation of points on a straight path between the programmed starting and end points.

Up to 4 axes can already be linearly interpreted in the basic configuration of the SINUMERIK 810D/840Di/840D controls.

Optional expansions are described in Section 2 of this catalog. Limited functionality with 810DE/840DiE/840DE: interpolation with max. 4 axes.

Link axis (option)

Link axes are axes that are physically connected to another NCU and are governed by that NCU's position controller.

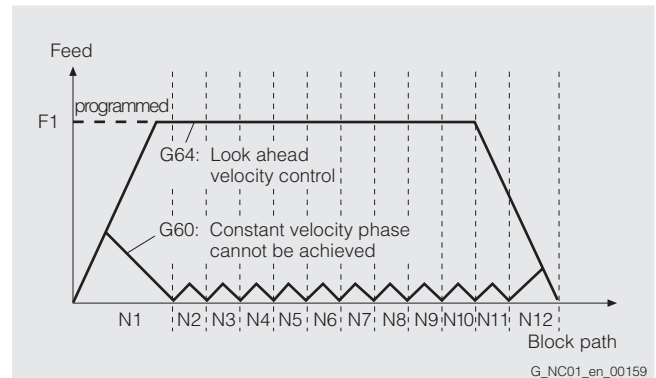
Link axes can be assigned dynamically to channels on another NCU.

Look Ahead

During the machining of complex contours, most of the program blocks describe very short paths with sharp changes in direction. If a contour of this type is processed with a fixed programmed path velocity, an optimum result cannot be obtained. In traversing blocks with tangential transitions, the drives cannot attain the required final velocity because of the shortness of the paths. Corners are cut off.

With the "look ahead" function, a specifiable number of traversing blocks is read in advance in order to calculate the optimum machining velocity. On tangential block transitions, the axis is accelerated and decelerated beyond block boundaries so that no drops in velocity occur. On sharp changes of direction, bevelling of the contour is reduced to a programmable dimension.

→ Continuous path mode with programmable rounding distance



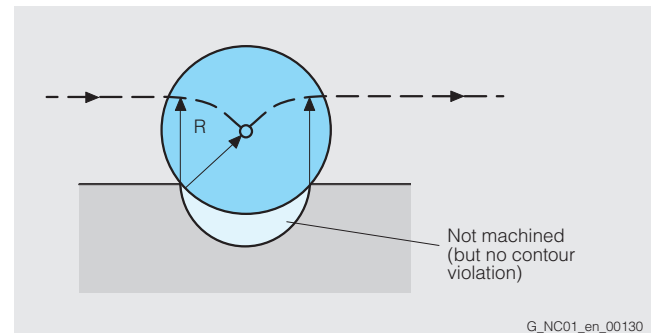
Comparison of velocity response with exact stop G60 and continuous-path mode G64 with look ahead for short paths

Look-ahead detection of contour violations

With CDON (Collision Detection ON) and active tool radius compensation, the control monitors tool paths through look-ahead detection of contour violations. This makes it possible for the control to actively detect and avert possible collisions.

The control detects the following critical machining situations, for example when the tool radius is too large, and to compensate through tool path modification.

- "Bottleneck" detection:
Because the tool radius is too large to produce a narrow inside contour, the "bottleneck" is bypassed and an alarm signalled.
- Contour path shorter than tool radius:
The tool circumvents the workpiece corner on a transition circle, then continues on the programmed path.
- Tool radius too large for inside contouring:
In such cases, the contours are machined only as much as is possible without a causing a contour violation.



Result when tool radius > circle radius

13

Machining channels

→ Mode group

Via a channel structure, parallel sequences of movements, such as positioning a loading gantry during machine, can shorten unproductive times. A machining channel must be regarded as a separate CNC control system with decoding, block preparation and interpolation. The channel structure makes it possible to process the individual channels' part programs simultaneously and asynchronously. The relevant channel, with the associated

images, is selected by pressing the "channel switch" button on the operator panel. Part programs can then be chosen and started for that specific channel.

With SINUMERIK 810D/840D each of the maximum possible channels can be operated in its own mode group.

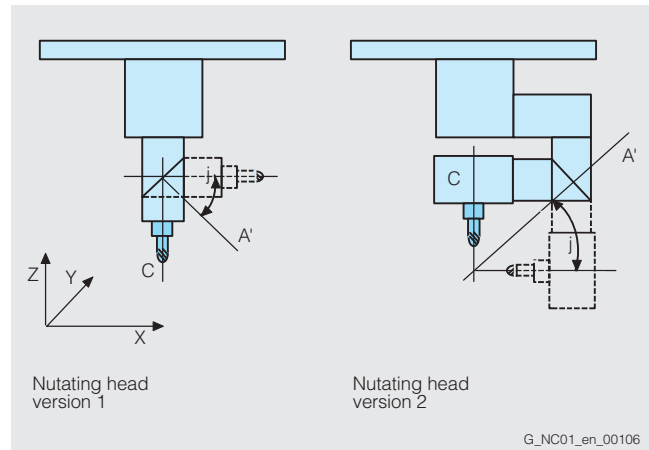
Additional machining channels are optional (see Section 2).

Machining package for 5 axes (option)

Five-axis machining tasks, such as the cutting of freeform surfaces, can be solved easily and in a user-friendly manner.

To this end, the "5-axis machining package" provides the following functions:

- 5-axis transformation with tool orientation
In 5-axis machining, geometric axes X, Y and Z are supplemented by additional axes (such as rotary axes for tilting the tool). The machining task can be completely defined in Cartesian coordinates with Cartesian position and orientation. The path vector is converted in the control with position and orientation into the machine axes via 5-axis transformation.
- 5-axis tool length compensation for 5-axis machining
When machining with the 4th/5th axis, the lengths of the selected tool are automatically included and compensated in the axis movement.
- Oriented tool retraction
If machining is interrupted (because of tool breakage, for example), a program command can be used for defined oriented tool retraction.
- Tool-oriented RTCP
With the RTCP (remote tool center point) function, the tool swivel axes can be positioned in manual mode, as long as there is compliance with the tool center point marked by the tool tip. The RTCP function simplifies the inclusion of program interpolation points in manual mode with orientation of the tool.



Nutating head

- Nutating head
Prerequisite: 5-axis machining package with 5-axis transformation. A nutating head used in conjunction with the "nutating head" function makes it possible to machine spatially formed parts at a high rate of feed. To do this, the control executes a 5-axis transformation. Three translatory main axes (X, Y, Z) determine the tool center point; two rotary axes, one of which is an inclined axis (angle can be set in the machine data), make virtually any orientation in the working space possible. Version 1 and version 2 nutating heads are supported. In the case of version 2, the position of the center point does not change when the tool is swivelled; the compensating movements required for orientation changes are minimal.

Main programs and subroutines

If machining operations recur frequently, it is advisable to store them in a subroutine. The subroutine is called from a main program (number of passes ≤9999). Eleven subroutine levels (including 3 levels for interrupt routines) are possible in a main program.

A main program can also be called from within another main program or subroutine.

Glossary

Functions and Terms

SINUMERIK 810D/840Di/840D

Master/slave for drives (option)

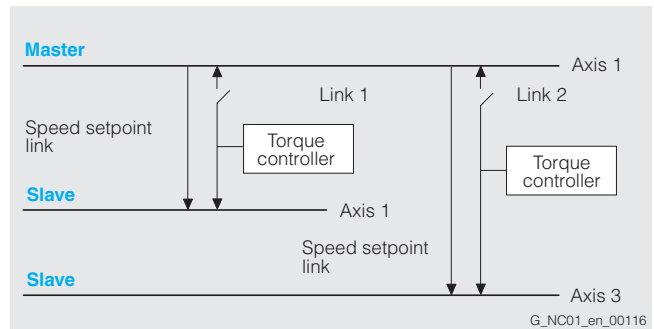
The "master/slaves for drives" function is required when two electrical drives are mechanically linked to an axis. In a link of this kind, a torque controller ensures that both drives produce the exact same amount of torque, as otherwise the two motors would work against each other. In order to attain a tensioning between master and slave, a tension torque specifiable via machine data can be applied on the torque controller.

Sample applications:

- Power gain and (occasional) mechanical linking of drives
- Drive with two motors that operate on a gear rack
- Remachining of wheel sets for rail-bound vehicles
- Backlash-free reversing of mutually tensioned drives

An axis can also be a pattern axis for multiple links.

Starting with software release 6.2, this functionality is already included in the NCU system software, i.e. a technology PC card is no longer required.



Example: axis 1 simultaneously master axis for axis 2 and axis 3

Master value coupling and curve table interpolation (option)

For special technologies (presses, transfer lines, printing machines, etc.), the replacement of mechanical, cyclic transport tasks with electronic functionality in AUTOMATIC mode requires constant coupling and decoupling functions between leading and following axes. To this end, the "synchronous spindle" functionality has been expanded to include the "master value coupling" function, which makes it possible for linear leading and following axes to be coupled via curve tables in the CNC program.

Any and all functional associations between axis positions can be approximated.

Soft coupling avoids the sudden change in velocity that occurs when the leading axis is activated. Offsets (e.g. 12 °), scaling (e.g. 1.00023) and mirroring using frame instructions are possible.

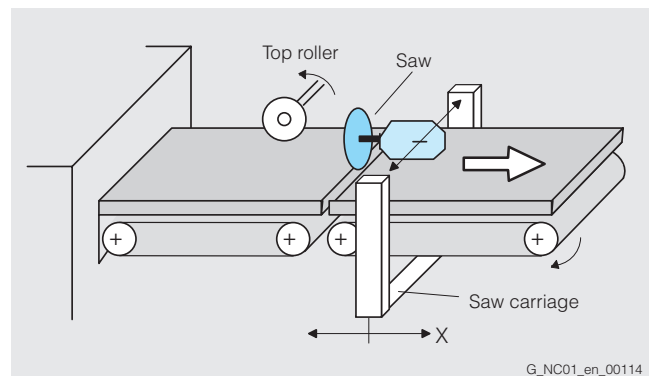
Electronic curve interpolation replaces the cam plates that were once required for the control of cyclic machines.

Complex sequences of movements can be easily defined using familiar CNC language elements. The external reference variable (e.g. "line shaft") is formed by the control's master value. The functional association between leading and following axis can be subdivided into segments of the leading axis (curve segments). In these curve segments, the link between master value and following value is described through mathematical functions (normally through 3rd degree polynomials).

So-called "cyclic machines" are distinguished by constantly repeated cyclic operations with high throughput and high productivity in machining, transport, packaging and parts handling (for example packaging machines, presses, wood processing machines, printing machines).

SINUMERIK makes it possible to implement such technological functions as synchronism, electronic transfer and positioning for cyclic machines. The mechanics (line shaft, gearing, cam discs, couplings and cams) are replaced by the electronic solution (master value coupling, curve tables, synchronous actions and electronic cams).

→ Measuring, stage 2; Synchronous spindle



Example for cyclic machines: On-the-fly saw

In addition, the electronic functionality permits fast, axis-specific optimization, high-speed phase- and path compensation, fast responses to bad or missing parts, and fast synchronization and resynchronization, as well as decoupling from the leading axis and executing autonomous movements.

Axis cycles and synchronization calculations are carried out in the IPO cycle.

Measuring from within synchronous actions, for example, is used for detecting edges on continuous workpieces and for measuring pressure marks (for example on continuous film).

Starting with software release 6.3 of the NCU 572/573, the tables can also be saved and processed in the DRAM. The memory size can be set during the user memory configuration (maximum value is system-dependent).

Limited functionality on the SINUMERIK 840DiE/840DE:

The number of simultaneously traversing axes is restricted to four.

Measuring stage 1

You can connect up to two touch probes to the control at the same time. In the case of channel-specific measuring, the measuring procedure for a CNC channel is always activated from the part program executing in the relevant channel. All of the axes programmed in the measuring block take part in the measuring procedure.

You can program a trigger event (rising or falling edge) and a measuring mode (with or without deletion of the distance to go) for each measuring procedure.

The result of a measurement can be read in the part program or with synchronous actions in both the machine and the workpiece coordination system. You can test the deflection of the touch probes by scanning a variable and outputting it over the PLC interface and deriving responses in the part program. The "measuring stage 2" option provides you with expanded functionality (for example for axial measuring, evaluating up to 4 trigger events, and cyclic measuring).

Measuring stage 2 (option)

While the measuring function in motion blocks in the parts program is limited to one block, you can activate measuring functions from synchronous actions at any time independent of the parts program. The measuring events can be assigned to the axes in the CNC block. In the case of simultaneous measuring, up to 4 trigger events can be evaluated per position control cycle. Measured values are read as a function of the three parameters: touch probe, axis, and measuring edge.

In the case of continuous (cyclic) measuring, the measurement results are written to a FIFO variable. Endless measuring can be achieved by reading out the FIFO values cyclically.

Measurement results can be optionally logged in a file on the controller or output to a printer or PC via the V.24 (RS 232) interface. The standard protocol contained in the measuring cycles can be modified by the user.

Limited functionality on the SINUMERIK 810DE/840DiE/840DE: Measuring from synchronous actions and cyclic measuring are not possible.

Measuring systems

On the SINUMERIK 840D/840Di, the measuring systems are evaluated by the SIMODRIVE 611 digital drive modules with high resolution.

The SINUMERIK 810D evaluates the measuring systems directly on the CCU module.

With the SINUMERIK 810D, additional measuring systems can be connected via axis expansion with SIMODRIVE 611 digital modules.

Measuring system 1 and 2, selectable

For special applications, two encoders can be assigned to one axis, e.g. a direct measuring system for the machining process with high demands on accuracy, and an indirect measuring sys-

tem for high-speed positioning. The switchover between one measuring system and the other is performed in the PLC.

Measuring system error compensation

→ Leadscrew error/measuring system error compensation

Mode group (BAG)

A mode group combines CNC channels with axes and spindles to form a machining unit.

A mode group contains the channels which must always be in the same mode at the same time during the machining sequence. Within a mode group, every axis can be programmed in every channel.

A mode group can be regarded as an independent, multi-channel CNC.

Additional mode groups are optional (see Section 2).

Glossary

Functions and Terms

SINUMERIK 810D/840Di/840D

Modes

In the "machine" control area, you have a choice of three modes:

- JOG

JOG mode (jogging) is intended for the manual movement of axes and spindles as well as for setting up the machine. The setting-up functions are approach to reference point, repositioning, handwheel control or incremental mode, and redefinition of control zero (preset/set actual value)

- MDA

In MDA (Manual Data Automatic/Manual Input) mode, you can enter program blocks or sequences of blocks, then execute them immediately via NC Start. The tested blocks can then be saved in part programs.

The Teach in submode allows you to transfer movements to the MDA program by returning and storing positions.

- AUTO

In AUTO mode, your part programs are executed automatically once they have been selected from the workpiece, part program and subroutine directories (normal operation).

During Automatic mode it is possible to generate and correct another part program.

While AUTOMATIC mode is in progress, the user can write or edit another part program.

In MDA and AUTO mode, you can modify the sequence of a program with the following functions:

- SKP Skip block (up to 8 skip levels)
- DRY Dry run feed rate
- ROV Rapid traverse override
- SBL1 Single block with stop after machine function blocks
- SBL2 Single block with stop after every block
- SBL3 STOP in cycle
- M01 Programmed stop
- DRF Differential resolver function
- PRT Program test

Monitoring functions

The controls contain watchdog monitors which are always active. These monitors are there to detect problems in the CNC, PLC or machine in time to prevent damage to workpiece, tool or machine. When a problem occurs, machining is interrupted and the drives brought to a standstill. The cause of the problem is stored and displayed as alarm. At the same time, the PLC is informed that a CNC alarm is pending.

There are monitors for the following:

- Read-in
- Format
- Position encoder and drive
- Contour
- Position

- Zero speed
- Clamping
- Setpoint speed
- Actual speed
- Enable signal
- Voltage
- Temperatures
- Microprocessors
- Serial interfaces
- Transfer between CNC and PLC
- Backup battery voltage
- System memory and user memory

Motion control with PROFIBUS-DP

Compatible extension of the PROFIBUS-DP standard for the synchronization of bus nodes, making it possible to implement reliable control algorithms, such as the closing of a position control loop, via the bus.

Clock-pulse synchronization

The mechanisms for synchronization of the internal time levels in master and slave global control (broadcast message), PLL (phase lock loop), as well as the constant bus cycle time

(isochronous mode), give the application/control cycles in the master and in the participating slaves a fixed time relationship to one another.

Data Exchange Broadcast (internode communication)

Efficient data interchange between slaves without delays imposed by the master. Data sent by one slave can be monitored by the slaves that have been requested to do so, allowing them to respond (e.g. actual position values).

Motion-synchronous actions

→ Synchronous actions

Multi-axis interpolation (option)

On the SINUMERIK 810D/840Di/840D, the number of interpolating axes is expandable. The number of interpolating axes is limited by options and machine data as well as by the number of axes in the channel.

Multi-axis interpolation is not possible for SINUMERIK 810DE/840DiE/840DE.

Multiple feedrates in one block

Depending on external digital and/or analog inputs, you can use this function for motion-synchronous activation of up to 6 different feedrates, a dwell time, and a retraction in a single CNC block. The input signals are combined in an input byte with permanently assigned function. The retraction is initiated by an amount defined in advance within an IPO cycle. Retraction movement or dwell time (e.g. sparking out during grinding) lead to deletion of the residual distance.

Typical applications involve analog or digital calipers or when changing from infeed feedrate to machining feed-rate via proximity switches. During internal cylindrical grinding of a ball bearing ring, for instance, in which calipers are used to measure the actual diameter, the feedrate required for roughing, finishing or smooth-finishing can be activated in dependence on threshold values.

Multipoint interface (MPI)

The operator panel and the machine control panel communicate with the CNC via a multipoint interface. Via this interface, several devices can be connected and communicate with the CNC as would be the case in a bus system.

On the SINUMERIK 810D, the MPI is located on the front panel of the CCU module. The data signalling rate is 187.5 Kbit/s.

On the SINUMERIK 840Di, the MPI is located on the MCI board. The data signalling rate is 1.5 Mbit/s.

In addition to the PG-MPI (187.5 Kbit/s), the SINUMERIK 840D's NCU modules are also equipped with a high-speed operator panel interface (BTSS) that has a data signalling rate of 1.5 Mbit/s and is used to connect operator panel, machine control panel, hand-held programming unit, and pushbutton panel.

NCU-independent setpoint linkage (option)

→ Link axis

This functionality permits coupled axes beyond NCU limits as an extension to the »Link axis«: the master axes and the following axis can execute on different NCUs. This option can be used as a setpoint linkage for the following coupled axes: master value

coupling (plus simulated master value), coupled motion, synchronous spindle, electronic gear unit and tangential control.

Applications include e.g. multi-spindle turning machines, or transfer controls of presses.

Offline ISO dialect/CNC program converter

This program converter allows you to convert both external and Fanuc0 programs as well as workpiece programs for the

SINUMERIK 800 controls into the format for the SINUMERIK 810D/840Di/840D.

Online ISO dialect interpreter (option)

With the online ISO dialect interpreter, part programs in other ISO dialects (for example G codes from other manufacturers)

can be read into the SINUMERIK 810D/840Di/840D, edited, and processed.

Operator interface HMI-Advanced/MMC 103 system software on PC/PG

When an MPI card is installed in a PC or PG, the complete user interface is available on that PG or PC. This allows user-friendly start-up and servicing of the control when the system is operated

with an OP or without a console. It also makes setting up the machine and editing and executing workpiece programs easy and problem-free.

Glossary Functions and Terms

SINUMERIK 810D/840D/840D

Oscillation functions (option)

With this function, an axis oscillates at the programmed feedrate between two reversal points. Possible applications e.g. for grinding machines.

Asynchronous oscillation across block boundaries

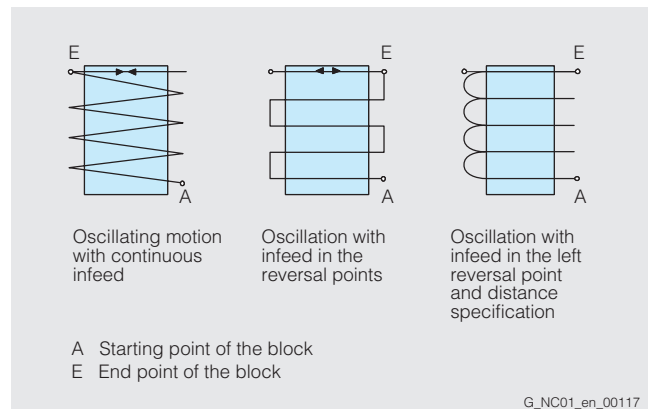
Several reciprocating axes may be active. During reciprocating movement, other axes can interpolate at will. The reciprocating axis can be the input axis for the dynamic transformation or the leading axis for gantry or coupled axes.

Block-related oscillation

- Oscillation with infeed in both or only in the left or right reversal point. Infeed is possible along a programmable path prior to the reversal point is possible.
- Sparking out strokes after oscillation are possible.

Behavior of the reciprocating axis in the reversal point:

- A change of direction is initiated
 - without reaching the exact stop tolerance range (soft reversal)
 - after reaching the programmed position or
 - after reaching the programmed position and expiration of the dwell time.



Oscillation functions

- The following manipulations are possible:
 - Oscillating motion and infeed can be terminated by deleting the distance-to-go.
 - Modification of the reversal points via CNC program, hand-wheel or direction keys.
 - Manipulation of the reciprocating axis's feedrate via CNC program, PLC or Override.
 - Control of the oscillating motion via the PLC.

