

1D/3D clearance control in position control cycle with free direction, Run MyCC /CLC-FD

Option; order code M65

Article No.: 6FC5800-0AM65-0YB0

Available as a supplement to the function SINUMERIK Integrate Run MyCC /CLC, the function SINUMERIK Integrate Run MyCC /CLC-FD can be used to specify the spatial direction of the controlled motion with 3 simulated axes. It is then possible to program the control direction independently of the tool orientation (laser beam orientation) specified by the 5-axis transformation.

1D/3D clearance control in position control cycle, Run MyCC /CLC

Option; order code M40

Article No.: 6FC5800-0AM40-0YB0

1D/3D clearance control in the position control cycle Run MyCC /CLC controls one machine axis (or 3 machine axes in conjunction with 5-axis transformation) plus a maximum of one gantry axis and makeSiemens NC 62 · 2017s it possible to automatically maintain the constant clearance between the head and the workpiece that is technologically required for the machining process. The clearance information is read in via a high-speed analog input.

The most important application for this option is laser cutting of flat or spatially formed metal sheets or tubes. The function can also be used with a force sensor in order to implement force control (friction welding).

Restricted functionality of export versions:

Clearance control can be applied to only one axis because 5-axis transformation is not available. The number of interpolating axes is restricted to 4 without Run MyCC /CLC and to 3 with Run MyCC /CLC.

1D clearance control in the IPO cycle

1D clearance control in the IPO cycle can be used, for example, to evaluate sensor signals via a high-speed analog input. 1D clearance control in the IPO cycle can also be used to compute a position offset \$AA_OFF for an axis via a synchronized action.

2D representation of 3D protection areas/working areas

You can use protection areas to protect various elements at the machine, their components and the workpiece against incorrect movements. The 3-dimensionally programmed protection areas are displayed in 2D. This display also applies to the programmed working area limitations.

→ [Working area limitation](#)

→ [Protection areas 2D/3D](#)

3D simulation 1 (finished part)

Option; order code P25

Article No.: 6FC5800-0AP25-0YB0

The simulation can be extended to 3D representation by means of the 3D simulation 1 (finished part) option. This extension also applies to the simultaneous recording.

Glossary

Functions and terms

SINUMERIK 840D sI

Acceleration with jerk limitation

To achieve an optimum acceleration pattern with reduced wear on the machine's mechanical parts, you can select SOFT in the part program to ensure a continuous, jerk-free acceleration profile. When you select acceleration with jerk limitation, the speed characteristic over the path is generated as a bell-shaped curve.

Access MyMachine /Ethernet

Article No.: 6FC5864-4AP41-0YB0

SINUMERIK Integrate Access MyMachine /Ethernet enables worldwide, secure remote operation and monitoring of a machine tool automated with SINUMERIK. Remote access is always established via a secure connection based on TLS (previously SSL) via the Internet (worldwide). The technical implementation of access to the Internet depends on the local conditions at the machine. Access to the Internet can be implemented in different ways.

Access MyMachine /P2P

Option; order code P30

Article No.: 6FC5800-0AP30-0YB0

The function SINUMERIK Integrate Access MyMachine /P2P permits remote access to the SINUMERIK HMI to quickly diagnose the machine's condition. It supports data uploads and downloads, analog and ISDN telephone links as well as access via the Internet.

Access MyMachine /P2P increases machine availability as it can quickly go online. Furthermore, it enables users to prepare essential service callouts more effectively.

Access protection

Protection level	Type	PLC DB10 DBB 56 bit...	User
1	Password	–	Machine manufacturer: Development
2	Password	–	Machine manufacturer: Commissioning engineer
3	Password	–	End user: Service
4	Red key switch position 3	7	End user: Programmer, machine setter
5	Green key switch position 2	6	End user: Qualified operator who does not program
6	Black key switch position 1	5	End user: Trained operator who does not program
7	Switch position 0	4	End user: Semi-skilled operator

Access to programs, data and functions is protected in a user-oriented hierarchical system of 7 access levels.

- 3 password levels (protection levels 1 to 3) for 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 CNCs thus have a multistage concept for controlling access rights.

Protection level 1 has the highest access rights and protection level 7 the lowest. A higher protection level automatically includes all protection levels below it.

Access rights for protection levels 1 to 3 are preprogrammed by Siemens (default setting). An entered password has priority over a keyswitch position, and machine manufacturers or end users can change access rights for protection levels 4 through 7. Subprograms can be completely protected against unauthorized reading and display.

Passwords should be regularly changed from an industrial security perspective.

Action log

The action log records all operator actions and pending alarms for diagnostics purposes.

Advanced Position Control APC

Option; order code M13

Article No.: 6FC5800-0AM13-0YB0

The natural frequency of the machine can have a detrimental effect on the maximum speed of the machine and the surface characteristics. The Advanced Position Control APC function raises the k_v factor, improves the surface and therefore increases productivity without requiring any changes to the mechanical components.

Advanced Surface

Option; order code S07

Article No.: 6FC5800-0AS07-0YB0

The Advanced Surface function is used to optimize the motion control. Accurate contours and perfect surfaces can be achieved even at high machining speeds. With optimized speed management, Advanced Surface delivers better workpiece surfaces with a higher workpiece yield.

Alarms and messages

- Alarms and messages:
All messages and alarms are output separately on the operator panel in plain text with date and time as well as the appropriate symbol for the delete criterion. All alarms are saved in an alarm log that can be configured according to size.
- Alarms and messages in the part program:
Messages can be programmed to provide the user with information about the current machining situation during program execution. Message texts may be up to 124 characters long, and are displayed in 2 lines (2 × 62 characters). The contents of variables can also be displayed in message texts.

Example 1:

N10 G1 F2000 B=33.333

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

Display in message line following traversal of block N10

Rotary table position: 33.333 degrees

Example 2:

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

Display: Check ... X position!

In addition to programming messages, alarms can also be set in a CNC program. An alarm always goes hand in hand with a response from the CNC according to the alarm category. The alarm text must be configured, alarm numbers 65000 to 67999 are available. A description of the responses to the various alarms can be found in the Commissioning Manual.

Example 3:

N100 SETAL (65001) Effect:

Display interlocking CNC start

Deletion: with reset

- Alarms and messages from the PLC:
Machine-specific alarms and messages can be displayed directly from the PLC program in plain text. Messages comprise status messages and error messages. In the case of status messages, the display is immediately deleted when the condition is no longer active, error messages must be acknowledged. User-specific alarm numbers from 40000 to 89999 can be assigned to general, channel-specific, axis-specific and spindle-specific user alarms and messages. The response of the CNC 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 value control

System variable \$A_OUTA(n) enables values from up to 8 analog outputs to be directly entered in the part program. The value specified by the NCK can be changed by the PLC before it is output to the hardware of a SIMATIC DP ET 200 analog module. The hardware outputs are written in the interpolation cycle.

Analyze MyCondition

Article No.: 6FC5684-7AP41-0YB0

Article No.: 6FC5684-7BP41-0YB0

Article No.: 6FC5684-7CP41-0YB0

SINUMERIK Integrate Analyze MyCondition provides test cycles for testing synchronized axes and universal axes and for performing circularity tests; it also offers functionality for continuous data acquisition during production. It also supports the reporting of key parameters regarding the wear of mechatronic components.

A condition-oriented maintenance routine keeps production machines running longer and reduces downtimes and outages.

Animated elements

Using short film sequences, animated elements provide support by allowing the user to look ahead during operation and programming, especially in processes where the motion sequence is the primary consideration.

Glossary

Functions and terms

SINUMERIK 840D sl

Asynchronous subprograms ASUB

An asynchronous subprogram is a CNC program which can be started in response to an external event (e.g. a digital input) or from the PLC. Inputs are allocated to subprograms and activated by programming SETINT.

If the relevant event occurs, the CNC block currently being processed is immediately interrupted. The CNC program can be continued later at the point of interruption. Multiple asynchronous subprograms must be assigned different priorities PRIO so that they can be processed in a certain order. Asynchronous subprograms can be disabled and re-enabled in the CNC program (DISABLE/ENABLE).

→ [Interrupt routines with fast retraction from the contour](#)

Auto Servo Tuning AST

Auto Servo Tuning AST automates the process of adapting parameters to the control equipment, which controls the axes of a CNC machine. The parameters are adapted according to the frequency response measurement of the machine dynamics. One of the benefits of Auto Servo Tuning AST is that it facilitates the measuring process.

The axis control loops are individually optimized according to the target parameters selected by the user for an adaptive strategy. In a second step, the control loop parameter settings are adjusted for axes that are identified as being involved in an interpolation path, with the result that the correct dynamic response is obtained for all axes. This adaptation ensures coordinated movement of all the axes along the interpolation path.

→ [Call Auto Servo Tuning from part program AST call](#)

Auxiliary function output

Auxiliary function output informs the PLC when the part program wants the PLC to handle certain machine switching operations. This is accomplished by transferring the appropriate auxiliary functions and their parameters to the PLC interface. The transferred values and signals must be processed by the PLC user program.

The following functions can be transferred to the PLC:

- Tool selection T
- Tool offset D/DL
- Feedrate F/FA
- Spindle speed S
- H functions
- M functions

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

Axes, coupled motion

When a defined leading axis moves, the coupled-motion axes (following axes) assigned to it travel the traverse 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 of a coupled-axis grouping take place simultaneously with the modal instruction TRAILON. A coupled-axis grouping can consist of any desired combinations of linear and rotary axes. A coupled-motion axis can be assigned up to 2 leading axes (in different coupled-axis groupings). A simulated axis can also be defined as the leading axis, in which case the real axis actually does the traveling, taking into account the coupling factor. Another application for coupled-motion axes is the use of 2 coupled-axis groupings to machine the 2 sides of a workpiece.

Axial coupling in the machine coordinate system MCS

The subfunction "Axis collision protection" is implemented by SINUMERIK Integrate Run MyCC /PROT.

The subfunction "Axial coupling in the machine coordinate system" is implemented by the generic couplings.

→ [Axis collision protection, Run MyCC /PROT](#)

→ [Generic coupling](#)

Axis/spindle

Option; order codes A01 ... A28

Article No.: 6FC5800-0AA00-0YB0

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

Functions:

- POS/SPOS/M3, M4, M5 (from CNC block)
- POSA/SPOSA (from CNC block, modally)
- FC18/POS/SPOS/M3, M4, M5 (PLC axes)
- PLC-VDI interface (M3, M4, M5 directly)
- OSCILL (asynchronous oscillation)
- OSCILL (synchronous oscillation)
- do POS/SPOS/M3, M4, M5 (synchronized actions)
- Couplings (TRAIL, LEAD, EG, CP, ...)
- Path/geometry/additional path axes/GEOAX()
- Spindles for thread cutting, tapping and thread cutting with compensating chuck
- Setpoint output and actual values are available
- Commissioning with SINUMERIK Operate

→ [Positioning axis/auxiliary spindle](#)

→ [Simulation axis/spindle](#)

→ [Virtual axis](#)

→ [Internal/external drives](#)

Axis collision protection, Run MyCC /PROT

Option; order code N06

Article No.: 6FC5800-0AN06-0YB0

SINUMERIK Integrate Run MyCC /PROT supports collision protection of up to 20 axis pairs that, for example, move along a common guide rail and that could collide with each other. The axes concerned can also be active in different channels. The traversing directions of the axes of an axis pair can differ. A maximum spacing can also be monitored.

[Restricted functionality of export versions:](#)

Not possible.

Glossary

Functions and terms

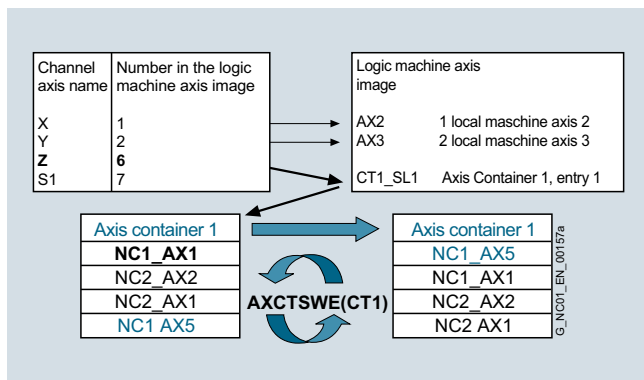
SINUMERIK 840D sl

Axis container

On rotary indexing machines/multi-spindle machines, the axes holding the workpiece move from one machining unit to the next. Since the machining units are handled through 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 available axes that are defined in the axis container can be changed by shifting the entries in the axis container. Shifting can be triggered by the part program or synchronized actions: Keyword AXCTSWE(CT1).



Example of an axis container:

Following rotation of the axis container through 1, the channel axis Z is assigned to axis AX5 on NCU 1 instead of axis AX1.

Axis data output via PROFIBUS, Run MyCC /ADAS

Option; order code N07

Article No.: 6FC5800-0AN07-0YB0

SINUMERIK Integrate Run MyCC /ADAS enables axis and spindle data to be output to a special PROFIBUS or PROFINET device module. This function can be used, for example, for process or machine monitoring functions in real time outside the CNC.

The required axis and signal type are selected by transferring a selection command (length: 8 bytes) to the CNC. SINUMERIK Integrate Run MyCC /ADAS in the NC kernel then sends up to 30 axis data items (of 4 bytes each) to the slave in each PROFIBUS cycle.

The transfer cycle can be equal to the position controller cycle, or a multiple thereof. Consistent receipt of data from each cycle can be guaranteed only when the slave is operating in isochronous mode (or with PROFINET IRT).

Axis limitation from the PLC

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 user program via the PLC interface. As a result, the relevant protection area is activated, for example, before a tool probe is swiveled into position in the working area, to see whether the tool or a workpiece is in the path of the swiveling part.

The PLC can put another axis limitation into effect by activating the 2nd software limit switch via a PLC interface signal. This reduction of the working area may become necessary, for example, when a tailstock is swiveled into position. The change is immediately effective, and the 1st software limit switch plus/minus is no longer valid.

→ [Protection areas 2D/3D](#)

Axis/spindle replacement

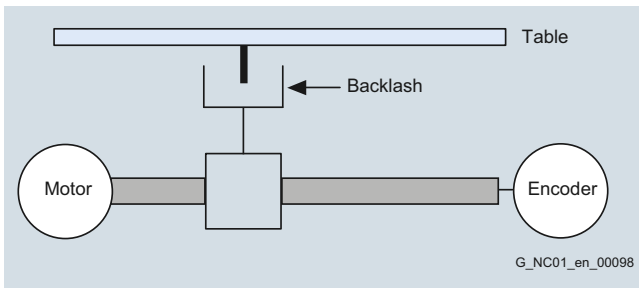
An axis/a spindle is permanently assigned to a specific channel via machine data. The axis/spindle replacement function can be used to release an axis/a spindle (RELEASE) and to assign it to another channel (GET), i.e. to replace the axis/spindle. The relevant axes/spindles are determined via machine 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 machine wear. In the case of axes/spindles with indirect measuring systems, mechanical backlash results in a corruption of the traversing path. For example, when the direction of movement is reversed, an axis will travel too much or too little by the amount of the backlash.

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 2nd 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)

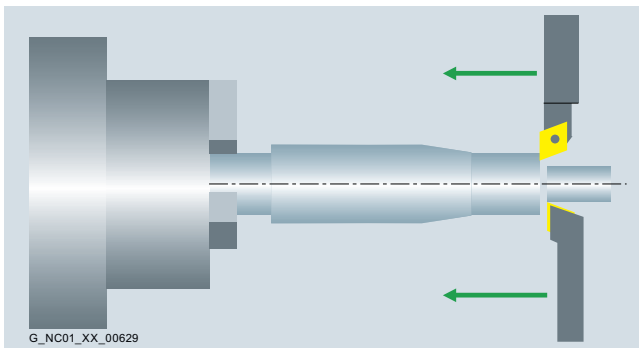
The actual encoder value is ahead of the true actual value (table): The table does not travel far enough.

Balance cutting

Option; order code S05

Article No.: 6FC5800-0AS05-0YB0

The Balance cutting option (rough machining in 4 axes) permits faster machining of workpieces. Simultaneous machining on opposite sides ensures balanced machining forces.



Bidirectional compensation

→ [Leadscrew error compensation, bidirectional](#)

Glossary

Functions and terms

SINUMERIK 840D sl

Block search

The block search function allows any point in the part program to be selected where machining should start or be continued. The function is provided for the purpose of testing part programs or to continue machining after it was canceled. Cascaded block search is also possible.