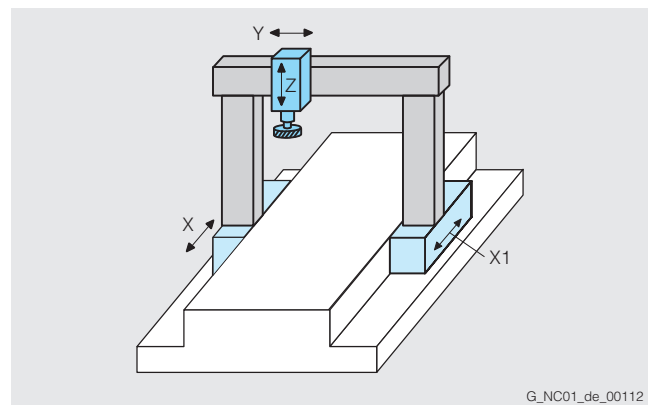
The spindles can also perform oscillation motion.

Pair of synchronized axes (gantry axes) (option)

With the "gantry axes" function, the axes of up to 3 pairs of axes can be traversed simultaneously without mechanical offset. The actual values are continuously compared and even the smallest deviations corrected.

During both operation and programming, the axes in a grouping are treated like machine axes. A gantry grouping consists of a leading axis and as many as two synchronized axes. Two leading axes can be coupled using curve table interpolation.

Up to three gantry groupings can be defined per control system (only one gantry grouping with CCU1/CCU2/NCU 571.3).



Gantry axes (pair of synchronized axes X/X1)

Part program management

Part programs can be organized according to workpieces. This permits clear allocation of programs and data to workpieces. The size of the user memory determines the number of programs

and the amount of data that can be managed. Each file (programs and data) can be assigned a name comprising up to 23 alphanumeric characters.

Path switch signals/cam controller (option)

→ Fast CNC inputs/outputs

Position-dependent interface signals for the PLC can be set using position signals. The positional values at which the signal output and a derivative action/delay time are to be set can be programmed in the part program and entered in the setting data. The function can be controlled via the PLC.

The function is used for applications such as activating protection areas or position-dependent triggering of movements (e.g. hydraulic reciprocating axes during grinding).

32 position signal pairs are available (16 signals in the case of SINUMERIK 810DE). The position signals are output in the IPO cycle. They can also be output with the function "high-speed digital CNC inputs/outputs" as switching outputs in the position control cycle.

Path velocity-dependent analog output (option)

"Path velocity-dependent analog output" makes it possible to output the current path velocity in the interpolation cycle. This value can, for example, be made available via a module "DMP Compact 1 A analog" (see Section 5) on the NCU terminal block

(on the SINUMERIK 840Di, via PROFIBUS-DP and S7-300 modules). The function is programmed via synchronous actions. One application is laser power control.

PCU card as additional program memory

With PCU 20, the existing PC card plug-in can be used with a PC card as an additional external program memory. Programs can be copied in both directions between the PC card and the parts program memory of the control. The programs can also be

written onto the PC card from the external PC via Ethernet or from the connected diskette drive. The function "External execution" enables direct processing of programs present on the PC card.

Plain text display of user variables

→ High-level CNC language

In addition to the predefined variables, the programmer can define and initialize his own variables. The variables are displayed in plain text format (e.g. definition: DEF INT NUMBER/

Display: NUMBER or definition: DEF REAL DEPTH-Display: DEPTH).

PLC area

SINUMERIK 810D

On the SINUMERIK 810D, a SIMATIC S7-300-compatible PLC 314 is integrated in CCU 1 or a compatible PLC 315-2DP is integrated in CCU 2/CCU 3. PROFIBUS I/O components can be operated on the PLC 315-2DP. As I/O modules, you can use either SIMATIC S7-300 components or single I/O modules.

SINUMERIK 840Di

On the SINUMERIK 840Di, a SIMATIC S7-300-compatible PLC 315-2DP is integrated on the MCI board. The SIMATIC DP ET 200 with 12 Mbaud capability can be connected to the PROFIBUS-DP as I/O.

SINUMERIK 840D

On the SINUMERIK 840D, a SIMATIC S7-300-compatible PLC 315-2DP is integrated in den NCUs 561.2/571.3/572.3 and NCU 573.3. The NCUs 572.4 and 573.4 contain a PLC 314C-2 DP compatible with the SIMATIC S7-300C, providing an increase in performance for the PLC application program of approx. 3 compared to the NCUs with PLC 315-2 DP, as well as a PLC user memory up to 480 Kbyte. The same components as on the SINUMERIK 810D can be used as I/O modules.

PLC programming (STEP 7)

The PLC in the SINUMERIK 810D is programmed using the user-friendly STEP 7 software.

The STEP 7 programming software is based on the Windows operating system, and combines the familiar STEP 5 programming functions with additional innovative functional developments. The user can program in STL (statement list), FBD (function block diagram), and LAD (ladder diagram) format, and can switch from one to the other using STEP 7 pull-down menus.

The following blocks are available for structuring programs:

- Organization blocks (OBs)
- Function blocks (FBs) and function calls (FCs)
- Data blocks (DBs)

In addition, system function blocks (SFBs) and system functions (SFCs) integrated in the operating system can also be called.

The STEP 7 software package (for SIMATIC S7-300) is a standard component of the SIMATIC programming units (e.g. Field PG). A software package for standard industrial PCs is also available.

The PLC can also be programmed in other SIMATIC S7 high-level languages, such as S7-HiGraph and S7-Graph.

PLC → NCK interface

A number of functions can be executed via the NCK and PLC interface, ensuring excellent machining flexibility. Some of these are:

- Controlling positioning axes
- Executing synchronous actions (auxiliary functions)
- Reading and writing of NCK system variables by the PLC
- Reading and writing of CNC user variables by the PLC

The PLC basic program, which is part of the toolbox, organizes the interchange of signals and data between the PLC user program and the NCK, PCU and machine control panel areas. In the case of signals and data, a distinction is made between the following groups:

- Cyclic signal exchange:

Commands from the PLC to the NCK (such as start, stop, etc.) and NCK status information (e.g. program executing). The basic program carries out cyclic signal exchange at the beginning of the PLC cycle (OB 1). This ensures, for example, that the signals from the NCK remain constant throughout a PLC cycle.

Glossary

Functions and Terms

SINUMERIK 810D/840D/840D

PLC area (continued)

- Event-driven signal exchange NCK → PLC:
PLC functions executed in dependence on the workpiece program are initiated via auxiliary functions in the workpiece program. If a block with auxiliary functions is executed, the type of auxiliary function determines whether the NCK has to wait for this function to execute (e.g. tool change) or whether the function will be executed together with the machining process (e.g. tool loading on milling machines with chain magazine). In order for CNC machining to be affected as little as possible, data transfer must be as fast as possible and yet reliable. It is therefore interrupt- and acknowledgment-controlled. The basic program evaluates the signals and data, sends an acknowledgment to the NCK, and transfers some of the data to OB40 and the rest, at the beginning of the cycle, to the user interface. If the data require no acknowledgment from the user, CNC machining is not affected.

- Event-driven signal exchange PLC → NCK:
Whenever the PLC sends a request to the NCK (such as a request to traverse an auxiliary axis), a "PLC → NCK event-driven signal exchange" takes place. Here again, the data transfer is acknowledgment-controlled. Such a signal exchange is initiated by the user program via an FB or FC. The associated FBs (function blocks) and FCs (function calls) are provided together with the basic program.
- Messages:
The acquisition and editing of user messages is handled by the basic program. The message signals are forwarded to the basic program via a specified bit array. Here, the signals are evaluated, then transferred to the PLC diagnostic buffer when one of the signalled events occurs. If an OP is available (e.g. on the PCU 50), the messages are transferred to it and displayed there.

PLC programming with HiGraph

The HiGraph method is used for describing technical systems and converting these descriptions into PLC programs.

With HiGraph, a machine or plant is seen as a combination of separate functional units. These functional units can be made up of basic mechanical and electrical elements. The HiGraph method is used in the automation of machines and plants where mechanical movement and sequences take priority, e.g. machine tools, transfer lines, and conveyor and transportation systems.

The HiGraph method can be used:

- during the machine and plant planning phase
- during the function planning phase
- during the design phase, e.g. of the mechanics
- for writing programs

- during the testing and start-up phase
- to operate the automated machine
- for maintenance and diagnostic tasks.

Advantages of the HiGraph method:

- Quicker from the design to the result
- Shorter testing phases
- Structuring using symbolic names
- Application-oriented
- Object-oriented thinking
- Graphic programming
- Easy to use
- Reliable software
- Faster and simpler diagnostics
- Local service at the machine

PLC status

In its "diagnostics" area, the operator panel allows you to check and modify PLC status signals.

This makes it possible for you to take care of the following right on site:

- Check the input and output signals of the PLC I/O
- Do limited troubleshooting
- Check the NCK/PLC and PCU (MMC)/PLC interface signals for diagnostic purposes

The status of the following data items can be displayed on the operator panel:

- Interface signals from/to the machine control panel
- NCK/PLC and PCU/MMC/PLC interface signals
- Data blocks, memory bits, timers, counters, inputs and outputs

For test purposes, you can also change the status of the above-listed signals. Signal combinations are also possible, and as many as 10 operands can be modified simultaneously.

PLC user memory

In the PLC user memory of the PLC CPU, the PLC user program and the user data are stored together with the PLC basic program.

The memory of the PLC CPU is divided up into load memory, working memory, and system memory.

Load memory is retentive, and takes the form of either integrated RAM or RAM module (plug-in memory card). It contains data, program and decompiling information.

The load memory and the high-speed work memory for execution-relevant program tests provide sufficient space for user programs.

Polar coordinates

Programming in polar coordinates, it is possible to define positions with reference to a defined center point by specifying the radius and angle.

The center point can be defined by a reference dimension or incremental dimension.

Polynomial interpolation (option)

With this function, curves can be interpolated for which the CNC axes follow the function:

$$f(p) = a_0 + a_1p + a_2p^2 + a_3p^3$$

(polynomial, up to 3rd degree)

or from software release 6 onwards:

$$f(p) = a_0 + a_1p + a_2p^2 + a_3p^3 + a_4p^4 + a_5p^5$$

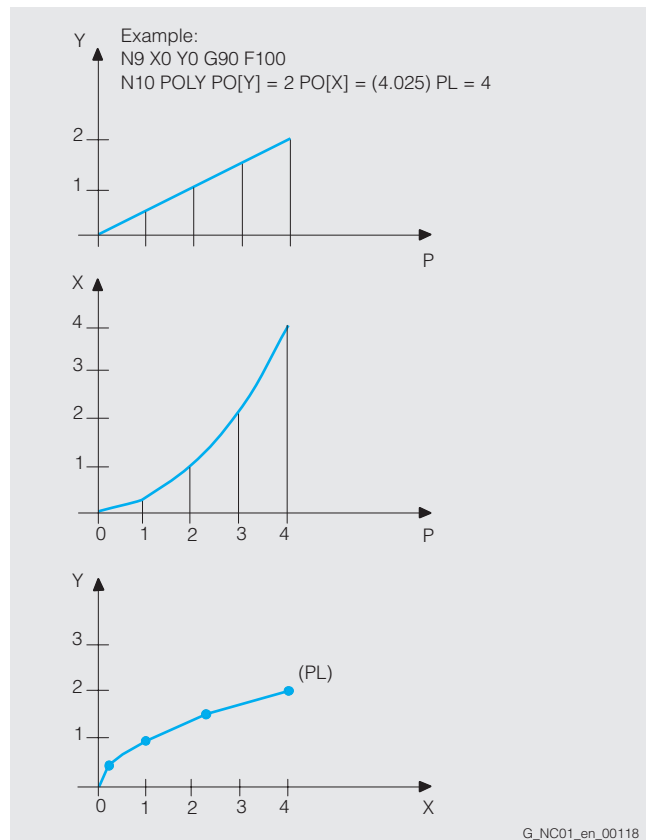
(polynomial, max. 5th degree)

The coefficient a_0 is the end point of the previous block, a_1 is calculated as the end point of the current block, a_2 , a_3 , a_4 and a_5 must be calculated externally and then programmed.

With polynomial interpolation, it is possible to generate many different types of curve, such as straight line-, parabolic-, and exponential functions.

Tool radius compensation can be used as in linear and circular interpolation.

5th degree polynomials, in contrast to 3rd degree polynomials, permit further approximation of defined contours. However, polynomial interpolation primarily serves as an interface for programming externally generated spline curves. 5th degree polynomials can optionally be used if the coefficients are obtained directly from a CAD/CAM system («closer to the surface»). A prerequisite for efficient utilization of this polynomial interpolation is therefore a corresponding CAD/CAM system.



Polynomial interpolation

Position monitor (positioning monitor)

SINUMERIK control systems provide extensive monitoring mechanisms for axis monitoring:

- Motion monitors:
Contour monitor, positioning monitor, zero-speed monitor, clamping monitor, set speed monitor, actual speed monitor, encoder monitor
- Static limits monitors:
Position switch monitor, working area limitation

The positioning monitor is always activated following "setpoint-based" termination of traversing blocks. In order to ensure that an axis is in position within a specified period of time, the timer

programmed in the machine data is started when a traversing block terminates; when the timer expires, a check is made to ascertain whether the following error exceeded the limit value (machine data). When the specified "fine exact stop limit" has been reached or following output of a new position setpoint other than zero (e.g. after positioning to "coarse exact stop" and subsequent block change), the positioning monitor is deactivated and replaced by the zero-speed monitor.

The positioning monitor is effective for linear and rotary axes as well as for position-controlled spindles. In follow-up mode, the positioning monitor is not active.

Positioning axes/auxiliary axes

Positioning axes can execute movements simultaneously with machining, thus considerably reducing non-productive times. They can be used to advantage to control workpiece and tool loaders or tool magazines. They can be programmed with an axis-specific feed in the part program. The axis movement is also possible beyond block boundaries.

Positioning axes can also be controlled by the PLC. This means that axis movements can be started independently of the part program without using up an additional channel.

Auxiliary spindles are speed-controlled spindle drives without actual position sensor, e.g. for machine tool drives.

Glossary

Functions and Terms

SINUMERIK 810D/840Di/840D

Positioning axes and spindles via synchronous action

You can position axes or spindles in dependence on conditions (the actual values of other axes, high-speed inputs, etc.) with a special feedrate or speed to a specific setpoint via synchronous actions. Synchronous actions are executed in the interpolation cycle, are carried out in parallel with the actual machining procedure, and are not limited to CNC block boundaries.

These so-called command axes and command spindles can be started in the IPO cycle from within the main program. The path to be traversed is either predefined or is calculated from real-time variables (with expanded arithmetic functions) in the IPO cycle. Spindles can be started, stopped or positioned asynchronously in dependence on input signals without PLC intervention.

Preset

With the preset function, you can redefine the control zero in the machine coordinate system. The preset values affect machine axes. "Preset" does not cause the axes to move, but a new position value is entered for the current axis positions. After new

actual values are set, protective areas and software limit switches are not reactivated until after a new reference point approach has been completed.

PROFIBUS-DP

PROFIBUS-DP is the protocol for the distributed I/O, and is based on the international open fieldbus standard as laid down in European fieldbus standard EN 50170 Part 2.

PROFIBUS-DP is optimized for high-speed, time-critical data transfer at the field level.

This fieldbus is used for cyclic and non-cyclic data interchange between a master and its assigned slaves.

Master, active bus nodes

Devices which control the data traffic on the bus are referred to as masters. They send requests in the form of control words and setpoint values.

Masters are divided into two classes:

- Class 1 DP masters:
These are master devices which interchange information with their slaves in fixed message cycles (such as SIMATIC controllers and SINUMERIK controls).
- Class 2 DP masters:
These are devices for configuring, commissioning, control and

monitoring during operation. Use DP/V1 (auxiliary services) for parameter initialization and diagnostics (PC's programming, control and monitoring devices).

Slaves, passive bus nodes

These are devices which receive messages, acknowledge messages, and forward messages to the master at the master's request (SIMODRIVE 611 universal, POSMO SI/CA/CD, SIMATIC I/O). They send responses in the form of status words and actual values.

Transfer

PROFIBUS supports data transfer in accordance with RS 485 and Optical Link.

Baud rates

9.6 kbaud, 19.2 kbaud,
45.45 kbaud, 93.75 kbaud, 187.5 kbaud, 500 kbaud,
1.5 Mbaud, 3.0 Mbaud, 6.0 Mbaud, 12 Mbaud.

A maximum of 1.5 Mbaud for optical link plugs (OLPs).

PROFIBUS tool and process monitoring (option)

Using the »PROFIBUS tool and process monitoring« function, the digital drive data for torque, active power and actual current

are made directly available for evaluation via the PROFIBUS-DP interface. One or two PROFIBUS slaves can be connected.

Programmable acceleration

With the function "programmable acceleration" it is possible to modify the axis acceleration in the program in order to limit mechanical vibration in critical program sections.

The path or positioning axis is then accelerated at the programmed value. The maximum acceleration value stored in the control can be exceeded by up to 100%.

This limitation is active in AUTOMATIC mode and in all interpolation modes. As part of intelligent motion control, this function provides a more precise workpiece surface.

Programming language

The CNC programming language is based on DIN 66025. The new functions of the CNC high-level language also contain macro definitions (combined sequences of commands).

Program preprocessing (option)

The execution time of a CNC program can be reduced considerably by the preprocessing of cycles. The programs in the directories for standard and user cycles are preprocessed at "power on" when the machine data is set. Especially in the case

of programs containing portions written in a high-level language and of compute-bound programs (e.g. programs containing check structures, motion-synchronous actions or cutting cycles), execution times can be reduced by up to $\frac{1}{3}$.

Protection zones 2D/3D

Protection zones allow you to protect various elements on the machine and its equipment, as well as the workpiece, from incorrect movements.

Some of the elements which can be protected are, for example:

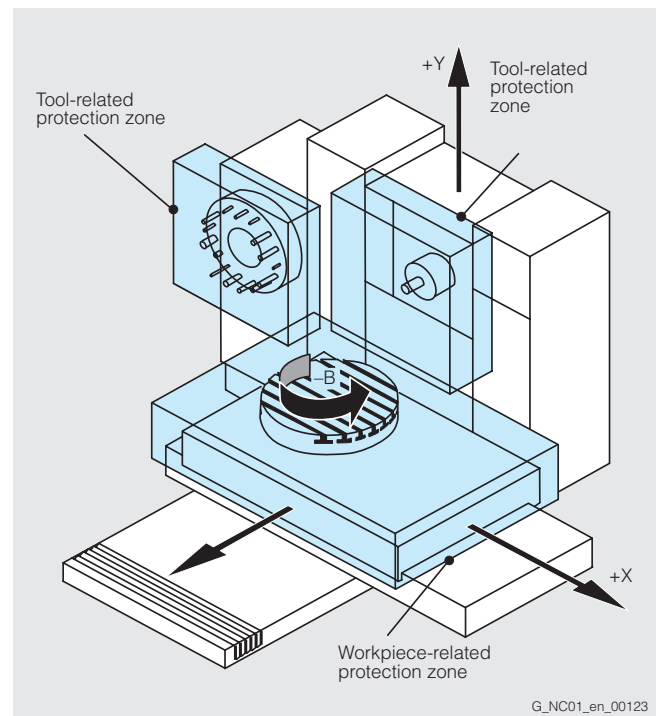
- Fixed machine components and built-on accessories (tool magazines, swivelling probes)
- Moveable parts belonging to the tool (tool carriers)
- Moveable parts belonging to the workpiece (clamping tables, clamps, spindle chucks, tailstocks)

For the elements to be protected, 2- or 3-dimensional protection zones are defined in the part program or via system variables.

These protection areas can be activated and deactivated in the part program. Protection zones must always be divided into workpiece-related and tool-related zones. During machining in JOG, MDA or AUTOMATIC mode, a check is always made to see whether the tool (or its protection zones) violate the protection zones for the workpiece.

Monitoring of the protection zones is channel-based, that is, all active protection zones for a channel are mutually monitored for collisions (protection zones not channel-specific with NCU system software for 2/6 axes).

A maximum of 10 protection zones and 10 contour elements which describe a protection zone are available (with CCU 1, CCU 2, NCU 561.2 and NCU 571.3: max. 4 protection zones and 4 contour elements).



Protection zones

Punching/nibbling (option)

The punching/nibbling functions are implemented essentially via the language commands, stroke control and automatic path division.

- Language commands
The punching/nibbling commands are activated and deactivated using simple, clear high-level language elements such as PON, SON, PONS, PDELAYON, and so on.
- Stroke control
CNC and punch are synchronized to each other by the high-speed signals that are input and output via the drive bus in the control's position control cycle, making it possible to attain high speeds and maximum precision.

- Automatic path division
You can choose whether you want the control to break the machining path down automatically by stroke length (SPP) or stroke rate (SPN). With SPP, the path is broken down into programmable segments of identical size (modal effect). SPN breaks the path down into a programmable number of strokes (affects individual blocks).