Search variants:

- With calculation on the contour
During the block search, the same calculations are performed as in normal program operation. The target 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.
- With calculation at block end point
This function allows a target position to be approached (such as tool change position). All calculations are also executed here as during normal program operation. The end point of the target block or the next programmed position is approached based on the interpolation valid in the target block.
- Without calculation
This method enables 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.
- External block search without calculation
In the menus "Search position" and "Search pointer", the softkey "External without calculation" can be used to start an accelerated block search for programs which are executed from an external device (local hard disk or network drive).
- Specify the search target by:
 - Directly positioning the cursor on the target block
 - Specifying a block number, a jump label, any character string, a program name, or a line number

Call Auto Servo Tuning from part program AST call

Option; order code S10

Article No.: 6FC5800-0AS10-0YB0

AST call allows machines to be optimized again automatically from the part program. The AST call function is thus particularly useful for processes involving a wide range of different workpiece weights and clamping operations, and for machines driven by linear/torque motors.

→ [Auto Servo Tuning AST](#)

Cartesian PTP travel

For handling and robot-related tasks, 2 types of movement are required: either in the Cartesian coordinate system Continuous Path CP, or as a Point-to-Point PTP motion. With PTP, the shortest way to reach the end point is with activated TRAORI transformation. PTP generates a linear interpolation in the axis space of the machine axis.

By smoothing from PTP to CP movement, it is possible to switch from fast infeed to a mounting or positioning movement with optimum timing. PTP travel does not result in an axis overload when traveling through a singularity, such as changing an arm position during handling. PTP travel is also possible in the JOG CNC operating mode and does not require Cartesian positions (e.g. from CAD systems) to be converted into machine axis values.

Cartesian PTP travel is also used for cylindrical grinding machines with an inclined axis: With active transformation, the infeed axis can be moved either according to Cartesian coordinates or at the angle of the inclined axis.

Cartesian travel to fixed stop, Run MyCC /FXSC

Option; order code N38

Article No.: 6FC5800-0AN38-0YB0

SINUMERIK Integrate Run MyCC /FXSC cartesian travel to fixed stop is used as a substitute for the standard travel to fixed stop function for machine kinematics with OEM transformation. With this function, tailstocks or sleeves, for example, can be traversed to a fixed stop in order to clamp workpieces. The clamping torque and a fixed stop monitoring window can be programmed in the part program and set in the machine data. The function regulates the force to an adjustable value in the approach direction. In this case, it can be utilized simultaneously in several channels.

Restricted functionality of export versions:
Not possible.

CCG compiler non-circular grinding (cam contour grinding)

Option; order code P10

Article No.: 6FC5800-0AP10-0YB0

Requirements: TRANSMIT option, order code M27 and polynomial interpolation option, order code M18.

This option is required for SINUMERIK 840D sl to allow the execution of part programs that have been generated with the CCG compiler tool. The OEM can either integrate the corresponding tool into the user interface on the PCU 50 or use it on an external PC.

The latest version of the tool is made available once on request, the option (runtime) must be ordered for each CNC. The CCG compiler generates CNC programs in the polynomial format to allow the machining of non-circular contours on a cylindrical grinding machine. The lift curves commonly used today, which describe the desired final contour in polar coordinates, are used as the input data for programming and generating a complete CNC program.

The technology data, i.e. the number of infeed revolutions, allowance, sparking-out revolutions, angle of infeed and velocities, are parameterized and taken into account when the part program is generated. The generated contour is not dependent on the tool (grinding wheel) radius, because radius compensation (G41/G42) is active in the CNC.

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 circular path (actual position in the block before the circle)
- Direction of rotation of circle
- Circle end position (target defined in circular block)
- Circle center

The circle center can be programmed as an absolute value with reference to the current zero point or as an incremental value with reference to the starting point of the circular path. If the opening angle is apparent from the drawing, then it can be directly programmed. In many cases, the dimensions from a drawing are taken so that it is more convenient to program the radius in order to define the circular path. In the case of a circular arc of more than 180 degrees, the radius specification is given a negative sign.

Glossary

Functions and terms

SINUMERIK 840D sI

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: the starting point, the intermediate point and the end point.

Clamping monitoring

Clamping monitoring is one of the many extensive axis monitoring mechanisms implemented in SINUMERIK CNCs. When an axis is to be clamped on completion of the positioning action, you can activate clamping monitoring using the PLC interface signal clamping in progress. This may become necessary because it is possible for the axis to be pushed beyond the standstill tolerance from the position setpoint during the clamping procedure.

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

→ [Position monitoring](#)

→ [Standstill monitoring](#)

CNC high-level language

To meet the various technological demands of modern machine tools, a CNC high-level language has been implemented in SINUMERIK CNCs 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, for example, 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 CNCs give 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 access 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 as required, for example, free selection of axis and spindle address designations.

User variables are either global GUD or local LUD. LUD can also be redefined via machine data to make them into global program user variables (PUD). They are displayed in the Parameters operating area under the user data softkey, where they can also be changed. Global user variables GUD are CNC variables that are set up by the machine manufacturer. They apply in all programs.

Local user variables LUD are available to the user for parameterizing CNC programs. These data can be redefined in every CNC program. These variables make programming more user-friendly and allow the users to integrate their 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 programmed directly, but are addressed via a variable in which their required address is then entered.

Program jumps

The inclusion of program jumps allows extremely flexible control of the machining process. Conditional and unconditional 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 jump destinations. The jump destination can be before or after the exit jump block.

Program coordination in several channels

Program coordination makes it possible to control the time-related execution in parallel operation of several CNC channels using plain text instructions in the part program. Programs can 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 4 basic arithmetic operations, there are also:

- Sine, cosine, tangent
- Arc sine, arc cosine, arc tangent
- 2. Power of 2 (squaring), square root
- Absolute value
- Integer component, round to integer
- Exponential function, natural logarithm
- Offset, rotation, mirroring
- Scale modification

Comparison operations and logic combinations

Comparison operations with variables can be used to formulate jump 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
- Concatenation of strings

The following logic combinations are also available: AND, OR, NOT, EXOR (EXclusive OR). These logic operations can also be performed bit by bit.

Macro techniques

Using macros, single instructions from a programming language can be grouped together to form a complex instruction. This shortened instruction 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 instructions.

Control structures

The CNC normally processes the CNC blocks in the order in which they are programmed. Control structures allow the programmer to define additional alternatives and program loops as well as program jumps.

The commands make structured programming possible, and make the programs much easier to read:

- Choice of 2 alternatives IF-ELSE – ENDIF
- Continuous loop control LOOP
- Counting loop FOR
- Program loop with start condition WHILE
- Program loop with end condition REPEAT

CNC operating modes

3 CNC operating modes can be selected in the Machine operating area:

- JOG
JOG CNC operating mode (jogging) is intended for the manual movement of axes and spindles, as well as for setting up the machine. The set-up functions are reference point approach, repositioning, traveling with the handwheel or in the predefined incremental mode, and redefinition of the CNC zero point (preset/set actual value).
- MDI
In MDI (Manual Data Input) CNC operating mode, it is possible to enter individual program blocks or sequences of blocks for immediate execution via CNC Start. These blocks can then be saved in part programs. With the Teach In function, motion sequences are transferred to a program by returning and storing positions. The Teach In function can be used in the MDI CNC operating mode.
- AUTO
In AUTO (automatic) CNC operating mode, the part programs are executed fully automatically once they have been selected in the workpiece, part program or subprogram directory (normal operation). During AUTO mode it is possible to generate and correct another part program.

In the MDI and AUTO CNC operating modes, the sequence of a program can be modified using the following program control functions:

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

Glossary

Functions and terms

SINUMERIK 840D sl

CNC program messages

All messages programmed in the part program and all alarms recognized by the system are displayed on the operator panel in plain text. Alarms and messages are displayed separately. Messages can be programmed to provide the user with information about the current machining situation during program execution.

→ [Alarms and messages](#)

CNC program transfer

→ [Manage MyPrograms](#)

CNC user memory

All programs and data, such as part programs, subprograms, comments, tool offsets, and zero offsets/frames, as well as channel and program user data, can be stored in the shared CNC user memory. The CNC user memory is buffered with a battery.

CNC user memory, additional

Option; order codes D01 ... D06

Article No.: 6FC5800-0AD00-0YB0

The CNC user memory on the NCU can be expanded by 2 MB in each case using this option.

→ [HMI user memory, additional on CF card of NCU](#)

CNC user memory, expanded

Option; order code P77

Article No.: 6FC5800-0AP77-0YB0

The function "CNC user memory expanded" increases the CNC user memory to 100 MB. With option P12 "HMI user memory, in addition on CF card of NCU", it is possible to increase the size of the CNC user memory up to 6 GB. With a PCU 50 it is possible to increase the size of the CNC user memory up to 40 GB.