Glossary

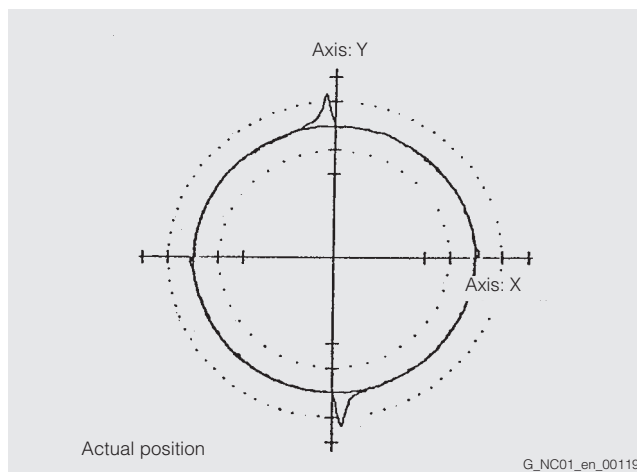
Functions and Terms

SINUMERIK 810D/840D/840D

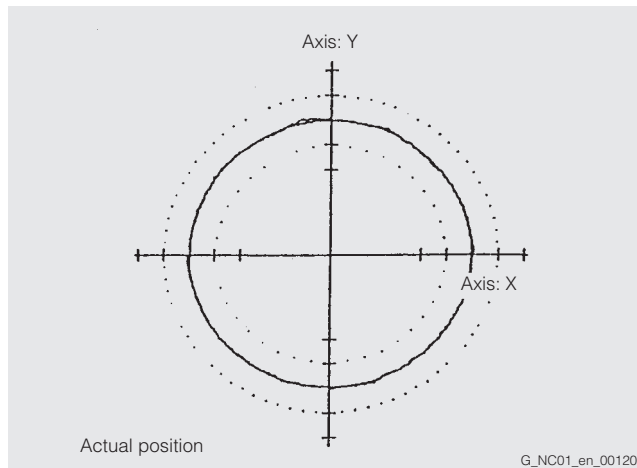
Quadrant error compensation

Quadrant error compensation (also referred to as friction compensation) ensures a much higher degree of contour precision, particularly when machining circular contours. At the quadrant transitions, one axis traverses at the maximum tool path velocity while the second axis is stationary. The different friction conditions can cause contour errors. Quadrant error compensation virtually eliminates this problem, and produces excellent results, without contour errors, in the very first machining operation.

In operator-controlled quadrant error compensation, you yourself set the intensity of the correction pulse as per an acceleration-based characteristic. This characteristic is determined and initialized on start-up with the aid of the circularity test. During the circularity test, deviations of the actual position from the programmed radius (particularly at the quadrant transitions) are metrologically recorded and graphically represented while the circular contour is being described.



Quadrant transitions without compensation



Quadrant transitions with quadrant error compensation

Quadrant error compensation, automatic (option)

To simplify start-up, the compensation characteristic for »Quadrant error compensation« with a neural network need no longer be set manually by the commissioning engineer. It is automatically determined during a learning phase, and saved in the buffered user memory.

The neural network can simulate the compensation characteristic far better, achieving an improved accuracy, and permits simple and automatic subsequent optimization on site at any time.

Reference point approach

When using a machine axis in program-controlled mode, it is important to ensure that the actual values supplied by the measuring system agree with the machine coordinate values.

Reference point approach (limit switch) is performed separately for each axis in a sequence that can be defined in the machine data at a defined velocity using the direction keys or automatically via program command G74. If linear measuring systems with distance-coded reference marks are used, reference point approach is shorter, as it is necessary to approach only the nearest reference point.

Referencing of an axis with absolute-value encoders is carried out automatically when the control is switched on (without movement of axis) if the corresponding axis is recognized as being calibrated.

Repos

Following a program interruption in AUTOMATIC mode (e.g. to take a measurement on the workpiece and correct the tool wear values or because of tool breakage), manual repositioning of the tool is possible after changing to JOG mode. In this case, the control stores the breakpoint coordinates and displays the differential travel of the axes in the actual-value window as Repos (repositioning) offset.

The contour can be reapproached:

- in JOG mode using the axis- and direction keys. It is not possible to overshoot the breakpoint; the feedrate override switch is effective.
- per program (with reference to the interruption block) either at the point of interruption, the start of the block, at a point between the start of the block and the interruption point, or at the end of the block. Modified tool offsets are taken into account. You can program approach movements as straight lines, in quadrants or in semicircles.

Representation (2D) of 3D protection areas/work areas

With the aid of protection areas, various elements on the machine, the equipment, and the workpiece can be protected against incorrect movements. The programmed 3D protection areas are displayed in 2D.

→ Working area limitation; Protection areas

This also applies to the programmed working area limitation.

Rotary axis, endlessly-turning

Depending on the application, the traversing range of a rotary axis can be limited via software switch (e.g. operating range between 0° and 60°) or to a corresponding number of rotations (e.g. 1000°), or unlimited (endlessly turning in both directions).

This function can also be used with absolute-value encoders.

Safety functions

Safety functions for humans and machines are provided by SINUMERIK Safety Integrated (see Section 5, "Basic components" for the SINUMERIK 840D and the SINUMERIK 840C).

Sag compensation, multidimensional (option)

Multidimensional compensation is also possible for the effects of physical influences such as sag or leadscrew errors. The compensation tables can be switched from the PLC.

When the reference axis and the compensating axis are identical, leadscrew errors can be compensated.

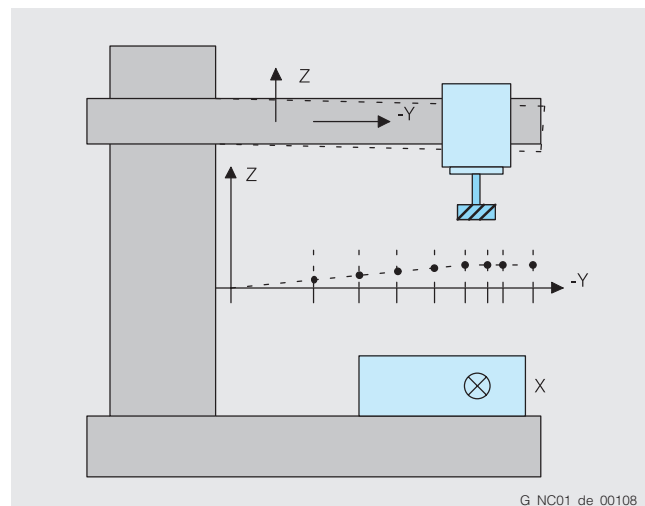
By transferring weighting factors (PLC interface), stored compensating characteristics can be adapted to different conditions (e.g. tools).

The most important features of interpolation and compensation using tables are as follows:

- Independent error characteristics can be defined, in number twice the maximum number of axes
- Freely selectable compensating positions, the number of which is configurable (dependent on the configuration of CNC user memory)
- Interpolating inclusion of the compensation values
- Weighting factor for compensation of tool weights
- Reference axis and compensating axis are selectable.

The function is available on the SINUMERIK 810D/840D and, with limited functionality, on the SINUMERIK 810DE/840DiE/840DE:

The correctable tolerance band is limited to 1 mm (0.039 in) (10 mm (0.394 in) on the SINUMERIK 810D/840Di/840D).



Example: Sag compensation

Glossary

Functions and Terms

SINUMERIK 810D/840D/840D

Scratching, determining zero offset

A zero offset can also be determined through "scratching", taking into consideration an (active) tool and, where applicable, the base offset, by moving the axis to the workpiece, entering the

desired setpoint position (e.g. "0"), and actuating the Input key. The control computes the offset.

Screen dimmer switch

When screen darkening is activated, both the screen and the backlighting for the operator panel are darkened, for example

under PLC control or after a programmable amount of time has elapsed. This increases the service life of the screens.

Separate path feed for corners and chamfers

To optimize solutions for machining tasks, a separate path feed can be programmed with FRCM (modal) or FRC (by block) for the contour elements "corner" and "chamfer". Feed reduction

thus makes it possible to achieve the desired geometrically precise definition of corners and chamfers.

Serial interface (RS 232C)

For data input/output, the OP 030/MMC 100.2 is equipped with one serial interface (RS 232C) and the PCU 50/MMC 103 two serial interfaces. These interfaces can be used to load and

archive programs and data. They can be operated and initialized under operator guidance on the operator panel.

Series start-up

Files called series machine start-up files can be generated to enable transfer of a particular configuration, its entirety, to other controls that use the identical software version, for example controls that are to be used for the same machines. Series machine start-up thus means bringing a series of controls to the same basic state as regards their data. You can archive/read CNC, PLC and PCU data for series start-up. Compensation data can

be optionally saved. The drive data are stored in binary, and cannot be modified. Series start-ups can even be performed readily and easily without a programming unit. Simply create a series start-up file in the MMC, save it on a PC card in the control, insert this card in the next control, and begin the series start-up procedure.

Set actual value

The "set actual value" function is provided as alternative to the "preset" function. To use this function, the control must be in the workpiece coordination system (WCS). With "set actual value", the workpiece coordination system is set to a defined actual

coordinate and the resulting offset between the previous and a newly entered actual value computed in the WCS as 1st basic offset. The reference points remain unchanged.

Setpoint exchange

The "setpoint exchange" function is used on milling machines with special milling heads on which, for example, the spindle motor is used both for driving the tool and for orientation of the milling head. In this case, the spindle and the milling head axes are defined as independent axes, but are traversed only in succession by one motor.

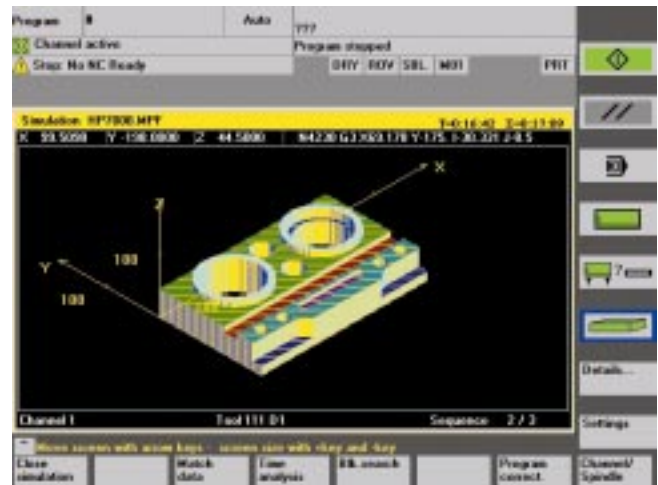
It is possible to switch up to four axes to one motor. The axes, between which a setpoint exchange takes place, can be assigned to different channels or mode groups.

Simulation

Machining simulations, with emphasis on drilling/milling and turning technologies, can be executed on the control's HMI in the workpiece coordinate system for certain machine kinematics depending on the active operating software (see also Section 2, summary of functions) and its versions:

HMI-Advanced/MMC 103 system software

- **Drilling/milling:**
For these technologies, simulation of multiface drilling and milling is possible with representation of the removed material and/or selectable linear tool path graphics. Simulation of removal is primarily designed for paraxial machining in a rectangular 3D workpiece space. Other kinematics which cannot be exactly represented using the 3D removal simulation, or machining operations based on incomplete tool data, can nonetheless provide informative approximations using the integral tool path simulation.
- **Complete turning machining:**
Turning operations can be displayed here in side views as linear tool path graphics with dynamic updating of the blank envelope in the dynamically balanced 2½D workpiece. Drilling and milling on the front face or on the peripheral surface of turned parts can be simulated with representation of the removed material and/or the tool path graphics with the same features as described under »Drilling/milling« starting from software release 5.1. Furthermore, display versions are available for variable machine arrangements (e.g. for turning in front of or behind the turning center, on the main spindle or counterspindle, for horizontal or vertical turning machine orientations).
- **General features:**
Simulation is supported by an autonomous program interpreter and a separate simulation data environment at the HMI/MMC level. The simulation interpreter extensively considers the complete language syntax of the SINUMERIK 810D/840Di/840D control range, including the possibility for incorporating special user options on the machine by comparing data with the NCK environment.
The simulation data can be matched as required statically with the NCK environment (initialization data, macros, user data, tool data, machining cycles) or also dynamically when tool data or machining cycles are changed. The current tool path interpolation points of the simulation interpreter, together with transfer of the dynamic tool data (if selected) are conditioned in a Cartesian 3D tool space for further processing in the graphics module, and additionally provided when turning with orientation angles of relevant rotary axes.
To permit display of tool graphics in the simulation, default tool graphics are generated directly from the available tool correction parameters (TOA data) depending on the main technological group (drilling/milling or turning) and certain subordinate types (e.g. end milling cutter, plunge-cutter etc.).



Simulation of drilling/milling with HMI-Advanced



Simulation of turning with HMI-Advanced

If tool information is missing or unsuitable, only a polymarker (cross) is displayed on the corresponding section of the path. In addition, the display as well as execution of the simulation are influenced by visualization attributes (feed, rapid traverse, selected type of tool etc.) and by status attributes (start/stop, single block mode, tool mode, output of labels as ASCII path marker, saving of intermediate model etc.) which can be defined simply using screen forms. Permanently visible status displays (actual position, current block, selected channel, active tool etc.) support simulation at all times. The selectable time determination for coarse estimation of workpiece machining times can be processed in tabular form for freely-definable machining sections (program labels, end of program M30, ...).

Simulation (continued)

Also taken into account are the programmed feeds and estimated idle times with 100% feed override and without acceleration factors. To achieve interactive program correction, the ASCII editor of the control can be selected with direct reference to the current position in the program from the current simulation interrupt status (program halt or alarm status). Following this program adaptation, new simulation preprocessing is automatically carried out at the interrupt position. Particularly with respect to multi-channel machining, it is possible from software release 5.1 onwards to carry out free selection of the partial machining operations (individual programs) which belong to the complete machining (workpiece) and which are to be simulated sequentially. This mode is supported by preprocessing mechanisms for optional continuation of machining at defined program sections including the associated intermediate graphics models of previously simulated programs. In this manner, the simulation results of several parts programs in succession can be superimposed in a complete representation on the same unmachined part (e.g. starting from preformed blanks, for multiface machining when milling, multi-saddle and multi-spindle machining when turning etc.).

The machined part finally originates from the sequential interaction of all individually simulated parts programs. When turning, it is possible to define a longitudinal offset for machining of the reverse side with mirrored tools or NC vocabulary words for dynamic spindle switchover, e.g. for main spindle and counter-spindle operation, amongst others.

Visualization of the simulation is largely oriented according to the VGA standard, and can be user-defined in many applications using screenforms for settings. The optional color assignment from the VGA color palette permits extremely clear identification of bank, zero point, toolholder, tool cutting edges and tool paths etc. Graphics can be observed in various views and sections, in zoom representations, or in several windows simultaneously.

HMI-Embedded/MMC 100.2 system software

- Drilling/milling (option):
The 2½D simulation in 3 views represents the tool path of the programmed workpiece with exact dimensions. The defined color shades represent different processing depths, and offer additional orientation support. This simulation serves to check the result of the programmed part on three-axis milling machines. It can be triggered with a dry run feedrate or in real-time.
In addition to this, it can also be started parallel to machining. This may be necessary if viewing of the workpiece is hindered by coolant or chips. Machining details can be emphasized using the zoom function.
- Turning:
HMI-Embedded includes broken-line graphics for turning in level G18. This permits checking of the program prior to actual machining. It can be triggered with a dry run feedrate or in real-time. Furthermore, the feed movements of the tool tip can also be recorded parallel to machining. The area to be magnified is selected using a cross-hair. The machining time is displayed during the simulation.

ManualTurn

ManualTurn contains 2D broken-line graphics in which every individual working step or the complete program can be simulated. This permits checking of the program prior to actual machining. Simulation can therefore be carried out prior to machining or also during machining. A zoom function is also present. The area to be magnified is selected using a cross-hair. The machining time is displayed during the simulation.

ShopMill

ShopMill uses SINUMERIK's high computing performance to achieve intelligent simplification of the programming of milling work. It also respects the knowledge that the solving of complex tasks (e.g. 3D surface sections) is reserved for appropriate CAD/CAM software. Therefore particular value has been placed on easy programming of simple workpieces, as encountered in the majority of parts for machining.

• General features:

The simulation implemented in ShopMill enables representation of the unmachined contour (parallelepiped only), of the tool diameter path with material removal in real-time or at high-speed, of the workpiece with plan and side views, the sectional representation of the 3D finished contour, representation of the machining zone with a variable zoom, and also calculation of the machining time.

With swiveled planes (inclined heads and tables), a simulation can be carried out for every individual swivel plane using an appropriate program structure and input of unmachined parts.

ShopMill possesses a static 3D simulation of the machined part. Users can convince themselves of the fault-free program even prior to machining of the first workpiece. Hidden contour elements can also be represented by cutting out the desired section following its selection using the cursor keys. The required machining time for the workpiece is displayed in the 3D simulation. The calculated machining time includes the dynamic acceleration processes of the traversing axes as well as tool replacement times.

The 2½D simulation in 3 views represents the tool path of the programmed tool with exact dimensions. The defined color shades represent different processing depths, and offer additional orientation support. This simulation serves in the sector "Programming" to check the result of the programmed part. In the sector "Automatic", it can be triggered with a dry run feedrate or in real-time. In addition to this, it can also be started parallel to machining in the optional mode "Monitoring". This may be necessary if viewing of the workpiece is hindered by coolant or chips. Machining details can be emphasized using the zoom function.

Simulation (continued)

- Graphics and simulation:

Wherever meaningful, ShopMill operates interactively with graphic support, providing the user with an overview of the plausibility of the entered data for each step. This provides additional assurance during program development, and shortens the programming and debugging times. Static auxiliary graphics and dynamic broken-line graphics are provided for each machining cycle.

The surface parameters required to describe the cycle are represented in the auxiliary graphic. In addition, the actual dimensions of the current elements are displayed in dynamic broken-line graphics.

If a tap hole drill and a tap are called, for example, their different diameters are displayed with the correct proportion to one another. In the case of superimposed technologies such as centering + drilling + deep-hole drilling + tapping, the broken-line graphics are superimposed on one another into a common representation. It can then be immediately seen whether the diameters of the used tools match one another. Drilling patterns are also represented with the correct proportions to the milled part just like the hole diameters.

Every input of a contour element is mapped immediately when the geometry processor has received enough information to calculate the contour. Ambiguous solutions are also displayed to permit the user to select the correct solution. The machining sequence can be graphically displayed following completion of the program or part of the program. The simulation uses the correct proportions for the tools and workpiece contours. The simulation is displayed as a plan view together with the two side views.

The side views are mapped in colored layers so that the machining depth can also be recognized in the plan view by differences in color. The tool movement in the workpiece shows the removed material in the form of animated graphics. The location of residual material can be clearly recognized.

When machining a workpiece with vertical spindle, chips or coolant frequently hinder viewing of the tool cutting edge. A real-time simulation running synchronously with the tool position permits the user to follow the machining status on the screen, permitting interventions if necessary for critical machining steps.

The machined part can be displayed as a volume model. It then vividly shows the result to be expected with the ShopMill programming. This guarantees that the appearance of the machined part corresponds to the user's ideas already prior to machining.

ShopTurn

ShopTurn contains a simulation for the produced program for horizontal turning machines. The simulation for machining on vertical turning machines is displayed horizontally. A differentiation is made with ShopTurn between »Simulation« (simulation prior to machining) and »Simultaneous recording« (real-time simulation during workpiece machining).

The machining with the tool cutting edge is displayed in both cases: the required data are obtained from the tool list, separate input of tool data for the simulation is unnecessary. Since the tool data are read directly from the NC memory, it is guaranteed that current data are used. The machining time is displayed in both simulation modes.

The dimensions of the unmachined part are entered in the program header of ShopTurn programs. If DIN/ISO programs are simulated, the unmachined part is not displayed.

- Simulation (prior to processing):

The ShopTurn simulation offers various display modes which can be selected using softkeys:

- Peripheral surface (default setting):

This view offers workpiece machining in the display most common to the operator.

The chuck is displayed in addition to the workpiece, and when machining with counterspindle also the rechucking process and further machining on the reverse side.

The workpiece is "divided into two":

The top half displays a section through the workpiece so that the internal machining can also be checked. The bottom half displays the peripheral surface of the workpiece. A selected area can be emphasized using the zoom function.

- Front face:

A softkey can be used at any time to switch over the display to the front face. In addition to zooming, sections can also be selected in the Z-direction in order to check the machining from the viewpoint of the front face on the peripheral surface.

- Simulation (prior to machining):

- 3D volume model (option):

The finished workpiece is displayed in this 3D view. Sections, rotations and zooms can be selected. This permits checking of internal machining.

- Simultaneous recording (option):

"Simultaneous recording" is understood to be simulation during machining of the workpiece. This display is appropriate if e.g. viewing of the workpiece is hindered by coolant or chips.

The machining data are delivered by the NC, permitting simultaneous display of movements. If "Program test" is selected prior to machining, "Simultaneous recording" permits influencing of the simulation in individual steps or with a feedrate override.

Glossary

Functions and Terms

SINUMERIK 810D/840D/840D

Skip blocks

CNC blocks which are not to be executed in every program pass can be skipped. Skip blocks are identified by placing a »/« character in front of the block number. The statements in the skip blocks are not executed, and the program resumes with the next block that is not identified as a skip block.

As many as 8 skip levels (level 0 to level 7) may be programmed. The individual skip levels can be activated via a data block in the PLC interface.

Spindle functions

Spindle modes are:

- Control mode
(constant spindle speed S or constant cutting speed G96)
- Oscillation mode
- Positioning mode
- Synchronous mode (synchronous spindle)
- Thread cutting/thread boring

Functions of the spindle modes:

- Spindle speed with spindle override
- 5 gear stages, specified in the
 - part program (commands M41 to M45) or
 - automatically via programmed spindle speed (M40) or
 - via PLC function block FC18.
- Oriented spindle stop (positioning mode) with SPOS ¹⁾
- Spindle monitoring with the functions ¹⁾
 - Axis/spindle stationary ($n < n_{min}$)
 - Spindle in setpoint range
 - Max. spindle speed
 - Programmable lower (G25) and upper (G26) spindle speed limitation
 - Min./max. speed of the gear stage
 - Max. encoder limit frequency
 - Target point monitoring for SPOS
- Constant cutting speed with G96 (in m/min or inch/min) at the tool tip for uniform turning finish and thus better surface quality. Spindle control via PLC for oscillation (for easier engaging of a new gear stage) and positioning

- Switch to axes operation:

For machining with a position-controlled spindle (end milling of turned parts, for example), the main spindle drive can be switched to axis mode with a program command. A common encoder can be used for both axis and spindle modes. The zero mark of the spindle is also the reference mark of the C axis. Referencing of the C axis is thus unnecessary (on-the-fly synchronization of the C axis).