→ [HMI user memory, additional on CF card of NCU](#)

COA interface for compiled OEM cycles, Run MyCCI /COOC

Option; order code M67

Article No.: 6FC5800-0AM67-0YB0

SINUMERIK Integrate Create MyCCI /COOC is an interface for the OEM's own developments relating to the CNC interpreter. Special calculations that are too complex to implement in the CNC language (e.g. intensive matrix algebra) can be programmed in C++ with a mathematics library to be provided by the user. CNC commands cannot be executed at C++ level.

The development of customized compile cycles in the interpreter is dependent upon a COA agreement and one-off purchase of the option SINUMERIK Integrate Create MyCCI /INT.

Collision avoidance

→ [Collision avoidance in real time](#)

Collision avoidance in real time

Option; order code S02

Article No.: 6FC5800-0AS02-0YB0

The SINUMERIK collision avoidance option provides reliable protection against undesirable collisions between moving and non-moving machine components in the machine's working area. The protection is effective in all operating situations.

SINUMERIK collision avoidance is focused on critical situations encountered in practice, e.g. during machine setup or machining interruptions – in other words, when the operator intervenes in the process.

- 3D collision detection in real time
- Monitoring non-moving and moving machine components
- Efficient modeling of collision objects in SINUMERIK Operate on the PC with NX SINUMERIK Collision Avoidance

Requirements:

- SINUMERIK CNC software version 4.5 SP2 and higher
- 1-channel machine with one NCU
- SINUMERIK NCU 720.3B PN or NCU 730.3B PN (recommended)
- Simultaneous recording
- 3D simulation 1 (finished part)

Communication interface to a KUKA robot, Run MyCC /RODI

Option; order code N65

Article No.: 6FC5800-0AN65-0YB0

SINUMERIK Integrate Run MyCC /RODI is the communication interface to a KUKA robot that is operated using interpolation in the SINUMERIK.

→ [Machining with robot, Run MyRobot /Machining](#)

Restricted functionality of export versions:
Not possible.

Compensation of a forced mechanical coupling in the machine coordinate system, Run MyCC /AXCO

Option; order code M81

Article No.: 6FC5800-0AM81-0YB0

SINUMERIK Integrate Run MyCC /AXCO allows movement of an axis that occurs due to mechanical coupling of an axis to a following axis, to be compensated such that the axis remains mechanically stationary despite the coupling. The motor of the coupled axis is rotated according to the set coupling ratio.

Restricted functionality of export versions:
Not possible.

Compile cycles interface, Run MyCCI /xy

SINUMERIK Integrate Run MyCCI /xy are loadable compile cycles and offer special interfaces for customized developments. This software (interface) is produced with versions of the development tools which are identical to those used for the NCK basic software of the SINUMERIK 840D sl.

For this special application, the customer uses software (GNU compiler/linker) in a Cygwin software shell on a Windows PC. This concept of loadable interfaces therefore allows the OEM to develop real-time applications in C/C++ and to load them as customized compile cycles.

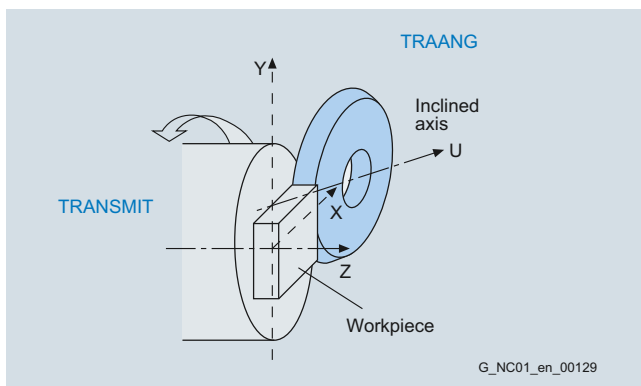
→ [Develop compile cycles, Create MyCC](#)
 → [Integrated tool monitoring and diagnostics IMD BASE, Run MyCC /IMD](#)
 → [Universal spatial compensation interface, Run MyCCI /UCI](#)
 → [COA interface for compiled OEM cycles, Run MyCCI /COOC](#)

Concatenated transformations

Two transformations can be concatenated using the TRACON command: TRAANG (inclined axis), as the base transformation, can be linked with TRAORI (5-axis transformation), TRANSMIT (face machining of turned parts) and TRACYL (cylinder surface transformation).

Applications:

- Turning-milling with mechanical non-orthogonal Y axis to X, Z (inclined-bed turning milling machine)
- Grinding contours programmed with TRACYL (cylinder processing)
- Finishing an out-of-center contour created with TRANSMIT.



Grinding a TRANSMIT contour with inclined axis

Glossary

Functions and terms

SINUMERIK 840D sl

Continue machining at the contour (retrace support), Run MyCC /RESU

Option; order code M24

Article No.: 6FC5800-0AM24-0YB0

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 (selected solely from the view of the workpiece) following an interruption in machining without exact knowledge of the part program in order to continue machining the workpiece from that particular location.

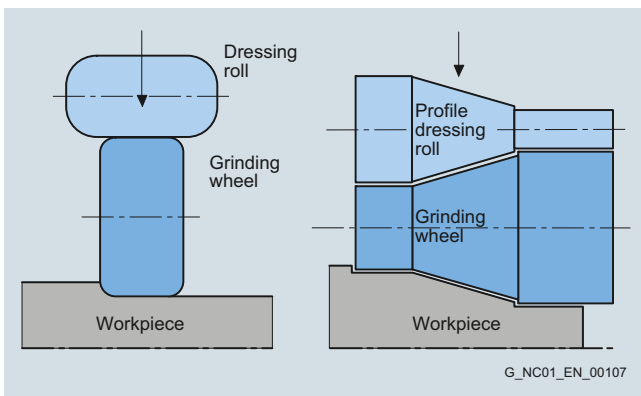
The Continue machining at the contour function (retrace support) contains a ring buffer for the geometric information of the blocks already executed. A new part program is generated from this for reverse travel.

Continue machining is used, for example, when the machine operator only notices the interruption in a 2D laser or water jet cutting operation a few blocks after the actual interruption. The head has usually already progressed further in the machining process, and must, therefore, be returned to continue machining.

Continuous dressing (parallel dressing)

With the continuous dressing 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 tool 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 a true tool radius compensation.

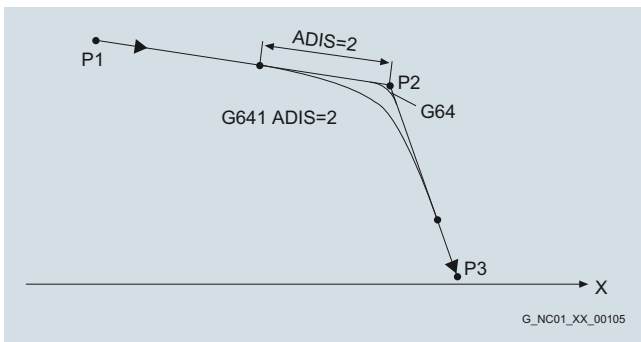


Continuous dressing (parallel dressing)

Continuous-path mode with programmable rounding clearance

The aim of the continuous-path mode is to avoid excessive deceleration at the block boundaries and to achieve as constant a tool path velocity as possible during tangential transitions from one block to the next. Because the tool does not stop at block boundaries, no undercuts are made on the workpiece.

If continuous-path mode G64 is selected, reduction in velocity takes place and contour corners are rounded at non-tangential transitions. A soft 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 input simple contours quickly. With the aid of help displays in the editor, you can program 1-point, 2-point or 3-point definitions with transition elements chamfer or corner easily and clearly by entering Cartesian coordinates and/or angles.

Contour handwheel

Option; order code M08

Article No.: 6FC5800-0AM08-0YB0

When the contour handwheel function is activated, the handwheel has a velocity-generating effect in AUTO and MDI CNC operating modes on all programmed traversing movements of the path and synchronized axes.

A feedrate specified via the CNC program becomes ineffective and a programmed velocity profile is no longer valid. The feedrate, in mm/min, results from the handwheel pulses as based on pulse weighting (machine data) and the active increment.

The handwheel's direction of rotation determines the direction of travel:

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

→ [Feedrate interpolation \(feed characteristic\)](#)

Contour monitoring

The following error is monitored within a definable tolerance band as a measure of contour accuracy. 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. If the channel is interrupted or in the reset state, contour monitoring is not active. Contour monitoring is also deactivated during execution of the travel to fixed stop function.