- Thread cutting with constant lead ¹⁾:

With G33 you can produce the following thread types: cylindrical-, conical-, and face thread, single-start or multiple-start thread, as left-hand or right-hand thread. In addition, multiple-block threads can be produced by concatenating threading blocks.

- Thread cutting with variable lead ¹⁾:

Starting with software release 5.3 (SINUMERIK 840D), thread cutting can also be programmed with linearly progressive (G34) or linearly degressive (G35) lead.

- Programmable run-in and run-out of thread:

When thread cutting, you can use DITS/DITE (displacement thread start/end) to program the path ramp for the acceleration or deceleration process as a distance. This makes it possible, for example, to adjust the acceleration on the thread shoulder when the tool run-in or run-out is too short and initiate smoothing at the next CNC start.

- Tapping with compensating chuck/rigid tapping:

When tapping with compensating chuck (G63), the compensating chuck equalizes differences between spindle movement and drilling axis. Prerequisite for rigid tapping (G331/G332) is a position-controlled spindle with position decoding system. The traversing range of the drilling axis is therefore not restricted. By using the method whereby the spindle, as a rotary axis, and the drilling axis interpolate, threads can be cut to a precise depth (e.g. for blind hole thread).

Spindle speed limitation

→ Spindle functions

13

1) Prerequisite: position actual-value encoder (measuring system) with corresponding resolution (mounted directly on the spindle).

Spline interpolation (option)

Using spline interpolation it is possible to obtain a very smooth curve from just a few defined intermediate points along a set contour. The intermediate points are connected by polynomials. The compressor converts linear blocks (e.g. from CAD) at block transitions to splines of constant speed (COMPON) or splines of constant acceleration (COMPCURV). This yields soft transitions that reduce wear on the mechanical parts of the machine tool. However, if the intermediate points are placed close together, sharp edges can also be programmed. The spline interpolation function also considerably reduces the number of program blocks required.

Extremely "smooth" workpiece surfaces are often extremely important with mold and tool making, both optically and technologically, e.g. for rubber gaskets.

Starting with software release 5.3 (SINUMERIK 840D), it is possible with the COMPCAD compressor to approximate such "smooth" curves within the compressor tolerance (parallel tool paths), thus obtaining surfaces with a high optical quality even with larger tolerances.

Tool radius compensation is possible in spline interpolation, as it is in linear or circular interpolation.

Every polynomial can represent a spline. Only the algorithm determines the type of spline.

- A spline is only true to the tangents.
- B spline is true to the tangents and the curvature but does not run through the nodes (intermediate points).
- C spline is true to the tangents and the curvature and runs through the nodes.

Start-up support for SIMODRIVE 611 digital

For fast, user-friendly initial installation of the SIMODRIVE 611 digital drives and for optimizing the control loops, installation software is available for standard industrial PCs/PGs with an MPI card. The installation software is integrated in the PCU 50 (MMC 103).

With this software, the drive configurations can be entered and the drives initialized. The configuration of motor and drive module determines which standard data records are loaded. The drive and control parameters can also be archived on the PG/PC.

Additional aids are available for optimization and diagnostics.

Time range measuring function

- For optimizing current, speed and position controllers
- Setpoint input (cyclic testing signals) from integrated function generator
- Recording of setpoint and actual value progression with storage oscilloscope function

Frequency range measuring functions

- For optimizing the complete controlled system and analyzing the mechanical characteristics (resonance)

- Setpoint input (noise signal) from integrated function generator
- Integrated Fourier analysis with displaying of amplitude and phase path.

The measurement diagrams can be archived and are suitable for documenting the machine settings. They are an excellent means of quickly optimizing the current, speed and position control. In addition to conventional means of recording in the time range (step response of the speed and position control loop is a familiar method), it is also possible to analyze the behavior of the drive and machine in the frequency range using FFT (fast Fourier transformation).

Start-up trace

No additional oscilloscope is required for axis optimization with SINUMERIK 810D or SINUMERIK 840D, as the implemented installation and start-up software can be used to record up to 4 servo signals per position control cycle. The control system response can be specifically measured, for example on a block change and in the event of a change in the level of a digital signal. Trigger conditions and measuring duration for measured-value recording are freely selectable.

Subroutine levels and interrupt routines

Subroutines can be called not only in the main program, but also in other subroutines. Subroutines can be nested to a depth of 12 levels, including the main program level. That means that a main program may contain as many as 11 nested subroutine calls. Three levels are needed when you are using Siemens machining and measuring cycles. If such a cycle is to be called from a subroutine, the call can be nested at a depth of no more than 9.

Starting with software release 6 (SINUMERIK 840D), programs can also be called event-controlled following resetting of the parts program start or end, or following booting of the control. Users can then make the basic settings of functions or carry out initialization using a parts program command.

A system variable can be used to scan the event which activated the associated program.

Glossary

Functions and Terms

SINUMERIK 810D/840Di/840D

Subroutine passes

In order to execute a subroutine several times in succession, the desired number of repetitions can be programmed in the block with the subroutine call at address P (range of values: 1 ... 999).

Parameters are forwarded only when the program is called or in the first program pass. The parameters remain the same for all

subsequent passes. If you want to change the parameters between passes, you should make the relevant declarations in the subroutine.

Synchronous actions

→ Cross-mode actions

Even in its basic configuration, a SINUMERIK control allows you to initiate up to 16 actions synchronous to the axis and spindle movements. These actions execute in parallel with workpiece machining, and their inception can be determined on the basis of conditions.

The starting of such movement-synchronized actions (or synchronous actions for short) is therefore not restricted to CNC block boundaries.

Synchronous actions are always executed in the interpolation cycle. Several actions can even be carried out in the same IPO cycle.

Synchronous actions without validity identifier are active non-modally only in automatic mode. Synchronous actions with validity identifier ID are active modally in the subsequently programmed blocks in automatic mode. Statically effective synchronous actions with the identifier IDS remain active in all operating modes (see "Mode-independent actions").

The synchronous actions provide you with an excellent tool which allows you to respond very quickly to events in the interpolation cycle.

Here are some typical applications:

- Comparison operation-dependent or external signal-dependent transfer of auxiliary functions M and H to the PLC user software and subsequent machine responses
- Fast, axis-specific, input signal-based deletion of the distance-to-go
- External signal-controlled read-in disable for the CNC block
- Monitoring of system variables such as speed, power and torque
- Controlling process variables (speed, velocity, distance, etc.)

Limited functionality with SINUMERIK 810DE/840DiE/840DE:

Only one active synchronous function (SYNFCT) is possible at a time.

Synchronous actions, stage 2 (option)

More than 16 synchronous actions can be active in the CNC block. As many as 255 parallel actions can be programmed in each channel. Technology cycles can be combined into programs using synchronous actions, making it possible, for example, to start axis programs in the same IPO cycle by scanning digital inputs.

Limited functionality on the SINUMERIK 810DE/840DiE/840DE: The number of simultaneously traversed axes is still limited to four (path and positioning axes).

Synchronous spindles/multi-edge turning (option)

Angle synchronization of one leading and one or more following spindles enables on-the-fly workpiece transfer, particularly for turning machines, from spindle 1 to spindle 2, for example for the purpose of finishing, without experiencing the non-productive times normally associated with rechucking.

In addition to the speed synchronicity, the relative angular position of the spindles to one another, e.g. on-the-fly, position-oriented transfer of edged workpieces is also specifiable.

On-the-fly transfer:

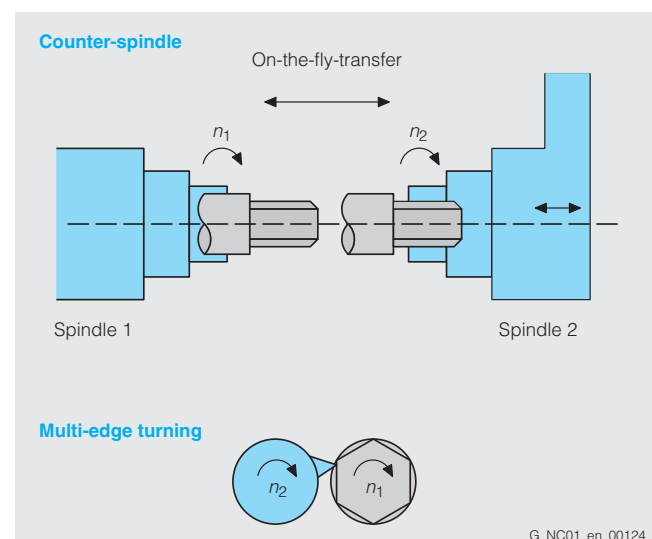
- $n_1 = n_2$
- Angle 1 = angle 2 or
- Angle 2 = angle 1 + angle Δ

Finally, specification of an integer transformation ratio between the main spindle and a "tool spindle" provides the prerequisites for multi-edge machining (polygon turning).

Multi-edge turning:

$$n_2 = T \cdot n_1$$

Configuring and selection take place in part via the CNC program or operator panel. Several pairs of synchronous spindles can be implemented.



Examples for synchronous spindles/multi-edge turning

G_NC01_en_00124

Tachograph

The so-called "tachograph" records all operator actions and pending alarms for diagnostics purposes.

Tangential control (option)

Tangential control makes it possible to correct a rotary axis in the direction of the tangents of two path axes. The two leading axes and the corrected axis lie in the same channel.

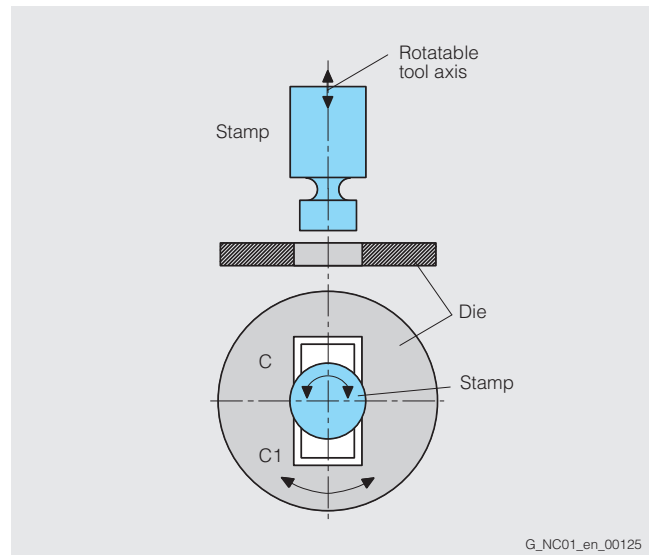
Applications:

- Tangential setting of a rotatable tool during punching/nibbling
- Correction of the workpiece alignment for a belt saw
- Setting of a dressing tool on a grinding wheel
- Tangential feed of a wire for 5-axis welding
- Setting of a cutting wheel for machining glass or paper

Tangential control is effective in all interpolation modes.

On punching and nibbling machines with rotatable punching tool and associated lower tool, the following functions may be used to ensure universality of the tool:

- Tangential control TANGON/TANGOF for vertical rotary axis alignment of the punching tool to the directional vector of the programmed path
- Coupled motion TRAILON/TRAILOF for homogeneous rotation of upper and lower tool (stamp and die)



Representation of a rotatable tool axis and die during punching/nibbling

Teach-in with HT 6 handheld terminal

"Teach-in" is generally taken to mean the transfer of current positions to the CNC program.

When teaching with the HT 6 handheld terminal in AUTO mode, it is possible not only to transfer the program but also to test and correct it immediately.

The program is stopped and the axes are moved into the desired position with the JOG keys. This position is transferred to the program as a traversing block and can then be started again at any point. A reset is not required. Positions already taught in the program can be corrected, and new positions can be inserted. Other program statements can be modified as required using the handheld programming unit's ASCII keypad.

Temperature compensation (option)

Heat causes machine parts to expand. The amount of expansion depends, among other things, on the temperature and on the thermal conductivity of the machine parts. The actual positions of the individual axes, which change on the basis of variations in temperature, have a negative effect on the precision with which workpieces are machined. These actual value modifications can be corrected with temperature compensation.

At a specific temperature, measure the actual-value offset over the positioning range of the axis to get the error characteristic for this temperature value. Error characteristics for different temperatures can be defined for each axis.

In order to ensure proper compensation of thermal expansion in changing temperatures, the temperature compensation value, reference position, and linear angle of lead must be forwarded from the PLC to the CNC via function blocks each time the temperature changes. Abrupt changes in these parameters are automatically smoothed by the CNC in order to prevent machine overload and avoid triggering watchdog monitors unnecessarily.

Thread cutting

→ Spindle functions

Thread cutting with compensating chuck/rigid tapping

→ Spindle functions

Glossary

Functions and Terms

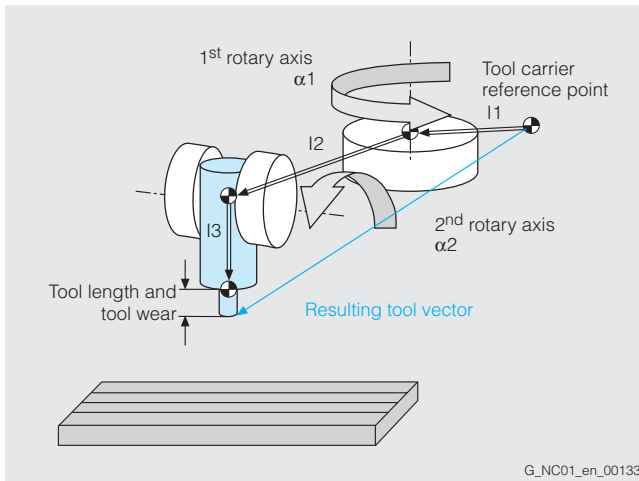
SINUMERIK 810D/840D/840D

Tool and process monitoring system

→ PROFIBUS tool and process monitoring

Catching errors before they happen. This is the motto for our SINUMERIK 840D, a control-integrated tool and process monitoring system. Active power monitors keep an eye on such things as breakage, wear, and missing tools. Also possible are precise status recognition and process optimization.

Tool carrier with orientation capability



Kinematics type T

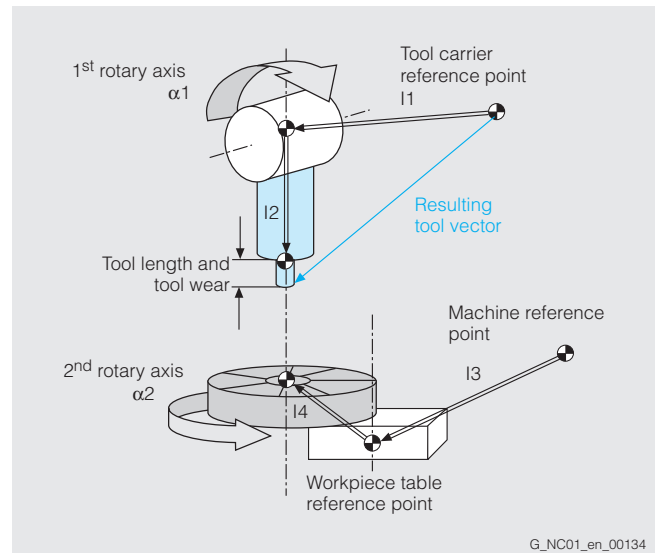
For machine tools with settable tool orientation, the SINUMERIK user can configure these kinematics without using 5-axis transformation. The "tool carrier with orientation capability" functionality enables $2\frac{1}{2}$ -/3-D machining with permanent spatial orientation of the tool/workpiece table.

Vectors I1 to I4 represent the geometrical dimensions of the machine. The rotary axes need not move in parallel to the Cartesian axes, but instead can be inclined at any angle (e.g. cardan milling head with 45° inclination).

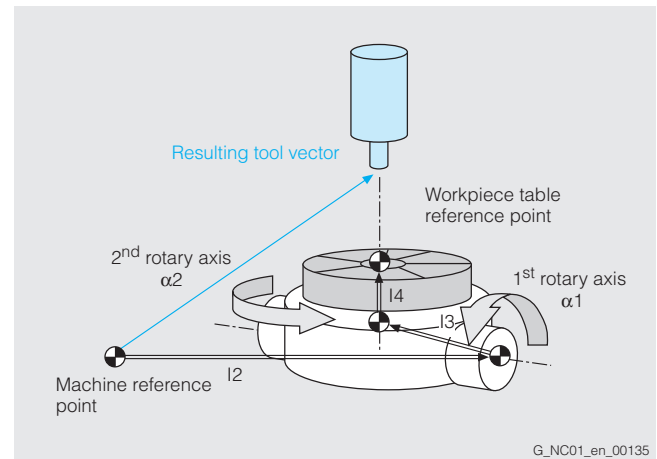
The angles α_1 and α_2 can be either specified or computed from the active frame and assigned to the tool carrier with orientation capability or to the workpiece.

The following kinematics can be flexibly configured:

- Rotatable tool: kinematics type T (tool)
- Rotatable tool/rotatable tool/workpiece table: kinematics type M (mixed)
- Rotatable workpiece table: kinematics type P (part)



Kinematics type M



Kinematics type P

Tool change via T number

In chain, rotary-plate and box magazines, a tool change normally takes place in two stages. A T command locates the tool in the magazine, and an M command inserts it in the spindle.

In turret magazines on turning machines, the T command carries out the entire tool change, that is, locates and inserts the tool. The tool change mode can be set using machine data.

Tool identification systems (option)

Within the framework of the tool loading and unloading dialog in the Siemens tool management system for SINUMERIK 810D/840Ds with HMI-Advanced/MMC 103 system software, you are provided with a link to an automatic tool identification system. This allows you to replace manual input of the tool data with automatic reading and writing of the tool code carrier.

During unloading, the data block for the tool is saved on the HMI-Advanced/MMC 103 system software; during loading, it is read from the HMI-Advanced/MMC 103 system software via the code

carrier and entered in the tool management system. In the interim, the tool data can be reedited as during tool selection from the tool catalog (correction data, etc.). Using an editable description file containing precise tool and cutting data, the code carrier data are converted during loading into dialog data which can be read by the tool management system. During unloading, the dialog data are converted back into code carrier data with the aid, once again, of the description file.

Tool management (option)

Tool management ensures that at any given time the correct tool is in the correct location and that the data assigned to the tool are up to date. Tool management is used on machine tools with circular magazines, chain magazines or box magazines. It also allows fast tool changes and avoids scrap by monitoring the tool service life and machine downtimes by using spare tools.

The most important functions of the tool management function are:

- Tool selection throughout all magazines and turrets for active tools and spare tools
- Ascertaining of a suitable empty location in dependence on tool size and location type
- Tool-dependent location coding (fixed and variable)
- Initiation of tool changes with T or M command
- Axis movements during a tool change with automatic synchronization when next D number is encountered
- Quantity and service life monitor with prewarning limit monitoring function

The HMI-Advanced/MMC 103 system software, the most user-friendly and most sophisticated configuration, makes it possible to utilize the tool management function to the limit of its capability, although HMI-Advanced/MMC 100.2 system software also provides you with the most essential task-related functions.

Missing tools can be loaded based on a decision made by the operator. Tools with similar wear characteristics can be combined into groups.

The tool management function also takes length compensation values for adapters that are permanently mounted at certain magazine locations and fitted with different tools into account.

With SINTDI, the SINUMERIK 810D/840D with HMI-Advanced/MMC 103 system software provides an upgrade to its tool management function which includes such things as tool balance and an online link to a tool presetting station.

Tool offset

By programming a T function (5-place integer number) in the block, you can select the tool. Every T number can be assigned up to 12 cutting edges (D addresses). The number of tools to be managed in the control is specified in the configuring data.

A tool offset block comprises 25 parameters, e.g.:

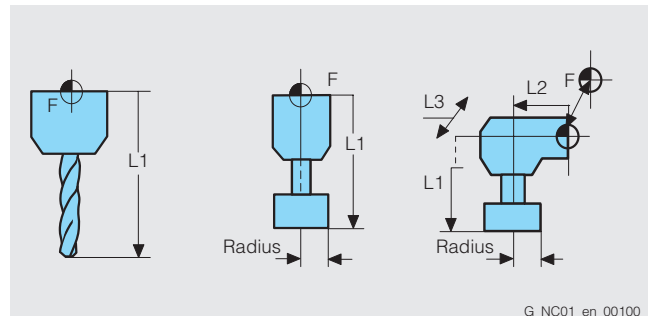
- Tool type
- Up to 3 tool length compensation values
- Radius compensation
- Wear dimension for length and radius
- Basic dimension

The wear and the basic dimension are added to the corresponding offset.

When writing the program, you need not take tool dimensions such as cutter diameter, cutter position or tool length into account.

You program the workpiece dimensions directly, for example following the production drawing. When a workpiece is produced, the tool paths, in dependence on the relevant tool geometry, are controlled so that the programmed contour can be produced with every tool used.

You enter the tool data separately in the control's tool table, and in the program you call only the required tool, with its offset data. During program execution, the control fetches the required offset data from the tool files and corrects the tool path for various tools automatically.



Tool offsets

Tool offset D can be programmed with reference to tool number T (when the Siemens tool management system is active, e.g. with monitoring functions and management of sister tools) or without internal references to existing tools.