→ [Travel to fixed stop](#)

Contour monitoring with tunnel function

Option; order code M52

Article No.: 6FC5800-0AM52-0YB0

With 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 for high-quality workpieces. A cylindrical tunnel (tolerance field) with a definable diameter is placed around the programmed path.

If during machining the deviation from the path caused by axis errors 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.

Coupling, transformation and sensor technology, Run MyCC /PCTS

Option; order code N21

Article No.: 6FC5800-0AN21-0YB0

SINUMERIK Integrate Run MyCC /PCTS is a package for coupling, transformation and sensor technology.

No general application.

Restricted functionality of export versions:
Not possible.

Crank interpolation, Run MyCC /CRIP

Option; order code N04

Article No.: 6FC5800-0AN04-0YB0

The SINUMERIK Integrate Run MyCC /CRIP crank interpolation function supports simple programming and machining of pin bearing seats on a crankshaft. The function can be utilized in more than one channel, which means that a single workpiece can be machined simultaneously with several grinding units. The function calculates the compensating movement of the grinding wheel in relation to the rotating workpiece surface. As with normal cylindrical grinding, the pin bearing journal is programmed as a radial distance X between the workpiece and the grinding wheel.

Restricted functionality of export versions:
Not possible.

Glossary

Functions and terms

SINUMERIK 840D sl

Cross-mode actions ASUB and synchronized actions

Option; order code M43

Article No.: 6FC5800-0AM43-0YB0

Asynchronous subprograms ASUB make it possible to respond immediately to high-priority events not only during program execution, but in all CNC operating modes and program states. In the case of such an interrupt, it is possible to start an asynchronous subprogram in JOG. The asynchronous subprogram can be used, for example, to bring the grinding wheel to a safe position to avoid collision.

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

→ [Interrupt routines with fast retraction from the contour](#)

Cycle protection

→ [Lock MyCycles](#)

Cycle support

The technology cycles for drilling, milling and turning and the measuring cycles are supported by cycle screens. Similar input screens are also available to program contours. Users can also define their own softkeys, input fields and displays using "SINUMERIK Operate runtime license OA Easy Screen".

→ [Use HMI applications, Run MyScreens](#)

<i>Overview of cycles - couplings</i>		
Coupled-motion axes (TRAIL)	Basic scope	CYCLE700 generic coupling: TRAILON CYCLE701 generic coupling: TRAILOF
Synchronous spindle/multi-edge turning (COUP) replaced by → Generic coupling	Shell cycle for program compatibility with 6FC5800-0AM14-0YB0	CYCLE704 generic coupling: COUPDEF CYCLE705 generic coupling: COUPON CYCLE706 generic coupling: COUPONC CYCLE707 generic coupling: COUPOF CYCLE708 generic coupling: COUPOFS CYCLE709 generic coupling: COUPDEL CYCLE710 generic coupling: COUPRES
Master value coupling and curve table interpolation (LEAD) replaced by → Generic coupling	Shell cycle for program compatibility with 6FC5800-0AM20-0YB0	CYCLE702 generic coupling: LEADON CYCLE703 generic coupling: LEADOF
Electronic gear function (EG) replaced by → Generic coupling	Shell cycle for program compatibility with 6FC5800-0AM22-0YB0	CYCLE711 generic coupling: EGDEF CYCLE712 generic coupling: EGON CYCLE713 generic coupling: EGONSYN CYCLE714 generic coupling: EGONSYNE CYCLE715 generic coupling: EGOFC CYCLE716 generic coupling: EGOFS CYCLE717 generic coupling: EGDEL
Generic coupling CP-Standard	Basic scope	CYCLE700 generic coupling: TRAILON CYCLE701 generic coupling: TRAILOF
Generic coupling CP-Static	6FC5800-0AM75-0YB0	CYCLE700 generic coupling: TRAILON CYCLE701 generic coupling: TRAILOF
Generic coupling CP-Basic	6FC5800-0AM72-0YB0	CYCLE702 generic coupling: LEADON CYCLE703 generic coupling: LEADOF CYCLE704 generic coupling: COUPDEF CYCLE705 generic coupling: COUPON CYCLE706 generic coupling: COUPONC CYCLE707 generic coupling: COUPOF CYCLE708 generic coupling: COUPOFS CYCLE709 generic coupling: COUPDEL CYCLE710 generic coupling: COUPRES
Generic coupling CP-Comfort	6FC5800-0AM73-0YB0	CYCLE700 generic coupling: TRAILON CYCLE701 generic coupling: TRAILOF
Generic coupling CP-Expert	6FC5800-0AM74-0YB0	CYCLE702 generic coupling: LEADON CYCLE703 generic coupling: LEADOF CYCLE704 generic coupling: COUPDEF CYCLE705 generic coupling: COUPON CYCLE706 generic coupling: COUPONC CYCLE707 generic coupling: COUPOF CYCLE708 generic coupling: COUPOFS CYCLE709 generic coupling: COUPDEL CYCLE710 generic coupling: COUPRES CYCLE711 generic coupling: EGDEF CYCLE712 generic coupling: EGON CYCLE713 generic coupling: EGONSYN CYCLE714 generic coupling: EGONSYNE CYCLE715 generic coupling: EGOFC CYCLE716 generic coupling: EGOFS CYCLE717 generic coupling: EGDEL

Glossary

Cycle overview

SINUMERIK 840D sl

<i>Overview of cycles - interpolation, measurement</i>		
Advanced Surface	6FC5800-0AS07-0YB0	CYCLE832 high-speed settings
Measuring cycles	6FC5800-0AP17-0YB0	CYCLE150 Measurement: Measurement result (measurement result display and selection/deselection of log)
		CYCLE961 milling measurement: Corner – right-angled corner/any corner
		CYCLE971 milling measurement: Measure tools
		CYCLE971 milling measurement: Measure tools – calibrate tool probe
		CYCLE973 turning measurement: Calibrate probe – length/radius at surface/radius in slot
		CYCLE974 turning measurement: Inner/outer diameter
		CYCLE976 milling measurement: Calibrate probe – length/radius in ring/radius at edge/calibrate on sphere
		CYCLE977 milling measurement: Hole/rectangular pocket
		CYCLE977 milling measurement: Edge distance – slot/web
		CYCLE977 milling measurement: Circular/rectangular spigot
		CYCLE978 milling measurement: Edge distance – set edge
		CYCLE979 milling measurement: Hole/inner circle segment
		CYCLE979 milling measurement: Spigot/outer circle segment
		CYCLE982 turning measurement: Measure tools (turning tools, milling tools, drills)
		CYCLE982 turning measurement: Measure tools – calibrate tool probe
		CYCLE994 turning measurement: Inner/outer diameter
		CYCLE995 milling measurement: Measure 3D - angular deviation spindle
		CYCLE997 milling measurement: Measure 3D – 1 sphere/3 spheres
		CYCLE998 milling measurement: Measure 3D – align plane
		CYCLE998 milling measurement: Edge distance – align edge
CUST_MEACYC Manufacturer cycle for measuring cycles (formerly CYCLE198 and CYCLE199)		
Measure kinematics	6FC5800-0AP18-0YB0	NoCYCLE996 milling measurement: Measure 3D – measure kinematics

<i>Overview of cycles - programming language</i>		
Online ISO dialect Interpreter	Basic scope	CYCLE301 ISO internal subprogram: Data conversion inch/metric
		CYCLE305 ISO high-precision contour control (G05)
		CYCLE308 ISO-M look-ahead control (G08)
		CYCLE3106 ISO lift rapid (G10.6)
		CYCLE322 ISO working area limitation set (G22)
		CYCLE323 ISO working area limitation cancel (G23)
		CYCLE328 ISO first reference point return (G28/G27/G30.1)
		CYCLE330 ISO 2.3.4. reference point return (G30)
		CYCLE3512 ISO-T polygonal turning (G50.2/G51.2)
		CYCLE370T ISO-T finishing (G70)
		CYCLE371T ISO-T longitudinal turning (G71/G77)
		CYCLE3721 ISO-M contour repeating (G72.1/G72.2)
		CYCLE372T ISO-T face turning (G72/G79)
		CYCLE373T ISO-T pattern repeating (G73)
		CYCLE374T ISO-T deep hole drilling and recessing (G74/G75)
		CYCLE375T ISO-T deep hole drilling and recessing
		CYCLE376T ISO-T thread cutting (G78/G76)
		CYCLE381M ISO-M drilling (G81/G82/G85/G86/G89)
		CYCLE383M ISO-M drilling (G73/G83)
		CYCLE383T ISO-T deep hole drilling (G83/G87)
		CYCLE3841 ISO-M rigid tapping
		CYCLE384M ISO-M tapping (G74/G84)
		CYCLE384T ISO-T tapping (G84/G88)
		CYCLE385T ISO-T drilling (G85/G89)
		CYCLE387M ISO-M drilling (G76/G87)
		CYCLE395 ISO-T roughing
		CYCLE396 ISO interrupt program call (M96)
		CYCLE398 ISO-T threading
		CYCLE861 ISO-M back boring

Overview of cycles - programming support		
Technological cycles drilling/milling/turning	Basic scope	CUST_800 Manufacturer cycle for swiveling (formerly TOOLCARR)
		CUST_832 Manufacturer cycle for high speed settings (formerly CYC_832T)
		CUST_M6 Manufacturer cycle for tracking tool changes with SERUPRO
		CUST_MULTICHAN Manufacturer cycle for multi-channel editor
		CUST_T Manufacturer cycle for tracking tool changes with SERUPRO
		CUST_TECHCYC Manufacturer cycle for technology cycles (formerly ST_CUST)
		CYCLE60 milling: Engraving
		CYCLE61 milling: Face milling
		CYCLE62 contour turning and milling: Contour - contour call
		CYCLE63 contour milling: Pocket, spigot, all with residual material
		CYCLE64 contour milling: Predrilling
		CYCLE70 milling: Thread milling
		CYCLE71 milling compatibility with 802D sl: Face milling
		CYCLE72 contour milling: Path
		CYCLE76 milling: Spigot – rectangular spigot
		CYCLE77 milling: Spigot – circular spigot
		CYCLE78 drilling: Thread – drill thread milling
		CYCLE79 milling: Multi-edge spigot
		CYCLE800 milling: Swivel plane, align tool
		CYCLE801 drilling: Positions – grid or frame
		CYCLE802 drilling: Positions – any positions
		CYCLE81 drilling: Centering
		CYCLE82 drilling: Drilling reaming – drilling
		CYCLE83 drilling: Deep-hole drilling 1
		CYCLE830 drilling: Deep-hole drilling 2
		CYCLE832 milling: High-speed settings
		CYCLE84 drilling: Thread – rigid tapping
		CYCLE840 drilling: Thread – tapping with compensating chuck
		CYCLE85 drilling: Drilling reaming – reaming
		CYCLE86 drilling: Boring
		CYCLE87 drilling compatibility with 802D sl: Boring pass 3
		CYCLE88 drilling compatibility with 802D sl: Boring with stop
		CYCLE89 drilling compatibility with 802D sl: Boring pass 5
		CYCLE899 milling: Slot - open slot
		CYCLE90 milling compatibility with 802D sl: Thread milling
		CYCLE92 turning: Parting
		CYCLE93 turning compatibility with 802D sl: Grooving
		CYCLE930 turning: Grooving
		CYCLE94 turning compatibility 802D sl: Undercut form E and F
		CYCLE940 turning: Undercut – form E, form F, undercut thread DIN, undercut thread
		CYCLE95 turning compatibility stock removal (with SW 2.5 and 802D sl)
		CYCLE951 turning: Stock removal
CYCLE952 contour turning: Stock removal, plunging, plunge cutting, all with residual material		
CYCLE96 turning compatibility 802D sl: Thread undercut form A, B, C, D		
CYCLE97 turning compatibility 802D sl: Thread cutting		
CYCLE98 turning: Thread – thread chain		
CYCLE99 turning: Thread – longitudinal, taper, face		
HOLES1 drilling: Positions – row of holes		
HOLES2 drilling: Positions – circle of holes		
LONGHOLE milling: Slot – elongated hole		
POCKET3 milling: Pocket – rectangular pocket		
POCKET4 milling: Pocket – circular pocket		
PROG_EVENT ASUB for supporting special modes		
SLOT1 milling: Slot – longitudinal groove		
SLOT2 milling: Slot – circumferential groove		
Pocket milling with free contour definition and islands	Basic scope	CYCLE63 milling contour pocket: Residual material detection
		CYCLE64 predrilling contour pocket: Residual material detection
Residual material detection and machining for contour pockets and stock removal	6FC5800-0AP13-0YB0	CYCLE63 milling contour pocket: Residual material detection
		CYCLE64 predrilling contour pocket: Residual material detection
		CYCLE952 stock removal along contour: Residual material detection

Glossary

Cycle overview

SINUMERIK 840D sl

Overview of cycles - programming support

ShopTurn/ShopMill machining step programming (ShopMill)	6FC5800-0AP17-0YB0	E_CALL subprogram
		E_CI_CO contour spigot
		E_CI_RE contour spigot residual material
		E_CLAMP multiple clamping
		E_CON contour
		E_CONFIG settings
		E_CP_CE centering a contour pocket
		E_CP_CO solid machining a contour pocket
		E_CP_DR predrilling a contour pocket
		E_CP_RE contour pocket residual material
		E_CR_HEL helix
		E_DR drilling
		E_DR_BGF drill and thread milling
		E_DR_BOR boring
		E_DR_PEC deep-hole drilling 1
		E_DR_PEC2 deep-hole drilling 2
		E_DR_REA reaming
		E_DR_SIN countersinking
		E_DR_TAP tapping
		E_END end of program
		E_HEAD program header
		E_MC_LP calibrate workpiece probe: Length on surface
		E_MC_MPT calibrate tool probe calibrate (tool probe)
		E_MC_RC calibrate workpiece probe radius at edge
		E_MC_RR calibrate workpiece probe radius in ring
		E_MC_RSP calibrate workpiece probe radius on sphere
		E_MI_CON path milling
		E_MI_EDG multi-edge
		E_MI_PL face milling
		E_MI_TR thread milling
		E_MI_TXT engraving cycle
ShopTurn/ShopMill machining step programming (ShopTurn)	6FC5800-0AP17-0YB0	F_BARLO bar loader
		F_CI_CO contour spigot
		F_CI_RE contour spigot residual material
		F_CON contour
		F_CONFIG settings
		F_CP_CE centering a contour pocket
		F_CP_CO solid machining a contour pocket
		F_CP_DR predrilling a contour pocket
		F_CP_RE contour pocket residual material
		F_DR drilling
		F_DR_BGF drill thread milling
		F_DR_BOR boring
		F_DR_PEC deep-hole drilling 1
		F_DR_PEC2 deep-hole drilling 2
		F_DR_REA reaming
		F_DR_SIN countersinking
		F_DR_TAP tapping
		F_DRILL deep-hole drilling centered
		F_END end of program
		F_GROOV grooving
		F_HEAD program header
		F_HOME tool change point
		F_MC_FCE measurement: Front edge
		F_MC_L calibrate workpiece probe: Length
		F_MC_LP calibrate workpiece probe: Length on surface
		F_MC_MPT calibrate tool probe calibrate (tool probe)
		F_MC_R calibrate workpiece probe: Radius
		F_MC_RC calibrate workpiece probe radius at edge
		F_MC_RR calibrate workpiece probe: Radius in ring

Overview of cycles - programming support

ShopTurn/ShopMill machining step programming (ShopTurn) (continued)	6FC5800-0AP17-0YB0	F_MC_RSP calibrate workpiece probe: Radius on sphere
		F_MC_SL measure workpiece slot
		F_MP_3SP measurement: 3 spheres
		F_MP_BAR measurement: Web
		F_MP_C90 measurement: Right-angled corner
		F_MP_COR measurement: Any corner
		F_MP_CSI measurement: Inner circle segment
		F_MP_CSO measurement: Outer circle segment
		F_MP_DMI measurement: Inner diameter
		F_MP_DMO measurement: Outer diameter
		F_MP_DR measurement: Bore hole
		F_MP_EDG measurement: Edge
		F_MP_LIN measurement: Align edge
		F_MP_PIC measurement: Circular spigot
		F_MP_PIR measurement: Rectangular spigot
		F_MP_PL measurement: Align plane
		F_MP_POR measurement: Rectangular pocket
		F_MP_SL measurement: Slot
		F_MP_SPH measurement: Sphere
		F_MT_DR measurement: Drilling tools
		F_MT_MIL measurement: Milling tools
		F_MT_TUR measurement: Turning tools
		F_MI_CON path milling
		F_MI_EDG multi-edge
		F_MI_PL face milling
		F_MI_TR thread milling
		F_MI_TXT engraving cycle
		F_PARTOF parting
		F_PI_CIR circular spigot
		F_PI_REC rectangular spigot
		F_PO_CIR circular pocket
		F_PO_REC rectangular pocket
		F_PS_CIR position circle
		F_PS_FRA position frame
		F_PS_MRX position matrix
		F_PS_ROW position row
		F_PS_SEQ position sequence
		F_REL_BC retraction with swivel
		F_RELEAS retraction
		F_ROT_C C translation
		F_ROU_Z extended stock removal
		F_ROUGH stock removal corner
		F_SL_CIR circumferential groove
		F_SL_LON longitudinal groove
		F_SL_OPN open slot
		F_SUB_SP counterspindle
		F_SWIV_H swiveling
		F_TAP tapping centered
		F_TCARR swiveling
		F_TCTOOL setting tool
F_TR_CHN thread cutting thread chain		
F_TR_CON cutting taper thread		
F_TR_LON cutting longitudinal thread		
F_TR_PLA face thread cutting		
F_TS tool		
F_UCUT_D undercut DIN		
F_UCUT_E undercut form E		
F_UCUT_F undercut form F		
F_UCUT_T undercut thread		
F_TR_LON cutting longitudinal thread		