You can define as many as 32,000 D values per control. D numbers can be assigned, checked, renamed, ascertained with the associated T number, invalidated, and activated on a site-dependent basis.

Tool types

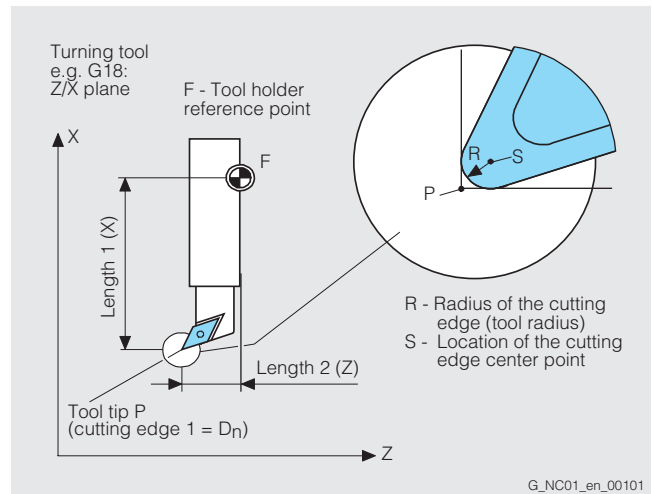
The tool type determines which geometry specifications are required for the tool offset memory, and how they are to be used. Entries are made for the relevant tool type in tool parameter DP. The control combines these individual components to produce a result variable (e.g. total length, total radius). The relevant total dimensioning goes into effect when the offset memory is activated. The use of these values in the axes is determined by the tool type and current machining level G17, G18 or G19.

The following tool types can be initialized:

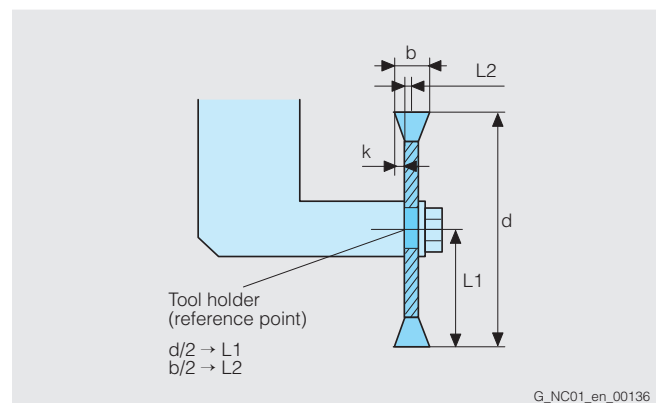
- Group 1xy: milling tools (from ballhead cutter to cone frustum cutter)
- Group 2xy: drills (from twist drill to reamer)
- Group 4xy: grinding tools (from surface grinding wheel to dresser)
- Group 5xy: turning tools (from roughing tool to threading tool)
- Group 700: slotting saw

The saving of all tool offsets is supported by input screen forms.

For wood, the "slotting saw" tool is available as tool type.

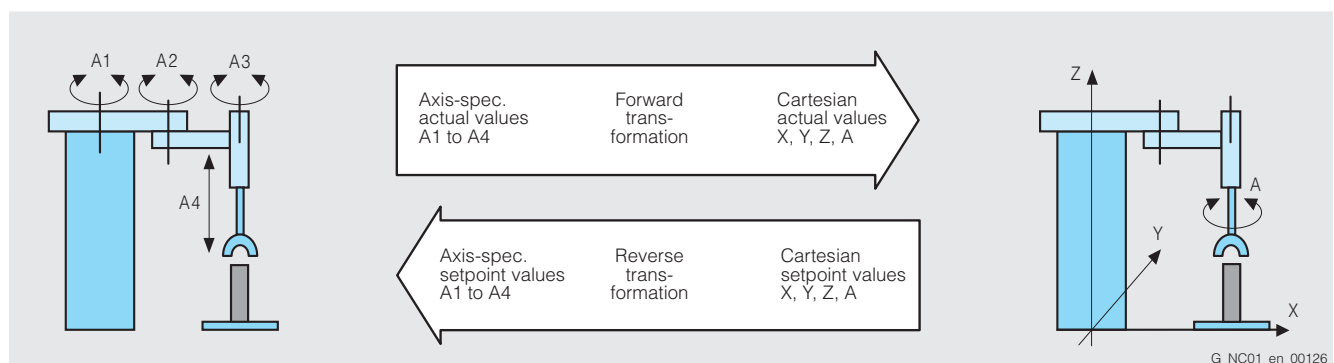


Geometry of turning tool



Geometry of slotting saw

Transformation package for handling devices (option)



Transformation package for handling device

The "transformation package for handling devices" contains the so-called standard transformation block, with whose help typical 2-axis to 5-axis handling devices such as gantries or SCARAs can be operated. This coordinate transformation package converts the axis-specific actual values for the axes (e.g. A1 to A4) to Cartesian values (e.g. X, Y, Z, A) and the programmed Cartesian setpoints back into axis-specific values for the handling devices. Thanks to this coordinate transformation, the movements of the handling device become simpler and more user-friendly. The handling device can be set up, that is, manual

traversed not only in the axis-specific coordinate system, but also in the handling device's own Cartesian coordinate system, using, for example, the jog keys on the handheld programming unit. Kinematic adaptation of the transformation is carried out via the machine data.

Glossary

Functions and Terms

SINUMERIK 810D/840D/840D

Transformation, generic

The function "Generic transformation" is used to define the tool orientation as desired in the space with the initial setting of the axes, and not just according to the Z-direction. It can then be used more flexibly and universally.

It is then possible to also control machine kinematics by the CNC where the orientation of the rotary axes is not exactly parallel to the linear axes.

Starting with software release 6 (SINUMERIK 840D), the generic 5-axis transformation was extended to the 3-axis and/or 4-axis transformation, i.e. it is also possible for machines with only one rotary axis (rotatable tool or workpiece).

TRANSMIT/peripheral surface transformation (option)

The "TRANSMIT" function is used for milling external contours on turned parts, e.g. square parts (linear axis with rotary axis).

As a result, programs become much more simple and complete machining increases machine efficiency. Turning and milling can be performed on one machine without rechucking.

3D interpolation with two linear axes and one rotary axis is possible. The two linear axes are mutually perpendicular and the rotary axis lies at right angles to one of the linear axes.

"TRANSMIT" can be called up in different channels simultaneously. The function can be selected and deselected with a preparatory function (straight line, helix, polynomial and activating tool radius compensation) in the part program or MDA.

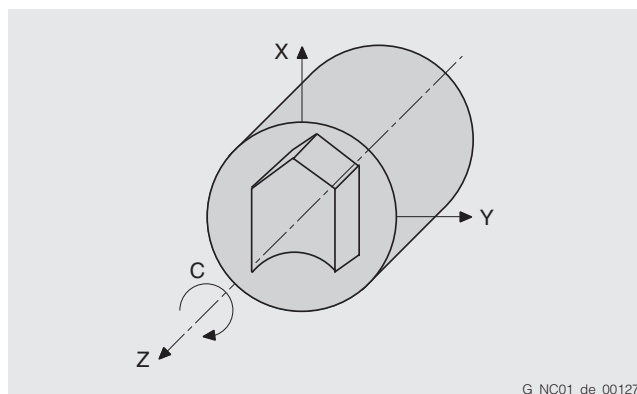
With TRANSMIT, the area of the transformation pole is reached when the tool center can be positioned at least to the turning center of the rotary axis entering the transformation.

TRANSMIT through the pole is implemented in different ways:

- When travelling through the pole, the rotary axis is turned automatically by 180° when the turning center is reached and then the remaining block is executed.
- When traversing close by the pole, the control automatically reduces the feed-rate and the path acceleration.
- If the path contains a corner in the pole, the position jump in the rotary axis is compensated by the control through automatic block insertion.

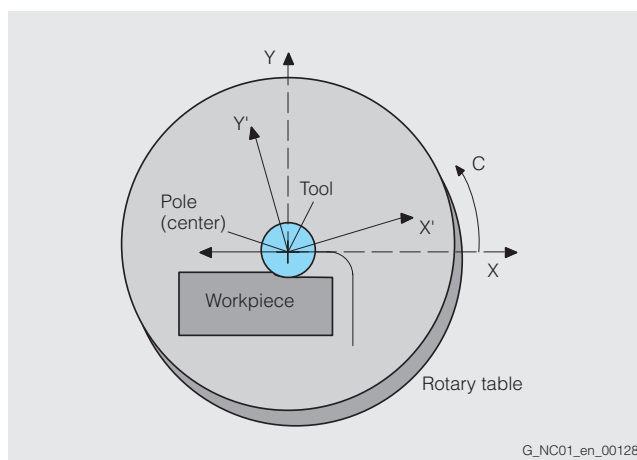
Peripheral "surface transformation" is used on turning machines and milling machines, and enables peripheral surface transformation, e.g. for turned parts.

The TRACYL peripheral surface transformation or cylinder surface transformation can be used to manufacture grooves of any shape on the surface of cylindrical bodies with or without correction of the groove side. The shape of the grooves is programmed referred to the plane cylinder surface processed.



G_NC01_de_00127

End face machining with TRANSMIT



G_NC01_en_00128

Tool center-point path through the pole

Travel to fixed stop (option)

With this function, tailstocks or sleeves can be traversed to a fixed stop to clamp workpieces. The pressure applied can be defined in the part program. Several axes can be traversed to a fixed stop simultaneously and while other axes are traversing.

The "extended travel to fixed stop" function can be used to adapt torque or force on a modal or block-related basis, travel with limited torque/limited force (force control, FOC) can be initiated, or synchronous actions can be used at any time to program traversing functions.

Traversing range

The range of values for the traversing ranges depends on the selected computational resolution. The following ranges of values can be programmed when the default value is specified in the machine data field "computational resolution specified in the table for linear or angular position" (1,000 increments per mm or degree):.

If the computational resolution is increased/decreased by factor 10, then the value range changes accordingly.

The traversing range can be restricted by software limit switches and operating ranges.

	G70 [inches, degrees]	G71 [inches, degrees]
Linear axes X, Y, Z, ...	± 399,999.999	± 999,999.999
Rotary axes A, B, C, ...	± 999,999.999	± 999,999.999
Interpolation parameters I, J, K	± 399,999.999	± 999,999.999

Universal interpolator NURBS

Internal motion control and path interpolation are performed using NURBS (non uniform rational B-splines). This provides a uniform method for all internal interpolations that can also be used for future complex interpolation tasks.

The following input formats are available irrespective of the internal structure: Linear, circular, helical involute interpolation, splines (A, B, C) and polynomials.

User interface

The user interface has a clear layout with eight horizontal and eight vertical softkeys. The use of windows permits simple and user-friendly operation.

The interface is subdivided into six control areas:

- Machine
- Parameters
- Program
- Services
- Diagnostics
- Start-up

In this way, it is possible to write a part program while machining is in progress and to transfer data from an external storage unit at the same time. On changing the control area, the last active menu is always stored. There are two hotkeys for switching control areas.

User machine data

The NCK makes machine data available for configuring the PLC user program. These user machine data are stored in the NCK-PLC interface during the control run-up-prior to PLC power-up. The PLC basic program reads these data from the NCK-PLC

interface during its initialization process. This means that specific machine configurations, machine expansions and user options can be activated.

Velocity

The maximum path velocity, axis speed and spindle speed are affected by the machine and drive dynamic response and the limit frequency of actual-value acquisition (encoder limit frequency and limit frequency of the input circuit). The resulting velocity from the programmed path lengths in the CNC block and interpolation cycle (IPO cycle) is always limited to the maximum velocity or, in the case of short path lengths, reducing to the velocity that can be travelled during one IPO cycle.

The minimum velocity must not go below 10^{-3} units/IPO cycle. The minimum and maximum axis speed are dependent on the selected computational resolution. The maximum velocity of the axis is generally limited by the mechanics or by the limit frequency of the encoder or actual-value acquisition. The velocity value range is not limited by the CNC (max. 300 m/s).

Glossary

Functions and Terms

SINUMERIK 810D/840D/840D

Working area limitation

→ Zero offsets

Working area limitations describe the area in which machining is permitted.

These limitations refer to the basic coordinate system. A watchdog checks to see whether the tool tip has penetrated the protected working area (also taking into account the tool radius). One value pair (plus/minus) per axis may be used to describe the protected working area.

The upper and lower working area limits, which can be set and activated via setting data, may be modified using the G25/G26 commands. Working area limitations restrict the traveling range of the axes, augmenting the restrictions made by the limit switches. Protective zones are thus set up in which tool movements are prohibited and which protect equipment such as tool revolvers, measuring stations, etc., from damage.

Working plane

→ Tool radius compensation

When specifying the working plane in which the desired contour is to be machined, the following functions are defined at the same time:

- The plane for the tool radius compensation
- The infeed direction for the tool length compensation depending on the type of tool
- The plane for circular interpolation

When calling the tool path correction G41/G42, the working plane must be defined so that the control can correct the tool length and radius.

In the basic setting, the working plane G17 (X/Y) is preset for drilling/milling, and G18 (Z/X) for turning G18 (Z/X).

Workpiece-related actual value system

The term "workpiece-related actual value system" is used to designate functions which allow the SINUMERIK user to:

- Begin machining in a workpiece coordinate system defined via machine data in JOG and AUTOMATIC mode without any additional manipulations after powering up the control
- Retain the valid settings relating to active level, settable frames (G54-G57), kinematic transformations, and active tool compensation at the end of the part program for use in the next part program

- Switch back and forth between the WCS workpiece coordinate system and the MCS machine coordinate system by making an appropriate entry on the PCU
- Change the workpiece coordinate system (e.g. by changing the settable frames or tool compensation)

Zero offsets

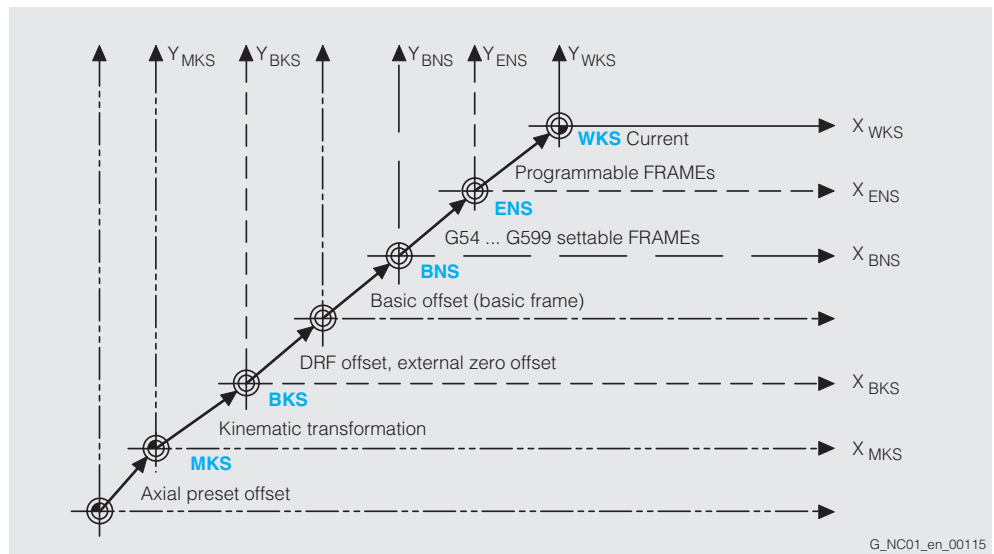
→ Frame concept

According to DIN 66217, right-handed, rectangular (Cartesian) coordinate systems are used in machine tools.

The following coordinate systems are defined:

- Machine coordinate system MCS
The machine coordinate system is formed by all the physical machine axes.
- Basic coordinate system BCS
The basic coordinate system consists of three Cartesian axes (geometry axes) as well as other non-geometric axes (auxiliary axes).
- BCS and MCS are always in conformance when the BCS can be mapped onto the MCS without kinematic transformation (e.g. TRANSMIT/interfacial transformation, 5-axis transformation, and max. three machine axes).
- Basic zero system BZS
DRF offsets, external zero offsets and basic frames maps the BCS on the BZS.
- Settable zero system SZS
An active settable zero offset G54...G599 transforms the BZS into the SZS.
- Workpiece coordinate system WCS
The programmable frame determines the WCS representing the basis for programming.

You thus use zero offsets to transform your machine zero point into the workpiece zero point to simplify programming. You may choose from among various zero offsets:



Coordinate systems

- Settable zero offsets:
You can enter offset coordinates, angles and scaling factors in up to 100 possible zero offsets (G54 ... G57, G505 ... G599), for example, to enable cross-program calling of zero points for different devices or gripping fixtures. The zero offsets can be suppressed by block.
- Programmable zero offsets:
Zero offsets can be programmed with TRANS (substitution function, basis G54...G599) or ATRANS (additive function). This allows you, for example, to work with different zero offsets for repetitive machining operations at different positions on the workpiece. G58/G59 make previously programmed zero offsets axially replaceable.
- External zero offsets:
You can also activate axis-related linear zero offsets via the PLC user software (function blocks) with assignment of system variable \$AA_ETRANS [axis].

Zero speed monitor

→ Positioning monitor

The zero speed monitor represents one of the most comprehensive mechanisms for monitoring axes. The monitor checks to see whether the following error has reached the zero speed tolerance limit following the elapse of a programmable time period. Upon termination of a positioning procedure, the zero speed monitor takes over for the positioning monitor, and check to see whether the axis moves further from its position than stipulated in the machine data's "zero speed tolerance" field. The zero speed monitor is always active following expiration of the "zero speed delay" or upon reaching the "fine exact stop" limit as long as no new traversing command is pending.

When the monitor responds, an alarm is generated and the relevant axis/spindle brought to zero speed with rapid stop via a set speed ramp. The zero speed monitor is effective for linear and rotary axes as well as for position-controlled spindles. The zero speed monitor is inactive in follow-up mode.

Glossary

Functions and Terms

Positioning modules

Acceleration with jerk limitation

To couple optimum acceleration performance of the machine with reduced wear on the mechanical parts, you can choose between sudden acceleration and continuous (jerk-free)

acceleration. With "jerk-free" acceleration the velocity curve describes an approximately sinusoidal shape.

Backlash compensation

"Backlash compensation" compensates for mechanical backlash, for example leadscrew backlash. Backlash compensation can be entered separately for each axis.

Block search

For testing part programs or after machining has been interrupted, it is possible to select any point in the part program using the block search function in order to start or resume at this point. The

block search can be executed forward or backward in the part program. The following can be selected as a search destination:

- Block number in the main program
- Number of passes and block number in a subroutine.

CNC programming language

The basis for the CNC programming language is DIN 66025.

Dimension specifications in metric measure and inches

Position and gradient values can be entered in the part program in inches. The control can be set to a basic system regardless of the programmed measuring system.

Example: The basic system is metric
An inch thread is to be machined within a metric program. All data such as tool offsets, zero offsets and feedrates remain metric.

Drift compensation

FM 354

During the axis's constant velocity phase, the analog speed control loop's drift is automatically compensated. The function can be selected via system parameters.

Drives for servomotors

FM 354/FM 357-2

The FM 354 and FM 357-2 each provide one analog ± 10 V interface per axis to the SIMODRIVE 611 analog converter system.

The PROFIBUS drives SIMODRIVE 611 universal as well as POSMO CD/CA and POSMO SI can additionally be operated on an FM 357-2 (not with FM 357-2H and HT 6).

Drives for stepper motor

FM 353/FM 357-2

One interface for clock and directional signals is available for each axis on the FM 353 and FM 357-2 for the control of power sections for stepper motors. The stepper motor can be driven via

the FM STEPDRIVE power module. On the FM 357-2, position-controlled operation with incremental and absolute value encoders as well as hybrid operation with analog servo axes is possible (e.g. 2 axes stepper/2 axes servo).

Follow-up mode

The setpoints are not output by the control, but the control remains synchronized with the machine (follow-up mode). Traverse paths can be checked on the display.

High-speed inputs/output

FM 353/FM 354

Four digital inputs and outputs are available for high-speed process signals. The following functions can be assigned via machine data:

- High-speed inputs:
External start, enable, measurement, reference point switch or reversing cams for reference point approach.
- High-speed outputs:
Position reached (stop), axis movement forward/backward, modification M97/M98, or start enable.

FM 357-2

The FM 357-2 is equipped with 18 high-speed inputs, four of which are used for reference point approach and 2 of which can also be evaluated without a delay for "high-speed measurement". Furthermore, the FM 357-2 has 8 freely-usable fast outputs. If required, 2 standard S7-300 I/O module can be inserted at the right of the FM 357-2 and be operated as high-speed NC I/O module by defining a "local segment". Authorized modules are 16-bit input/output modules and analog output modules (on request).

In-process measurement

The "in-process measurement" function enables delay-free storage of the current actual position value based on an external switching event. The measurement is executed independently of

the internal computing cycle. This value can then be post-processed in the CNC program for high-precision length acquisition or print-mark detection.

Installation and start-up support

A tool is available for each of the FM modules which will be integrated as auxiliary software in STEP 7. This tool is a component of the relevant configuring package, and must be obtained separately.

All FM start-up tools contain screen form-driven start-up assistants, program editors, and diagnostic functions.

In addition, every configuring package is accompanied by a "getting started" leaflet intended primarily for newcomers to help them to a successful start.

Languages

The display texts for user guidance and the system messages and alarms are available in the four languages German, English, French and Italian.

Main programs/subroutines

If machining operations recur frequently, it is advisable to put them in a subroutine. The subroutine is called from a main program (250 passes, ≤ 9999 on the FM 357-2).

One subroutine level (11 on the FM 357-2) are possible in a main program.

Modes

- AUTOMATIC
In AUTOMATIC mode, the part programs to be executed are selected and started. Control of AUTOMATIC mode by:
 - Skip block
 - Programmed stop
 - Rapid traverse override
 - AUTOMATIC single block
- MDI (manual data input)
In MDI mode, it is possible to enter program blocks or sequences of blocks with-out reference to a main program or subroutine and then to execute them immediately via start.

- JOG
In JOG mode, it is possible to set up the machine. The axis can be moved in this mode using the direction keys.
- Reference point approach
The modes are supplemented by the functions preset (set actual value) and teach-in.

Glossary

Functions and Terms

Positioning modules

Monitoring functions

The position controls contain permanently active monitoring functions which recognize faults in the control, the PLC and the machine so early that damage on the workpiece, tool or machine can be avoided. In the event of a fault, the machining sequence is interrupted, and the drive stopped. The cause of the fault is saved, and displayed as an alarm. The PLC is informed at the same time that an alarm is present. Monitoring functions exist for the following areas:

- Reading
- Format
- Measuring circuit
- Encoder and drive
- Enabling signals
- Voltage
- Microprocessors
- Serial interfaces
- Transmission between control and PLC
- System memory and user memory

Multi-point interface MPI

FM 353/FM 354/FM 357-2

When FM 353, FM 354 and FM 357-2 are used, the SIMATIC S7-300 CPU provides the multi-point interface (MPI).

Operator interface, OP7/OP17/OP27

FM 353/FM 354

Standard screen forms are displayed on the OP7 and OP17 for FM 353/FM 354.

FM 357-2

Example screen forms for OP17 and OP27 are delivered with the FM 357-2. These screen forms can be loaded and modified using ProTool.

Override/time override

The override acts as a pure velocity override, i.e. it modifies the velocity of the traversing movement by a percentage value. The time override also acts on the set acceleration and brake ramps.

PLC program memory

In the PLC user memory of the PLC CPU, the PLC user program and the user data are stored together with the standard software technology functions.

The memory of the PLC CPU is divided into load memory, working memory, and system memory. The load memory is a permanent memory, and in addition to the data required for execution also contains data, program and decompilation info. The load

memory can be integrated in RAM or read-only memory (flash) or is available in the form of a slot-in memory card. When the PLC CPU powers up, the code required for execution and the data from the load memory are loaded into working memory.

All standard S7-300 CPUs can be used in conjunction with FM modules. The minimum configuration, which differs from one FM to another, must be observed in all cases.

Positioning axes

Positioning axes can execute movements simultaneously with machining, thus reducing non-productive times considerably. They can be used to advantage for controlling workpiece and

tool loaders or tool magazines. They can be programmed with an axis-specific feedrate in the part program. Axis movement beyond block boundaries is also possible.

Preset

With the machine function "preset" (set actual value), the control zero can be redefined within the machine coordinate system.

Preset does not move the axes, but a new position value is entered for the current axis positions.

Programmable acceleration

With the "programmable acceleration" function, it is possible, for example, to modify the axis acceleration in the program in order to limit mechanical vibration in critical program sections.

The path or positioning axis is then accelerated at the programmed value. The maximum acceleration value stored in the control is not exceeded. As part of intelligent motion control, this function provides a more precise workpiece surface.

Reference point approach

When using a machine axis in program-controlled mode, it is important to ensure that the actual values supplied by the measuring system agree with the machine coordinate values.

The reference points (limit switches) are approached at a defined speed. If linear measuring systems with distance-coded reference marks are used, reference point approach is reduced.

Rotary axis

Depending on the application, the traversing range of a rotary axis can be selected to be limited to less than 360 degrees or to

be endlessly turning in both directions. Endlessly-turning rotary axes are used for out-of-round turning, grinding, and winding.

Software limit switches

Software limit switches limit the traversing range of an axis. The axis stops at the software limit switch.

User memory

All programs and data such as part programs, subroutines, comments, tool offsets and zero offsets can be stored in a common user memory. The user memory is located on the module.

Glossary

Functions and Terms

AC motors

Bearing service life

The motors have permanently lubricated bearings. They are designed for a minimum ambient temperature in operation of -15°C (+5 °F).

Recommended bearing replacement interval: 8,000 to 20,000 h (depending on motor type; for details see motor operating instructions). Grease renewal after: 8,000 h.

Cable outlet

AS = cable outlet → direction drive end

BS = cable outlet → direction non drive end

Degree of protection


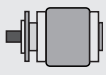
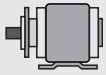

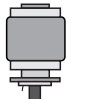
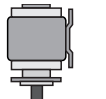


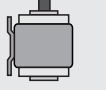
The designation for the degree of protection according to EN60034-5 and IEC60034-5 is made using the letters "IP" and two digits (e.g. IP 64). The second digit in the designation represents the protection against water, the first digit the protection against penetration of foreign matter.

Since coolants are used which contain oil, are able to creep, and may also be corrosive, protection against water alone is insufficient. The designation for the degree of protection should only be considered here as a guideline. Our sealing systems are based on many years of practical experience, exceed the IEC definitions by far, and are appropriate to the requirements of machine tools.

The table opposite can serve as a decision aid for selecting the proper degree of protection for servomotors. With the IM V3/V19 designs, permanent liquid on the flange is only permissible with IP 67/IP 68.

Fluids	General shopfloor environment	Water; gen. coolants (95% H ₂ O, 5% oil); oil	Oil creepage; petroleum; aggressive coolants
Dry	IP 64	-	
Water-enriched environment	-	IP 64	IP 67 ¹⁾
Mist		IP 65	IP 67
Splatter	-	IP 65	IP 68
Stream	-	IP 67	IP 68
Surge, brief immersion; constant inundation	-	IP 67	IP 68

Design

Design	Designation	Design	Designation	Design	Designation
	IM B3		IM B5 IM B14		IM B35
	IM V5		IM V1 IM V18		IM V15
	IM V6		IM V3 IM V19		IM V36

Holding brake

1FT/1FK servomotors

1FT/1FK servomotors can be supplied with an integrated holding brake for holding the feed axis stationary without play or when the system is deenergized.

The holding brake is actuated by the PLC of the control or via SIMODRIVE 611 universal. It can thus be made to react to any machine state.

1PH4 main spindle motors

An electromagnetic single-disc brake can be mounted on the drive-end shield to hold the motor shaft and the main spindle stationary without backlash.

1) IP 64 with dry running at shaft exit.

Noise emission

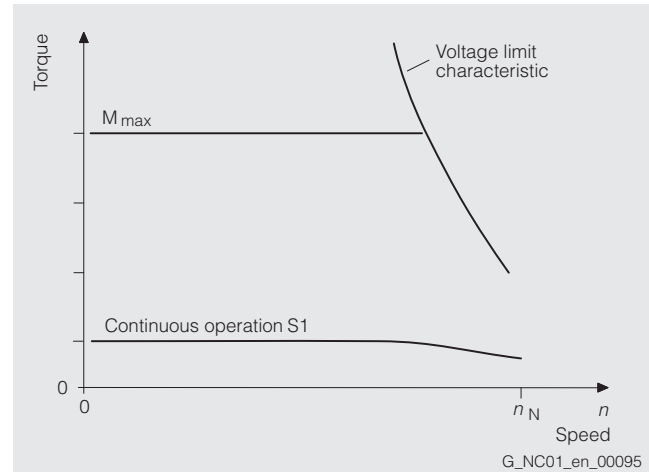
The noise emission values for a distance of 1 m from the motor being run with the SIMODRIVE 611 converter system are defined according to EN 21680.

Overload capability

1FT/1FK motors

If there is a requirement for short non-productive times (e.g. machine tools) or short pulse-dependent cycles (transfer lines), the drives must be able to endure high dynamic and thermal loads.

The AC servomotors therefore provide a practically constant torque and constant overload capability (4x static torque) over a wide speed setting range. The SIMODRIVE 611 converter is rated such that the overload capability can be adapted to the dynamic demands and the friction of the drive. The switch-on duration is standardized to IEC 60034-1. If no on/off cycle duration is specified, it is 10 minutes.



Paint finish

1FT/1PH4 motors

The 1FT and 1PH4 motors have a standard coat of high-quality two-component epoxy resin paint that is resistant to most additives used in coolants. This paint finish makes the motor suitable for use in the tropics without a special paint finish. The housing of the servomotors consists of an aluminium continuous casting. This material is very corrosion-resistant. Even if the paint finish is damaged, the motor can still be exposed to hostile environments.

1FN linear motors

The housing of the primary sections consists of stainless steel and is supplied without any paint finish. The carrier of the secondary sections is painted cobalt blue, RAL 5013.

Radial and axis concentricity and coaxiality

The concentricity tolerance is decisive if gears are to be mounted. Too large a tolerance causes the gears to run noisily and the system to vibrate. By using the most modern CNC

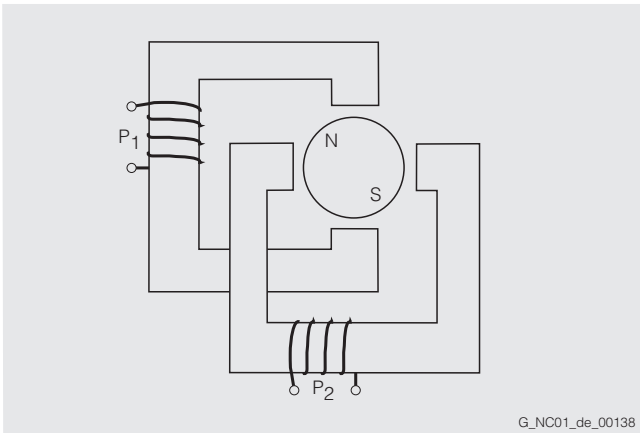
machines to manufacture the motors, tolerances have been reached that are way above grade N (normal).

Glossary

Functions and Terms

Stepper motors

3-phase microstep procedure



Fundamental function of permanent-magnet stepper motor

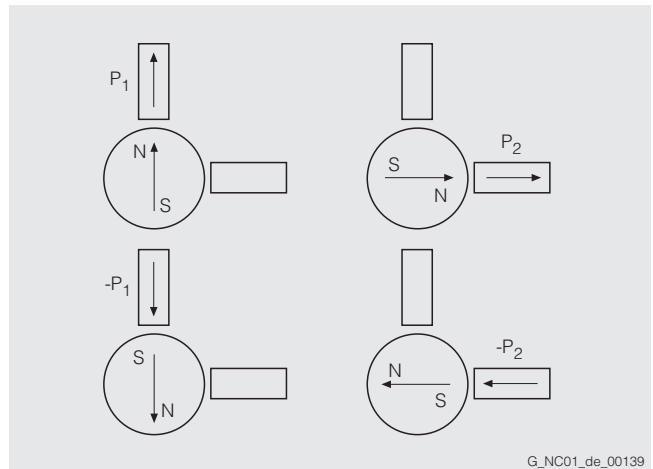
The stepper motor is controlled by the alternate flow in the windings. The rotor follows the rotating field.

The control procedure can be readily recognized using the example of a 2-phase motor. With continuous stepping mode, only one phase is switched on.

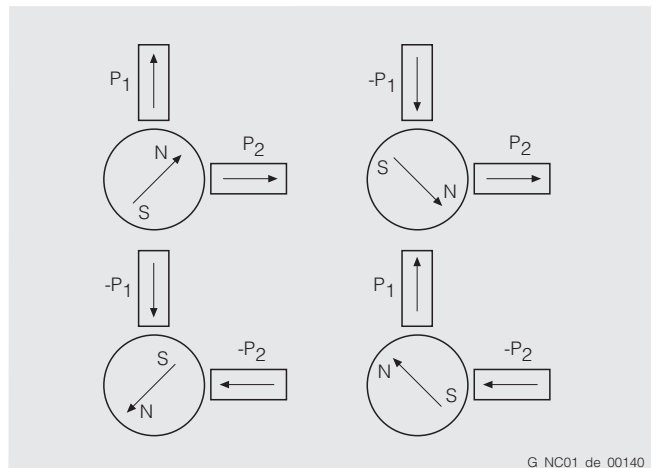
The positive and negative flows of the phases result in four possible rotor positions.

If the two phases flow simultaneously, the rotor is positioned between the poles. The periodic change in the two control procedures results in half-step mode with 8 possible rotor positions.

With the 3-phase microstep procedure, any rotor position between the phases can be controlled by discrete current values of sinusoidal shape.



Control procedure: one phase flowing



Control procedure: two phases flowing

Analogies

Position \triangleq number of steps
 Speed \triangleq frequency \triangleq steps/time

Acceleration \triangleq change in frequency with respect to time.

Encoder feedback

An encoder feedback is not present with stepper motors. With correct mechanical design, the motor does not fall out of

synchronism, and does not lose any steps either.

Holding torque

In every step position, the rotor is held because of the electric DC excitation of the windings, providing its holding torque M_H is not exceeded on the motor shaft.

Frequency

The revolving field, in the context of a synchronous motor, is prescribed in increments by the control frequency and by a directional signal. The speed is the result of the specified control frequency and the set number of increments per motor revolution. The acceleration and braking phase is normally handled by an upstream control with a ramp function so that the motor does not pull out of synchronism.

An important motor parameter is the start/stop frequency, which determines the maximum frequency with which the motor can be directly controlled without controlled acceleration or braking ramp.

Speed

Above a certain control frequency which depends on the motor type and the mechanical load, the step-by-step motion of the motor shaft becomes a continuous rotation.

The following then applies to the speed n of the motor:

$$n = \alpha / 360^\circ \cdot f_z(\text{Hz}) \cdot 60 \text{ rpm}$$

f_z = control frequency

Step

A step is understood to be the procedure where the motor shaft rotates by the angle α as the result of a control pulse.

The number of rotor steps per revolution is referred to as the step number.

Torque

If the rotating motor shaft is loaded with a torque M_L , the motor follows the control frequency in synchronism unless the load torque exceeds the operating limit torque M_{Bm} . In this case, the rotor can no longer follow the control frequency, the control frequency and step frequency are no longer equal, and the motor loses steps.

With correct selection of the motor and control, this case does not occur.

A constant torque is available up to a speed of approx. 2000 rpm.

Glossary

Functions and Terms

SIMODRIVE 611 converter systems

C-axis operation

Analog system

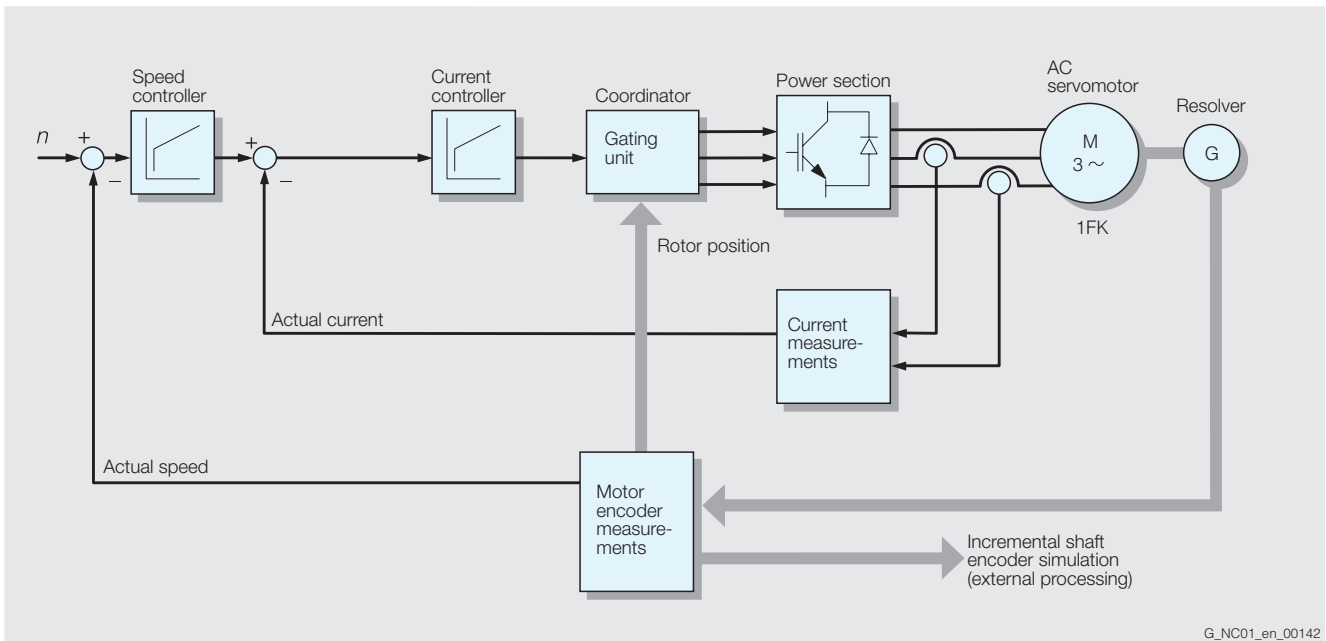
The SIMODRIVE 611 drive modules with analog closed-loop plug-in unit for main spindle drives are also designed for C-axis operation.

Digital system

With the SIMODRIVE 611 digital system, the C-axis functions are implemented in direct communication with the

SINUMERIK 810D/ 840D/840C. The standard high-resolution actual position acquisition provided by the servo drive control system allows the integrated motor measurement system to achieve a maximum resolution of 4,194,304 positioning increments per revolution (performance control). This produces, for example, a positional resolution of approximately 1/10,000 degrees.

Closed-loop control, analog system for 1FK motors with resolver

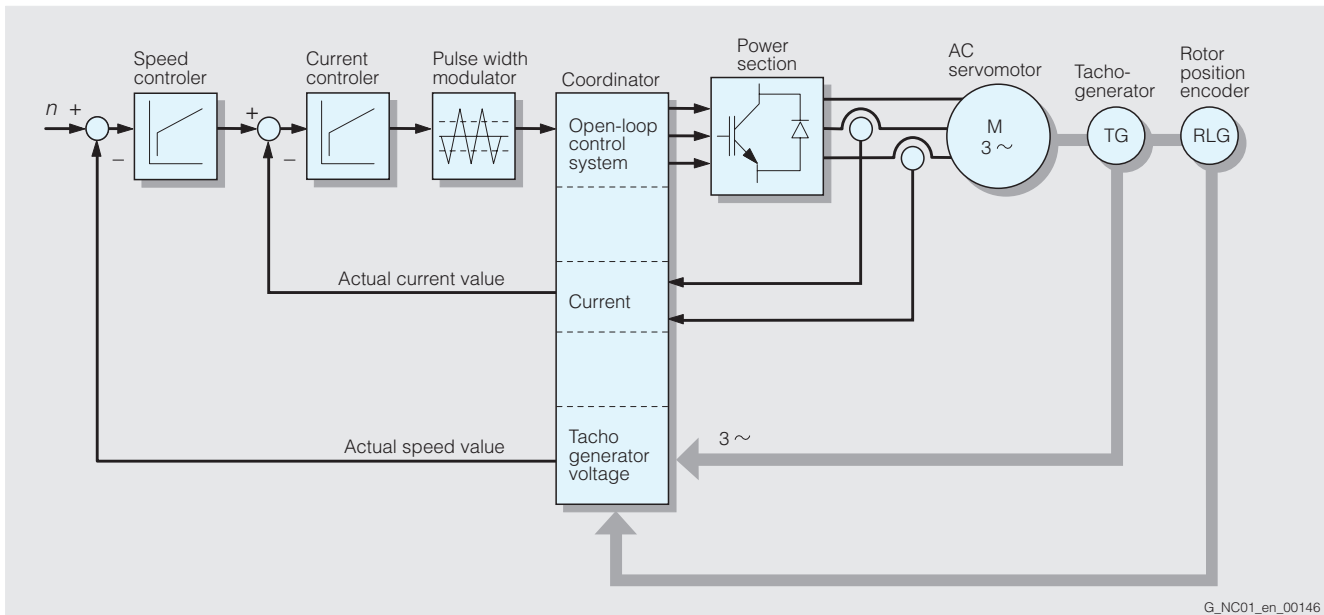


Feedrate control in the analog system with 1FK motors with resolver

This control variant is intended for applications in which positioning accuracy is not a primary concern. The control has been specially designed for use with sine-commutated 1FK motors.

The resolver encoder system integrated in the motor supplies the signals required by the control system, such as motor rotor position and actual speed, as well as an incremental shaft encoder interface with TTL signals for external processing.

Closed-loop control, feed, analog



Analog feed control with 1FT5 motors

Analog closed-loop control with 1FT5 motors

Closed-loop control of the analog feed modules uses tried and tested analog technology and includes a speed control system with a subordinate current control system for square-wave current commutated 1FT5 AC servomotors with tachogenerator and rotor position encoder.

The setpoint output from the higher-level position control is implemented using a standardized ± 10 V interface. It is possible to set either current or speed control using externally switchable terminals.

Closed-loop control, feed, digital

Closed-loop control, digital drive system

Closed-loop control of the digital feed modules is based on a powerful signal processor which performs the axis-specific current and speed control functions. Data transfer to the position control of the SINUMERIK 810D/840D/840C is handled by a communication block. The current and speed control systems are based on an easy-to-set state controller. The closed-loop control system is specially designed for use with the sinusoidal-current commutated 1FT6 and 1FK three-phase servomotors as well as with the 1FN linear motors (with performance control). In addition to excellent dynamic response characteristics, the digital closed-loop control system also has programmable filters with which mechanical resonances can be attenuated. This attenuation means that the speed control gain can be significantly increased in many cases, and possibly also the position control (K_V).