Glossary

Cycle overview

SINUMERIK 840D sl

Overview of cycles - commissioning functions

Friction compensation with adaptive characteristics	6FC5800-0AS06-0YB0	CYCLE790
Automatic servo tuning	6FC5800-0AS10-0YB0	CYCLE750 AST command: Internal cycle
		CYCLE751 AST command: Open / perform / close an optimization session
		CYCLE752 AST command: Add axis to an optimization session
		CYCLE753 AST command: Select optimization mode
		CYCLE754 AST command: Add / remove data set
		CYCLE755 AST command: Back up / restore data
		CYCLE756 AST command: Activate optimization results
		CYCLE757 AST command: Save optimization information
		CYCLE758 AST command: Change parameter value
		CYCLE759 AST command: Read parameter value

Data exchange between machining channels

In the program coordination function, variables shared by the channels (NCK-specific global variables) can be used for data exchange between the programs. The program message itself is separate for each channel.

→ [CNC high-level language](#)

Develop compile cycles, Create MyCC

Article No.: on request

The openness in the NC kernel permits the user to expand the real-time area of the SINUMERIK CNC through the development of customized compile cycles using SINUMERIK Integrate Create MyCC.

Restricted functionality of export versions:
Not possible.

→ [Execute compile cycles, Run MyCC](#)

Develop compile cycles on a user-specific interface, Create MyCCI

Article No.: 6FC5863-1YP00-0YB8

SINUMERIK Integrate Create MyCCI supports the development of loadable compile cycles based on application-specific interfaces with a predefined development environment. For this special application, the customer uses development tools (GNU compiler/linker) in a Cygwin software shell on a Windows PC. The concept allows OEMs to develop their own applications in C/C++ and to load them as their own compile cycles.

→ [Compile cycles interface, Run MyCCI /xy](#)

Diagnostic functions

For service purposes, a self-diagnostics program and testing aids have been integrated in the CNCs. 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 bit memories, timers and counters
- PLC inputs and outputs

For test purposes, output signals, input signals, and bit memories can be set. All alarms and messages are displayed in plain text on the operator panel along with the corresponding delete 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
- Position setpoint
- Following error
- Speed setpoint
- Actual speed value
- Trace of CNC and drive variables

Differential tower transformation, Run MyCC /DTOW

Option; order code N68

Article No.: 6FC5800-0AN68-0YB0

SINUMERIK Integrate Run MyCC /DTOW is a specific function and is not intended for general use.

Restricted functionality of export versions:
Not possible.

Display external positions, Run MyCC /EXPD

Option; order code N64

Article No.: 6FC5800-0AN64-0YB0

SINUMERIK Integrate Run MyCC /EXPD can be used to display external positions of a robot coupled via a command interface in SINUMERIK Operate. The WCS and MCS positions of the robot are displayed in the machine basic screen of the robot channel. The target positions to be approached by the robot are programmed in the NC part program as parameters of a cycle call. The cycle writes the commands to the communications buffer variables \$A_DBR and \$A_DBD between the NC and PLC. The PLC transfers them to the robot control.

→ [Operate robot, Run MyRobot](#)

Glossary

Functions and terms

SINUMERIK 840D sl

Display functions

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

- Block currently being executed
- Previous and following block
- Actual position, distance-to-go
- Current feedrate
- Spindle speed
- G functions
- Auxiliary functions
- Workpiece name
- Main program name
- Subprogram name
- All data entered, such as part programs, user data and machine data
- Help texts, tooltips

Important operating states are displayed in plain text, for example:

- Alarms and messages
- Position not yet reached
- Feedrate stop
- Program is running

DRF offset (differential resolver function)

The differential resolver function generates an additional incremental work offset in Automatic mode via the electronic handwheel. This function can be used, for example, to correct tool wear within a programmed block.

→ [Handwheel override](#)

Drive current measurement, Run MyCC /KPXT

Option; order code M82

Article No.: 6FC5800-0AM82-0YB0

SINUMERIK Integrate Run MyCC /KPXT is a specific function and is not intended for general use.

→ [Auto Servo Tuning AST](#)

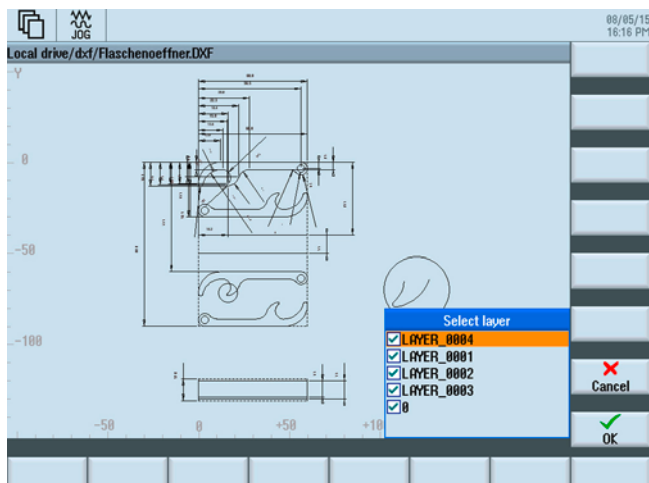
Restricted functionality of export versions:
Not possible.

DXF Reader

Option; order code P56

Article No.: 6FC5800-0AP56-0YB0

The DXF Reader option allows DXF files to be opened directly on the CNC within SINUMERIK Operate so that contours and points can be extracted from the files.

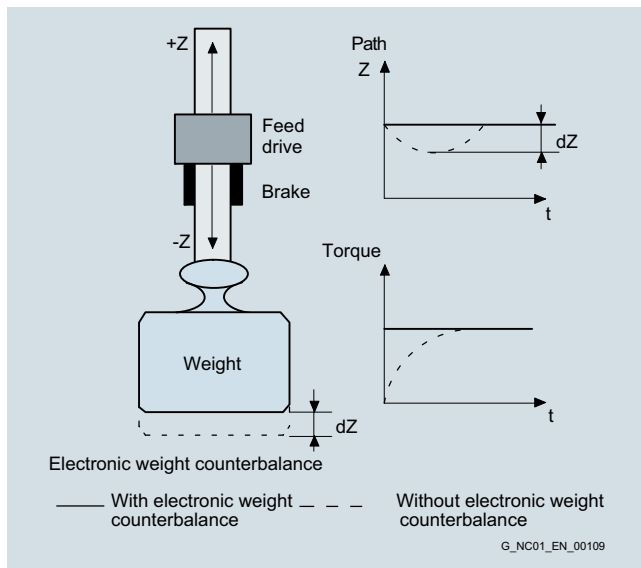


Electronic counterweight

With weight-loaded axes without mechanical or hydraulic weight counterbalance, the vertical axis drops when the brake is released and the servo enable is switched on. The undesired lowering dZ of the axis can be compensated by activating electronic weight counterbalance. After releasing the brake, the constant weight counterbalance torque maintains the position of the vertical axis.

Sequence:

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



Electronic counterweight

Electronic gear

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

The electronic gear function makes it possible to control the movement of a following axis, depending on up to 5 leading axes. The relations between the leading axis and the following axis are 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 and 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). Using the electronic gear with non-linear coupling, it is possible to create convex tooth faces during gear cutting and also to compensate the non-linear properties of the process, for example.