Closed-loop control, SIMODRIVE 611 universal

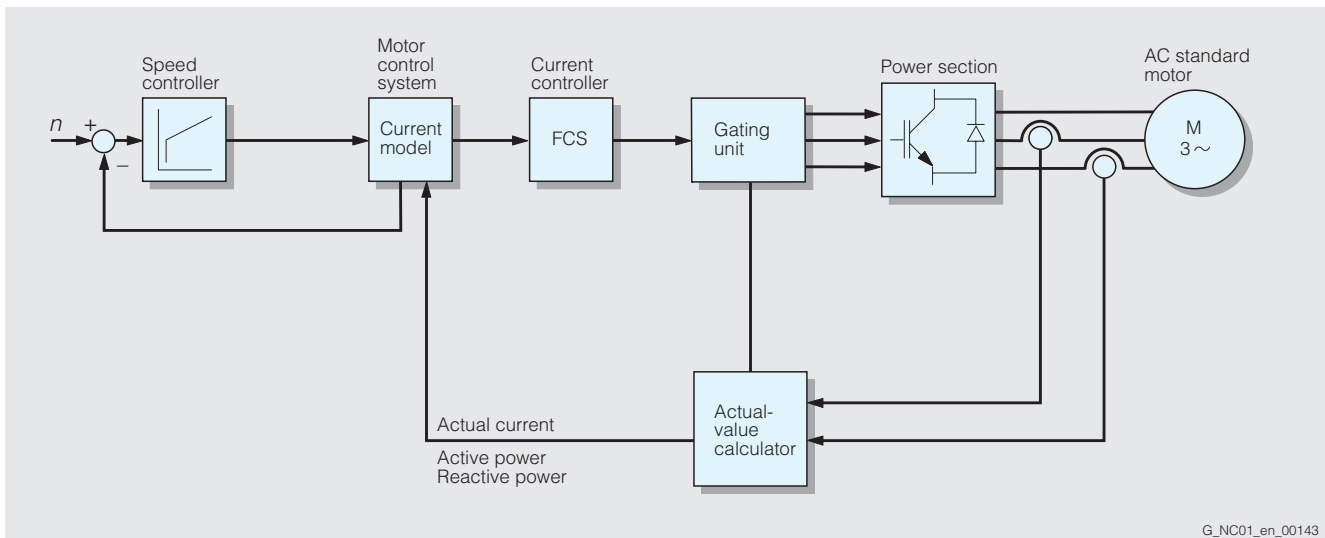
Closed-loop control of the digital feed motors is based on a powerful signal processor with current and speed controls for axis-specific current and speed control. The closed-loop control system is optimally tuned to both the sinusoidal-current commutated 1FK and 1FT6 servomotors and the 1FN linear motors (with performance control).

Glossary

Functions and Terms

SIMODRIVE 611 converter systems

Closed-loop control, induction motor



Closed-loop control for induction motor in analog system

Closed-loop control in analog system

On the analog SIMODRIVE 611 system, closed-loop control for standard induction motors is performed with the help of a micro-processor that executes current and speed control. High control quality is achieved through the use of a field-oriented control algorithm, a controlled-system simulation function based on a motor model, and the derivation of the actual value variables for the control. In the frequency range > 10 Hz, it is possible to implement high-speed and exact speed control without additional encoder systems in addition to a dynamic torque control. The integrated self-start routines simplify the commissioning process.

Closed-loop control in digital system

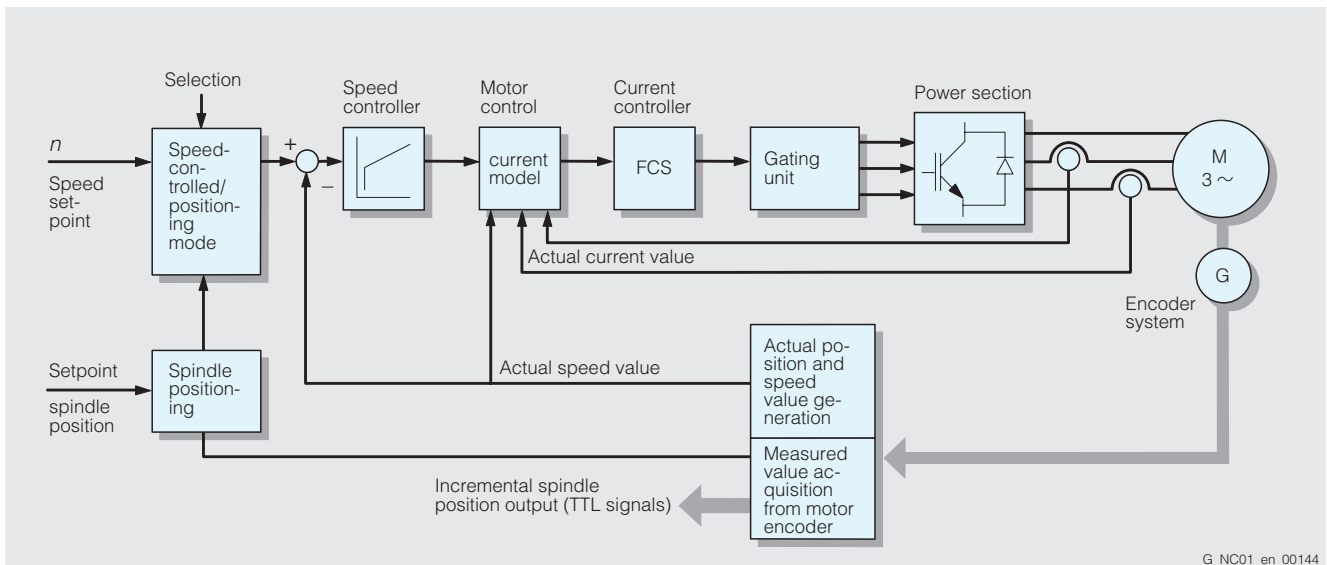
Closed-loop induction motor control is an offshoot of the digital closed-loop control system for main spindle drives. The closed-loop control system for induction motors is based on a powerful signal processor with current and speed controllers for operating encoderless standard induction motors. Excellent control quality is achieved through a field-oriented control algorithm, motor model-based controlled-system simulation, and extraction of actual-values for the closed loop. A speed/torque/frequency feedforward control has been integrated to improve the dynamic performance of the control. The bidirectional data exchange with the SINUMERIK 840D via the drive bus is managed by a communication block.

The start-up tool automatically initializes optimum control settings. When the application requires positioning and high speed capabilities in one drive system, it is possible to operate 1PH motors in mixed main spindle/encoderless induction motor mode on one converter.

Closed-loop control with SIMODRIVE 611 universal

The closed-loop control of the standardized induction motors is based on a powerful signal processor with current and speed controllers for operating encoderless standard induction motors. Excellent control quality is achieved through a field-oriented control algorithm, motor model-based controlled-system simulation, and extraction of actual values for the control loop. A speed/torque/frequency feedforward control has been integrated to improve the dynamic performance of the control.

Closed-loop control, main spindle, analog



G_NC01_en_00144

Closed-loop control for main spindle in analog system

Closed-loop control in analog system

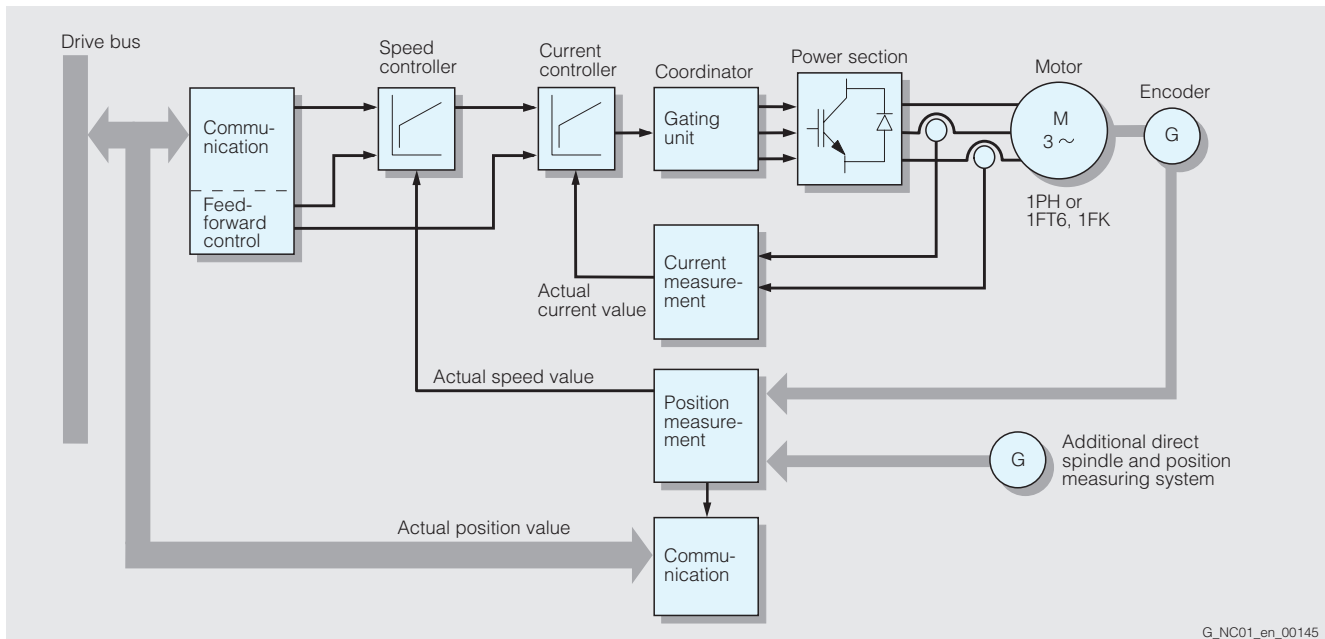
The closed-loop control system is digital and based on a successful microprocessor. It controls the 1PH main spindle modules with sine-cosine encoders. The main spindle drive controller receives the setpoint from a higher-level position controller via a standardized ± 10 V interface.

Both speed-controlled and torque-controlled operation are possible and can be set on externally selectable terminals. A programmable filter can be activated in the torque setpoint channel.

Glossary Functions and Terms

SIMODRIVE 611 converter systems

Closed-loop control, main spindle, digital



Closed-loop control for main spindle or feedrate control in digital system

Closed-loop control in digital system

The closed-loop control system for the digital main spindle modules is based on a powerful signal processor with closed-loop current and speed control of the 1PH spindle motors with sine-cosine encoder. Data exchange with the SINUMERIK 810D/840D is controlled bidirectionally via the drive bus by means of a communication closed-loop control system for induction motor in analog block. The current and speed controls are based on an easy-to-set state controller. With the high control dynamics

and programmable filters for attenuating mechanical resonances, excellent positioning and high C-axis quality are achieved.

Closed-loop control, SIMODRIVE 611 universal

Closed-loop control of the main spindle motors is based on a powerful signal processor with current-speed controller for operating 1PH the main spindle motors with sine-cosine encoder.

CNC link

Analog system

Closed-loop control plug-in modules with provenly reliable analog setpoint interface are available with the SIMODRIVE 611 system to provide a link to the SINUMERIK 840C/802C or other controls.

Digital system

Closed-loop control plug-in modules with digital drive bus interface are available with the SIMODRIVE 611 system to provide a digital link to the SINUMERIK 810D/840D/840C.

Connections

A practical wiring technique increases electromagnetic compatibility and saves time on installation and wiring. To ensure that shielded cables are effectively contacted, pre-assembled shield contacting points are provided on the modules and/or in the system. As signal and power leads must usually be connected separately when configuring a control cabinet, the connection points on the SIMODRIVE 611 are located as follows:

- Power connectors or power terminals on the underside of the module, with cables being led downwards
- DC link connection on the module front plate
- Unit bus and drive bus connections on the module front plate
- Measuring system connections and interface connections to machine control on the module front plate, the cable being led upwards.

The SIMODRIVE 611 has power connectors for connecting cables for lower power ratings and power terminals for cables for higher ratings.

The interface connections in the form of clamp-type terminals can be optionally coded.

Coded plug-in connections are provided on the front of the modules to allow additional external wiring of signal leads. The signal leads for the motor measuring system and the direct position measuring system are connected to the front of the module using subminiature D connectors.

A shielded, easy-to-handle drive bus cable provides the communication between the SINUMERIK 810D/840D/840C and the drive modules of the digital SIMODRIVE 611 system.

On the SIMODRIVE 611 universal a communication link via PROFIBUS-DP and the cyclic motion control system with PROFIBUS-DP is possible. It is thus possible to use existing signal interfaces for solving drive tasks as well as positioning tasks, and for applications involving cyclic interpolation. This connection method, which is both captive and non-interchangeable, thus facilitates installation, start-up and servicing.

G_NC01_en_00145

Cooling

Depending on the application, module heat loss can be transferred to the environment in various ways. Three different heat removal systems are available. They differ primarily in the design of the module rear panel and the arrangement of the separate fan. The front panel with its connections and the module width are unaffected by any of the heat removal systems.

Internal cooling

In this standard solution, the lost heat from the electronics and power sections of the converter components is removed by natural cooling or by a forced-ventilation system and routed to the interior of the control cabinet.

Hose cooling

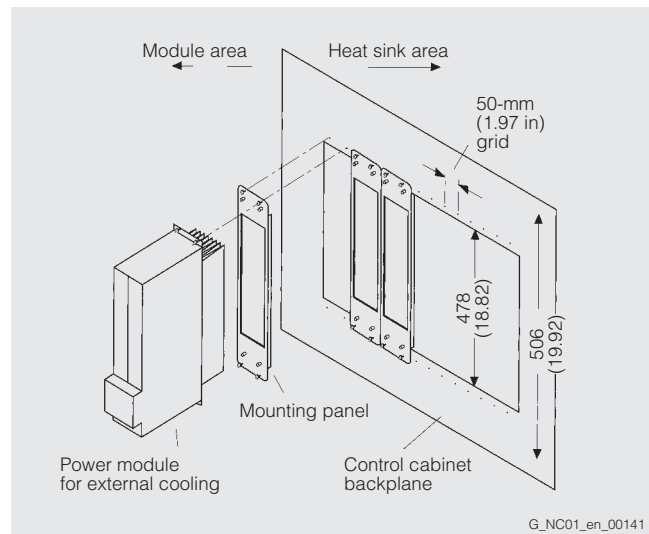
The "hose cooling" option is directed at control cabinets with no separate ventilation channel for sizeable heat losses. Using flexible hoses, the dissipated heat from the specially prepared 300 mm (11.81 in) modules is routed directly to the air surrounding the cabinet by a special fan unit.

In this drive combination, these modules are combined with smaller, internally cooled modules. IP 54 protection can be attained at the "mechanical interface" to the outside air in hose-cooled systems.

External cooling

The modules' power section heat sinks pass through the mounting tier of the control cabinet and can thus release the heat losses of the power section to a separate external cooling circuit. The only heat losses that remain in the cabinet are those emitted by the electronics. Degree of protection IP54 is achievable at the "mechanical interface", the external heat sink.

The module-related heat sink cutouts can be inserted directly in the mounting tier or rear panel of the control cabinet in accordance with the drawings shown in the planning instructions or,



alternatively, they can be integrated in the control cabinet with the mounting frame specially designed for the modules and available as an accessory.

With the mounting frame adapted to the module in question, the heat sink cutouts can be mounted modularly in one large mounting panel cutout at 50-mm (1.97 in) intervals. On modules with a width ≤ 200 mm (7.87 in), the separate fan and the air conduction panel are included with the module. For modules with a width of 300 mm (11.81 in), the available fan boxes and separate fans must be ordered separately.

Current/frequency control

On main spindle drives with induction motors, current/frequency control during e.g. the start-up phase without motor encoder can

be useful. On induction motors modules, only this mode is used in the lower frequency range (< 5 Hz).

Device bus

The device bus is a ribbon cable that leads via plug-in connectors from one module to the next. It carries the internal power from the central electronics power supply to the individual feed

and main spindle modules and the special function modules. The main supply or monitoring modules also exchange fault and monitoring data with the other modules over the device bus.

Digital filter

Filter functions can be programmed in the torque setpoint channel of the digital drives and main spindle drives with analog setpoint interface. Filters with bandstop or low-pass characteristic

offer settings for blocking frequency, transition frequency, and filter quality. These filters dampen oscillations produced in the control loop or by the transmission elements.

Direct line connection

With the use of high blocking-capability power transistors, the infeed/regenerative feedback module can be connected directly via a commutating reactor or a filter module and commutating reactor in series connection without the use of a trans-

former. The standard setting is made using a simple coding switch and setting it to either direct regulated operation or regulated operation with regenerative feedback.

Electronics power supply

The power supply to the electronics of the feed and main spindle modules is provided centrally for the drive group by the infeed modules, and is designed for the connection of several power modules. In the SINUMERIK 810D/840D and SIMODRIVE 611 digital system, the SINUMERIK 810D/840D is also provided cen-

trally by the infeed modules. This means that a uniform concept can be devised, for example, to deal with power failures. An additional power supply in the form of a monitoring module can be used for larger drive configurations.

Glossary

Functions and Terms

SIMODRIVE 611 converter systems

Enabling signals

Three enabling signals/terminals are provided for enabling the drives. Terminal 48 "start" has the highest priority and also directly switches the line contactor integrated in the mains supply module. The terminal is also instrumental in determining the ON/OFF sequence. Terminal 48 can also be used to apply the central pulse disabling and enabling signals.

Terminal 63 "pulse enable" has the highest priority for disabling and enabling pulses. The enabling and disabling commands take immediate (not delayed) and simultaneous effect on the power modules of all axes. When the signal is removed, the

drives coast to a standstill with no electrical braking action. The same applies to the axis-specific pulse enabling signal at terminal 663 on the analog and digital closed-loop control plug-in modules.

The controllers of all modules in the drive group are enabled or disabled simultaneously, or for specific axes via the "controller enable" terminals, with the "drive enable" terminal. The enabling signals can also be used to brake the axes, since the speed setpoint can be set internally to zero when these terminals are opened.

Feed as main spindle

Analog system

Main spindle functions are also available in the analog feed control system (with enhanced interface and main spindle option). An AC servomotor of the 1FT5 series can thus be used as a main spindle motor.

The function expansion includes:

- Ramp function generator from 10 ms to 10 s
- Variable limit value increments
- Speed-dependent current limit (quasi-constant output)
- C-axis mode, switchover from main spindle to feed mode possible on the fly
- Output for display of the actual power (default) or current value (switchable)
- Output of the actual speed value.

Digital system

In conjunction with the digital drive controls, 1FT6/1FK servomotors can also be used as main spindle motors with certain restrictions (e.g. as a compact spindle drive or as a rotating tool with spindle functionality). The main spindle functions required for operation with the SINUMERIK 810D/840D are integrated in the drive control as standard.

Galvanic isolation

The drive group can be isolated from the supply by means of a mains contactor integrated in the infeed module.

Potential energy can, however, remain stored for a short period of time in the DC link.

Gantry operation

In "gantry operation" mode, two coupled machine axes are traversed synchronously via mutually independent axis drives.

Only one axis is programmed, but both gantry axes traverse as if they were one. Several gantry axis pairs can be generated.

Gear stages

The control parameters can be adjusted in the main spindle axes on the basis of switchover between up to 8 gear stage, thus allowing setpoint scaling, torque limitation, speed and torque monitoring and controlled system parameters to be adapted to

→ Parameter set switchover, Oscillation

the properties of the relevant gear stage. Adjustable oscillation setpoints can also be input to achieve optimum gear engagement.

HPC axis

The HPC axis (high-precision C-axis) improves the performance under load and response to setpoint changes, particularly when the main spindle is operating in feed mode. The speed controller sampling time is significantly reduced when the functions that are not required for this operating mode, such as ramp-function generator, gear stage switchover, spindle positioning, etc., are deselected, thereby achieving a better dynamic response from the speed control loop and finer setpoint scaling.

In the case of start-/delta motors, the HPC axis can only be selected in star mode.

Switchover of the star motor data block can be selected and deselected online by means of a selector terminal. A flux adjustment can be made to reduce magnetic motor noise, especially for HPC mode.

Increment multiplication

Multiplication factors are provided on the drive side to allow actual values supplied by the measuring systems to be processed with greater precision in the CNC. A multiplication factor of 4 is available for analog drives.

A maximum factor of 2048 ("performance" control version) is available for the digital drives, providing the CNC with a pulse number of over 4,000,000 pulses/revolution for motor encoders with 2048 marks/revolution ("performance" control).

Integrator disable

Shorter positioning times can be achieved by switching off or deleting the integral component when moving into the material from higher speeds.

I/O

Analog and/or digital inputs and outputs are integrated in the SIMODRIVE 611 universal systems in order to make it possible to pick up and process signals from the process.

Mains buffering

The potential and kinetic energy currently stored in the DC link or in the mechanics can be used to maintain the power supply

to the electronics. The duration of any possible mains buffering depends on the amount of energy currently available.

Mechanical couplings

Analog system

A power or torque increase in main spindle or feed drives can be achieved by a mechanical coupling of two or more drive motors operating in parallel. For "torque division" mode, a rigid mechanical coupling of the drives is required. One of the drive modules can be initialized to be the master and the other the slave

module. The master module supplies the torque setpoint to be used by the slave drive(s) via a terminal. The slave function can, if so configured, be made selectable via the terminals. A slip monitoring function for the slave drive can also be activated for main spindle modules.

Messages and alarms

Analog system

With main spindle and induction motor controls, the messages are output to the control unit's six-character display. Every status and diagnostic message can also be output via floating terminals (relay contacts) and post-processed externally. Axis-specific operational and fault messages are output on feed modules via the standard interface with LEDs or via the enhanced interface with 7-segment display.

The current status of the starting lockout relay is provided via the positively driven signal outputs according to module for evaluation in the control interlocks of the machine.

Digital system

Messages and alarms are output to the display on the SINUMERIK 810D/840D. Additional support is provided by drive-specific service graphics in which all drive-related data can be visualized, e.g.:

- Enables
- Ramp-up phases
- Monitoring functions
- Motor temperature and heat sink overtemperature
- Terminal status of the starting lockout

Every status message and diagnostic message from the drives can be post-processed by the SINUMERIK 810D/840D's PLC. The PLC can use a comprehensive set of status messages in an axis-specific data block. The PLC can control the drive. External wiring is minimized by the digital drive bus between the SIMODRIVE 611 and the SINUMERIK 840D.

On SIMODRIVE 611 universal PROFIBUS can be used for diagnostics. Diagnostics are supported by the SimoCom U software tool.

Modular design

Drive configurations with almost any number of axes and main spindles are made possible by the modular design of the SIMODRIVE 611 converter system. The DC-link power and motor size required determine which central modules for the

mains supply infeed are needed. The SINUMERIK 810D/840D modules have been designed so that they can be smoothly integrated into the existing module network.

Monitoring of critical parameters

During operation, parameters and states can be monitored on the drive modules for violations of limit values. Parameters and states are e.g.:

- Motor temperature
- Heat sink temperature
- Speed, torque and current limits
- Values defined by the user.

Analog system

On the drive modules in the analog system, the signals are available at terminals.

Digital system

In the digital system, limit violations are reported directly to the SINUMERIK 810D/840D, and can be post-processed there, for instance in the PLC.

The signals must be processed in the machine interface controller.

Glossary

Functions and Terms

SIMODRIVE 611 converter systems

Motion Control with PROFIBUS-DP

→ SINUMERIK 810D/840Di/840D

Oriented spindle stop (NC function M19)

Analog system Main spindle drives
Oriented spindle stop is an auxiliary CNC function with which a finer speed setpoint scale is set when the speed drops below a

programmable limit in the selected operating mode. Apart from deactivation of the speed controller, a setpoint rounding or smoothing effect and an offset correction can be programmed.

Oscillation

→ Gear stages

Oscillating setpoint injections with programmable amplitude are output when the "oscillation" function is activated.

The function supports gear changes that protect the gear teeth.

Parameter set switchover

→ Gear stages

Various setting values for the control with up to 8 parameter sets are available for SIMODRIVE 611 main spindle controls, asynchronous motor modules, and for digital feed controls, such as:

- Speed control parameters
- Filter settings
- Messages and monitoring functions
- Current and power limit

In the case of the main spindle controls of the analog system, the parameter sets can be selected externally via assignable-function terminals. In the digital drive system, the parameter sets can be switched via the SINUMERIK 810D/840D/840C. This makes it easy to adapt the control to changing conditions during machining, e.g.:

- Workpieces with different-dimensions
- Changing a spindle head
- Switching between different monitoring limits in dependence on machining

Parking axis/spindle

The "parking axis/spindle" function allows an axis or spindle to be immobilized so that a motor or encoder can be replaced without disconnecting the power.

Position decoding, direct

Suitable measuring systems:

- Rotary encoders with TTL signals (on analog MSD modules only)
- Rotary encoders with sine/cosine-shaped voltage signals
- Linear scales with sine/cosine-shaped voltage signals
- Distance-coded measuring systems (only SIMODRIVE 611 digital with CNC)
- Measuring systems with sine/cosine-shaped voltage signals and EnDat interface (linear scales, single-turn and multi-turn encoders)
- Measuring systems with SSI

The analog main spindle drive modules and the digital feed and main spindle drive modules can be supplied with a second measuring system evaluation, e.g. for a table-top measuring system or for spindle position decoding.

A direct measuring system is needed, for example, when a high degree of accuracy has to be achieved on the workpiece with a linear scale or exact positioning is required with a multi-stage gear.

Main spindle drive module, analog system

For direct spindle position decoding, an optional, additional position measuring system with TTL signals can be connected to the main spindle control, or the spindle signals can be output for external applications.

SIMODRIVE 611 digital/universal

The optional measuring system for position decoding is suitable for the evaluation of incremental encoders with sine/cosine-shaped voltage signals. It is possible to connect linear scales and rotary encoders with sinusoidal voltage signals and distance-coded measuring systems to drive controls for operating 1FT6 and 1FK feed motors.

Position decoding, indirect

Analog system

The modules are equipped as standard with the connection for the measuring system integrated in the feed and main spindle motors.

SIMODRIVE 611 digital/universal

When the SINUMERIK 810D/840D/840C and SIMODRIVE 611 are digitally linked, the measuring systems are connected to the digital closed-loop control plug-in modules. The modules are equipped with the connection for the measuring system integrated in the feed and main spindle modules as standard.

Together with the high-resolution position decoding of the digital control modules, the integrated motor measuring system achieves a resolution of 4,000,000 increments per revolution (performance control). This makes an additional C-axis encoder unnecessary, even on the main spindle. The high-resolution actual position value can also be transferred to the CNC position control loops via the drive bus so that, given the right mechanical conditions, a table-top measuring system is no longer required. The same marginal conditions apply to SIMODRIVE 611 universal. The one difference is the drive link, which is established via PROFIBUS.

Positioning functionality

On the SIMODRIVE 611 universal it is possible to store and execute traversing blocks automatically with the aid of the inte-

grated positioning functionality. As many as 64 traversing blocks can be programmed for each axis.

Positioning to external zero mark

With SIMODRIVE 611 main spindle controls and digital feed drives, it is possible, in spindle positioning mode, to evaluate an external zero mark instead of the encoder zero mark.

Applications:

- One or several gear stages between the motor and the spindle
- No high precision requirements (e.g. M19 with mechanical indexing). In this case, it is possible to do without a spindle measuring system.

PROFIBUS-DP

→ SINUMERIK 810D/840Di/840D

Ramp-function generator, programmable

The ramp-up and ramp-down time ramps can be set independently of one another. The ramp-function generator can also be bypassed by means of a selector terminal (ramp-up time = zero). Separate ramps can be set for each motor data block. A ramp-function generator compensation function that adjusts the

ramp-up time to suit the acceleration characteristics of the drive (even when excessive ramp gradients have been set) can be initialized for the analog main spindle control. Two ramp-function generator settings can be initialized for each motor data block on the analog induction motor control.

Setpoint smoothing filter

The current and speed setpoint filters (low-pass or bandstop) can be used to dampen drive resonance. The controller parameters can therefore be optimally adjusted to the mechanical

characteristics of the machine, allowing the maximum dynamic capability of the control/drive system to be utilized.

Setup mode for feed axe

Analog system

The setup mode is possible by equipping the feed axes with 1FT5 motors and analog closed-loop control plug-in modules in the enhanced version, and must be used in the feed drive axes with self-locking mechanical transmission elements.

The DC-link rated voltage of 600 V must be reduced for setup mode to a safe value for the particular application via an externally supplied matching transformer with separate windings.

The SIMODRIVE 611 converter system permits blocking of the DC-link voltage control on deselection of the "setup mode" func-

tion on the power supply module, i.e. a DC-link voltage proportional to the supply voltage is provided. The system can be operated on a DC-link voltage of ≥ 34 V DC (depending on the motors).

Power feedback is deselected in this mode. Brake energy from the feed axes must not result in any hazardous increase in the DC-link voltage.

At the same time, the current limits are reduced for the drive axes and the monitoring function "speed controller at stop" is not displayed.

Slip monitor

Analog system Main spindle modules

If the drives are not coupled rigidly but friction-locked in master/slave operation, slipping may occur in the torque-controlled slave drive.

The slip monitor checks the deviation between the set speed and the actual speed in the slave drive and reduces the drive torque when the difference exceeds a programmable, standardized speed tolerance limit.

It is possible to initialize the torque reduction gradient that is applied when the monitor is activated and the rate of torque rise after delayed deactivation of the monitor for the application in question.

Glossary

Functions and Terms

SIMODRIVE 611 converter systems

Spindle positioning

Analog system

In the analog drive system, the "spindle positioning without CNC" function is integrated in the main spindle software and can be triggered via terminal.

Digital system

In the digital drive system, the spindle positioning function can only be triggered by the SINUMERIK 810D/840D.

Spindle power display

In digital systems, the spindle power is displayed directly on the monitor of the SINUMERIK 810D/840D/840C control in the form of a bar chart.

A terminal with proportional voltage signal (± 10 V) is available for displaying the current spindle power in analog systems.

Standard start-up

→ Start-up support

Analog system

In the case of feed modules with a standard interface, the axis-specific parameters are directly set on the front panel of the module or on the control module. For operation with the enhanced interface, a setting module is required for the axis-specific parameter settings. This module uses analog technology with passive components. All data specific to the machine and drive system can be stored on it. A slot is available in the

module for the setting module, which can be slotted in from outside.

Main spindle and induction motor modules are started up using a key panel with 6-character display that is integrated in the module. By entering the motor code or the data from the rating plate of the motor, the optimum parameters are automatically calculated. For easy start-up of the main spindle and induction motor modules, a PG/PC-based support tool is available.

Starting lockout

A starting lockout is integrated in every drive module in order to prevent accidental restart after the drives have stopped. The starting lockout blocks the power transistors with a circuit authorized by the German Employers' Liability Insurance association. The circuit interrupts the energy supply to the motor (in accordance with IEC 60204). The lockout circuit can be combined with a suitable external locking circuit to implement a machine control that complies with EN 954, category 1 or 3, or even cat-

egory 4 subject to certain conditions. The starting lockout can be used to provide the "safe standstill" safety function.

One starting lockout circuit is provided for each drive module on analog and digital closed-loop control modules.

One signalling contact is provided for the 3 integrated drives and for the 3 optional axes expansions on the SINUMERIK 810D.

Start-up support

Analog system

Start-up software for PCs/programming devices is available to facilitate start-up of the main spindle and induction modules. The software provides menu-assisted support for starting up and optimizing the drives. An optimum controller setting is calculated for the induction modules after the basic electrical data for the motor have been entered.

Digital system

To assure fast and easy start-up of the digital SIMODRIVE 611 drives and optimization of the control loops, a start-up software package is available on the PCU 50/MMC 103. The start-up software permits drive configurations to be initialized quickly and reliably on the basis of order numbers. Standard data blocks that are specially adjusted to the drives are loaded to suit the selected configuration. The set drive and control parameters are stored as data blocks in the SINUMERIK and are automatically available on every system restart. The data blocks can be duplicated via the PC/PG for identical machines.

Additional tools are available for optimization and diagnostics:

- Parameterizable function generator for traversing drives without CNC part program

User-initializable DACs (digital/analog converters) on the digital control modules for the output of any selected display and diagnostic values

- Time-range measuring functions for optimizing current, speed and position controllers
- Frequency-range measuring functions for optimizing the entire controlled system and analyzing mechanical properties (resonance).

SimoCom U

The SimoCom U software provides for fast, easy start-up of the SIMODRIVE 611 universal distributed drive system as well as for optimizing the drive parameters to the relevant application. This software makes the generation, initialization and optimization of drive configurations easy and reliable.

Storage of customer-specific setting data

Analog system

In the case of feed controls with enhanced interface, all customer-specific data are stored on a setting module.

On main spindle and induction motor modules, it is possible to save the drive and machine data in non-volatile storage on the closed-loop control hardware.

The data can be stored on an external PC via the RS 232C interface.

Digital system

The closed-loop control software and the customer-specific data are stored centrally in the SINUMERIK 810D/840D/840C and loaded to the drive module designed for feed and main spindle applications during system power-up. This ensures centralized management and simple software updating.

SIMODRIVE 611 universal

The control software and the customer-specific setting parameters are stored locally on a memory card in each control. When servicing is required, devices can simply be replaced, all setting data remain on the memory card, and the drive is immediately ready for operation.

Synchronous spindle

The "synchronous spindle" function allows a workpiece to be transferred on the fly from the workspindle to the speed-

synchronized and/or position-synchronized spindle. Workpieces can thus be machined without interruption.

Torque feedforward control

The torque feedforward control function can be applied to reduce contour deviations resulting from following errors. The function can be selected and deselected in the part program.

Travel to fixed stop

The drive can be programmed to approach a mechanical stop on the machine with a defined motor torque in speed-controlled mode. A contact force for the fixed stop selected specially for the machining process concerned can be input as an infinitely

variable quantity via a limiting input for the torque or current setpoint. The controller monitor is deactivated when this function is selected.

V/Hz operation

Digital system

Encoderless induction motors can be operated on the digital SIMODRIVE 611 universal in the V/Hz mode. V/Hz mode is primarily intended for simple applications, i.e. with minimum

start-up requirements and parallel connection of several induction motors on one inverter module, as well as for diagnostic purposes.

Weight compensation, electronic

The electronic weight compensation function makes it possible to fix a vertical axis without a mechanical counter-weight. An additional torque setpoint is injected to start the compensatory control immediately after the axis is enabled, thus ensuring that it is held in its position.

Glossary

Functions and Terms

SIMODRIVE POSMO converter systems

Closed-loop control, feed, digital

SIMODRIVE POSMO CD/CA

Closed-loop control of the digital feed motors is based on a powerful signal processor with current and speed controls for axis-specific current and speed control. The closed-loop control

system is optimally tuned to both the sinusoidal-current commutated 1FK and 1FT servomotors and the 1FN linear motors.

Closed-loop control, induction motor

SIMODRIVE POSMO CD/CA

The closed-loop control of the standardized induction motors is based on a powerful signal processor with current and speed controllers for operating encoderless standard induction motors. Excellent control quality is achieved through a field-oriented

control algorithm, motor model-based controlled-system simulation, and extraction of actual values for the control loop. A speed/torque/frequency feedforward control has been integrated to improve the dynamic performance of the control.

Closed-loop control, main spindle, digital

SIMODRIVE POSMO CD/CA

Closed-loop control of the digital main spindle motors is based on a powerful signal processor with current-speed controller for

operating 1PH the main spindle motors with sine-cosine encoder.

Cooling

SIMODRIVE POSMO SI

The SIMODRIVE POSMO SI is cooled via a built-on fan.

SIMODRIVE POSMO CD/CA

The SIMODRIVE POSMO CD/CA is self-cooled through the ambient temperature.

Electronics power supply

In SIMODRIVE POSMO distributed drive systems, the power supply for the electronics is generated internally from the supply voltage.

I/O

Analog and/or digital inputs and outputs are integrated in the SIMODRIVE POSMO SI/CD/CA systems in order to make it possible to pick up and process signals from the process.

Messages and alarms

On SIMODRIVE POSMO SI/CD/CA systems, PROFIBUS can be used for diagnostics. Diagnostics are supported by the SimoCom U software tool.

Modular design

With its "all in one" design, distributed drive configurations can be set up locally right on the machines. A distinction is made between two variants:

SIMODRIVE POSMO SI

is a distributed servo drive system with integrated power module, closed-loop control module, 1FK motor, positioning control and program memory

SIMODRIVE POSMO CD/CA

are distributed drive systems with integrated power module, closed-loop control module, positioning control and program memory. These systems can accommodate motors of different types.

Position decoding, direct

Suitable measuring systems:

- Incremental rotary encoders with sine/cosine-shaped voltage signals
- Incremental linear scales with sine/cosine-shaped voltage signals
- Absolute measuring systems with sine/cosine-shaped voltage signals and EnDat interface (linear scales, single-turn and multi-turn encoders)

SIMODRIVE POSMO CD/CA

The optional measuring system for position decoding is suitable for the evaluation of incremental encoders with sine/cosine-shaped voltage signals. It is possible to connect linear scales and rotary encoders with sinusoidal voltage signals and distance-coded measuring systems to drive controls for operating 1FT6 and 1FK feed motors. The measuring signals supplied by the encoder system are evaluated with high resolution.

Position decoding, indirect

Suitable measuring systems:

- Integrated incremental encoder in feed and main spindle motors

- Integrated absolute encoder with EnDat interface in feed motors

Positioning functionality

On the SIMODRIVE POSMO SI/CD/CA systems, it is possible to store and execute traversing blocks automatically with the aid of

the integrated positioning functionality. As many as 64 traversing blocks can be programmed for each axis.

PROFIBUS-DP

→ SINUMERIK 810D/840Di/840D

Standard start-up

→ Start-up support

Start-up support

SimoCom U

The SimoCom U software provides for fast, easy start-up of the SIMODRIVE POSMO SI/CD/CA distributed drive system as well

as for optimizing the drive parameters to the relevant application. This software makes the generation, initialization and optimization of drive configurations easy and reliable.

Storage of customer-specific setting data

SIMODRIVE POSMO CD/CA

The control software and the customer-specific setting parameters are stored locally on a memory card in each control. When servicing is required, devices can simply be replaced, all setting

data remain on the memory card, and the drive is immediately ready for operation.

Glossary

Abbreviations

A

A	outputs
AC	adaptive control
AC	alternating current
ACOP	advanced coprocessor
A/D	analog/digital
AH	shaft height
ANA	axis, analog
AM	asynchronous motor drive
AP	actuation point
AS	drive end of motor
ASM	interface module
ASM	asynchronous motor
ASUP	asynchronous subroutines
AUTO	automatic mode

B

BAG	mode group
BHG	handheld unit
BKS	basic coordinate system
BNS	basic zero system
BS	non drive end of motor
BTR	behind tape reader

C

CC	central controller
CCU	compact control unit
CDON	collision detection ON
CNC	computerized numerical control
CP	communications processor
CPU	central processing unit
CSB	central service board

D

DAC	digital-analog converter
DB	data block
DC	direct current
DC-PMM	DC power management module
DDE	dynamic data exchange
DMP	distributed machine I/O devices
DMS	direct measuring system
DNC	direct numerical control
DPR	dual-port RAM
DRF	differential resolver function (handwheel shift)
DRY	dry run feed
DSP	digital signal processor

E

ECU	economic control unit
EFP	single I/O module
EG	electronic gear unit
EMC	electromagnetic compatibility
EnDat	encoder data interface
EP	electronic point
ESR	extended stop and retract
EU	expansion unit
EXE	external pulse shaper electronics

F

FB	function block
FC	function call
FCS	frequency compensated spindle drive
FDD	feed drive
FFS	flash file system
FFT	fast Fourier transformation
FIFO	first in first out
FM	function module
FOC	force control
Frame	description of transformations in Cartesian space
FRC	feed (non-modal) for chamfer/rounding
FRCM	feed (modal) for chamfer/rounding
FUP	Function overview

G

GUD	global user data
-----	------------------

H

HLA module	hydraulic linear drive module
HMI	human machine interface
HPC axis	high precision C axis
HSC	high speed cutting
HT	handheld terminal

I

I	input
I/O	inputs/outputs
I/RF	controlled infeed/regenerative module
IBS	start-up
IM	interface module
IPO cycle	interpolation cycle
ISA	industrial standard architecture

J

JOG	jogging
-----	---------

K

KA	K statements (1 KA = 1024 statements)
----	---------------------------------------

L

LAD	ladder diagram
LCD	liquid crystal display
LEC	leadscrew error compensation
LMS	linear measurement system
LMS	linear scale
LR cycle	position controller cycle
LUD	local user data

M

M	milling
MCI	motion control interface
MCS	machine coordinate system
MCP	machine control panel
MDA	manual data automatic
MDI	manual data input
MDS	mobile data storage
MMC	human machine communication
MPC	multi point controller
MPI	multi point interface
MSD	main spindle drive
MSEC	measuring system error compensation

N

NC	numerical control
NCK	numerical control kernel
NCU	numerical control unit
NE	mains supply
NURBS	non uniform rational B-splines (universal interpolator)

O

OB	organization block
OEM	original equipment manufacturer
OI	open-loop control infeed module
OP	operator panel
OPI	operator panel interface

P

PCIN	data transmission program
PCMCIA	personal computer memory card international association
PCU	panel control unit
PG	programming device
PI services	program invocation services
PIT	parameterization and commissioning tool
PLC	programmable logic controller
PLG	planetary gearing
PPU-MF	protected power unit multifunctional
PRT	program test
PS	power section
PTP	point to point
PUD	program global user data

Q

QDS	quality data save
-----	-------------------

R

Repos	reposition
RISC	reduced instruction set computer
RLG	shaft position encoder
ROD	rotary incremental encoder system
ROV	rapid override
RTCP	remote tool center point

S

SBH	safe operational stop
SBL	single block
SCL	structured control language
SE	safe software limit switch
SFB	system function block
SFC	system functions
SG	worm gearing
SG	safe reduced speed
SGA	safety-relevant output signals
SGE	safety-relevant input signals
SH	safe stop
SKP	single block
SLG	read/write device
SM	signal module
SN	safe software cams
SPL	safe programmable logic
SPOS	position spindle
S/R	signals/revolution
SSI	serial synchronous interface
STL	statement list
STN	super twisted nematic
SVE	signal amplifier electronics
SYNACT	synchronized action
SZS	settable zero system

T

T	tool
T	turning
TFT	thin film transistor
TG	tachogenerator
TPM	total productive maintenance
TRANSMIT	transform milling into turning
TTL	transistor transistor logic

V

VDEW	Verband Deutscher Elektrizitätswerke
VDI interface	Verein Deutscher Ingenieure (signal interface (NCK-PLC) agreed between control manufacturers and machine manufacturers)
VGA	video graphic adapter
VPM	voltage protection module

W

WKS	workpiece coordinate system
WOP	graphic programming
WSG	shaft-angle encoder interface
WYSIWYG	what you see is what you get
WZEG	tool setting station
WZV	tool management

Glossary

Abbreviations

Notes

13