Restricted functionality of export versions:

The number of simultaneously traversing axes is restricted to 4.

→ [Generic coupling](#)

Electronic handwheels

Using electronic handwheels, it is possible to move selected axes simultaneously in manual mode. The handwheel clicks are analyzed by the increment analyzer. 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 100 kHz. A third handwheel can also be operated as a contour handwheel. The Contour handwheel function permits use of a handwheel on conventional turning machines (for ShopTurn applications, for example) 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 AUTO and MDI CNC operating modes, i.e. 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,

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.

Glossary

Functions and terms

SINUMERIK 840D sl

Electronic Key System EKS

Option; order code P53

Article No.: 6FC5800-0AP53-0YB0

Support of the Electronic Key System (EKS) in SINUMERIK MPPs (Machine Pushbutton Panels)

Electronic transfer CP

Option; order code M76

Article No.: 6FC5800-0AM76-0YB0

In presses with transfer step tools as well as in large-part transfer presses, a modern transfer system handles part transport. Positioning drives are controlled in step with the press's main motion. The Electronic transfer CP makes it possible to control motion sequences in transfer systems, such as gripper rails or suction systems, etc., depending on a master value which corresponds to the current ram position of the press.

The electronic transfer CP includes the options:

- Position switching signals/cam controller
- Polynomial interpolation
- Generic coupling CP-Comfort
- Cross-mode actions ASUB and synchronized actions
- I/O interfacing via PROFIBUS DP
- Synchronized actions stage 2
- Pairs of synchronous axes (gantry axes)

Combinations of these individual options satisfy all requirements for highly dynamic and positionally accurate transfer controls.

When using the Electronic transfer CP option, the spindle and tool offset functions cannot be activated.

Restricted functionality of export versions:

The number of simultaneously traversing axes is restricted to 4.

→ Position switching signals/cam controller

→ Polynomial interpolation

→ Generic coupling CP-Comfort

→ Cross-mode actions ASUB and synchronized actions

→ I/O interfacing via PROFIBUS DP and PROFINET

→ Synchronized actions stage 2

→ Pair of synchronous axes (gantry axes)

Engineered motion control, Run MyCC /EMC

Option; order code N47

Article No.: 6FC5800-0AN47-0YB0

SINUMERIK Integrate Run MyCC /EMC optimizes the feedforward control of axes by application of a mechatronic model.

Mechatronic support provided by Siemens assists with calculation of the axis model and parameterization of the filters that will act on the position setpoint and the feedforward control branches in real time.

The improved vibration behavior exhibited by the machine axes results in greater machining precision and often makes it possible to increase the jerk limitation settings in order to boost machine productivity.

Restricted functionality of export versions:

Not possible.

Engineering:

Automated commissioning, Create MyConfig

SINUMERIK Integrate Create MyConfig supports the automated commissioning of machines with SINUMERIK CNCs. Thanks to the modular concept, the software allows different machines of a series to be commissioned and upgraded with only a single software module.

The benefits to the user are a reduction in the time taken to perform commissioning and upgrades, avoidance of associated errors thanks to simplified commissioning and upgrade procedures and automated process sequences.

Engineering: Commissioning series machines

Files called series start-up files can be generated to enable transfer of a particular configuration, in its entirety, to other CNCs that use the exact same software version, for example, CNCs that are to be used for the same machines.

Series start-up means bringing a series of CNCs to the same initial state as regards their data. You can archive/read selected CNC, PLC and PCU data for series start-up. Compensation data can be optionally saved. The drive data are stored as binary data, and cannot be modified.

Series start-ups can even be easily performed without a programming device: Simply create a startup file in the PCU, save it on a PC card in the CNC, insert this card in the next CNC, and begin the series start-up procedure. Series start-ups can also be performed via a network drive or a USB stick.

Engineering: Configure, start up, optimize, service

Preferred tools for configuring, start-up, optimization and service

	SINUMERIK Operate		SINUMERIK Integrate Create MyConfig	SINUMERIK Integrate Access MyMachine	SIMATIC STEP 7	SinuCom NC
		Auto Servo Tuning				
Configure (without machine)		Optimize individual or grouped axes retrospectively offline using customer data	Modular machines, series machines with variations	Adapt user alarms, integrate user screens, tool management, bootstick	Configure, e.g. tool management	–
Commissioning (at the machine)	1. Axes, axis-drive assignment, compensations 2. Safety Integrated incl. acceptance test and DB-SI		Edit data with UPDiff. Axis-drive assignment, adapt topology directly	Access SINUMERIK Operate and the active file system	Read in HW configuration, PLC, e.g. configure DB18, transfer user program	SinuCom NC-SI acceptance test
Optimize	Optimize axes manually, current, speed and position controller. Filter settings, trace, ...	Automatically optimize individual or grouped axes		Adapt optimization data, possibly by remote access		SinuCom NC Trace
Service	Troubleshoot	Automatically optimize individual or grouped axes retrospectively	Expand machine	Remote access via TS Adapter	Troubleshoot, optimize processes	SinuCom NC Trace → Diagnostics

Engineering: Starting up drives and backing up drive data

Options for starting up drives and backing up drive data. As a general rule, SIMATIC STEP 7 must be installed in the CNC in order to start up drives (hardware configuration, PLC user program and basic program)

With SINUMERIK Operate	Internal drives (NCU,NX) and external drives (DP,PN) (with bus system)			Via terminals (without bus system)
	NCK drives	PLC axes (FC18)	PLC axes (FB283)	Traversing commands via terminal I/Os
Commissioning	✓	✓	✓	–
Back up drive data in drive archive	✓	✓	✓	–

With Starter	Internal drives (NCU,NX) and external drives (DP,PN) (with bus system)			Via terminals (without bus system)
	NCK drives	PLC axes (FC18)	PLC axes (FB283)	Traversing commands via terminal I/Os
Commissioning	✓ (current/speed controller level only)	✓ (current/speed controller level only)	✓	✓
Back up drive data in drive archive	✓	✓	✓	✓

Glossary

Functions and terms

SINUMERIK 840D sl

Evaluation of internal drive variables

Option; order code M41

Article No.: 6FC5800-0AM41-0YB0

The evaluation of internal drive variables can be used to control, i.e. by adaptive control, a second process variable (such as a path-specific or axis-specific feedrate) depending on a measured process variable, e.g. spindle current. This permits, for example, the cutting volume to be kept constant when grinding, or closing the grinding gap faster when scratching (first touch).

Evaluation of these drive variables also permits machines and tools to be protected from overload; this evaluation also facilitates shorter machining times and an improved surface quality of the workpiece. Evaluation of internal drive variables is a requirement for implementing adaptive control (AC). Adaptive control can be parameterized within the part program as follows:

- Additive influence:
The programmed value (F word) is corrected by adding.
- Multiplicative influence:
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 drive active power in W

\$AA_TORQUE drive torque setpoint in Nm

\$AA_CURR actual axis/spindle current in A

Execute compile cycles, Run MyCC

Option; order code M04

Article No.: 6FC5800-0AM04-0YB0

SINUMERIK Integrate Run MyCC enables users to execute compile cycles which they have developed themselves.

→ [Develop compile cycles, Create MyCC](#)

Restricted functionality of export versions:

Not possible.

Execution from external storage EES

Option; order code P75

Article No.: 6FC5800-0AP75-0YB0

When option "Execution from external storage" (EES) is installed, it is possible to execute part programs from external memories without EXTCALL. A machine with several NCUs can use a common part program memory. The size of the part program memory is virtually unlimited.

Extended stop and retract ESR (CNC-controlled and drive-autonomous)

Option; order code M61

Article No.: 6FC5800-0AM61-0YB0

A safe position is assumed from the machining level without any collision between tool and workpiece. As well as the drive-autonomous stop and retract function, the CNC-controlled stop and retract function is also available. To permit gentle interpolated retraction along the path or contour, the path interpolation can be processed further for a definable period following the trigger event.

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

Extended stop and retract ESR, drive-autonomous

Option; order code M60

Article No.: 6FC5800-0AM60-0YB0

The drive-autonomous extended stop and retract function can be used to separate the workpiece from the tool quickly and without risk of damage in the event of a fault. This is distinguished from the control-based ESR function by the autonomous, purely axial stop and retraction motion of the drive, which do not take into account any CNC coupling rules.

Drive-autonomous reactions are enabled by the user only in specific machining phases, which are critical for the tool and the workpiece.

Extrapolated switching signals, Run MyCC /XOUT

Option; order code N51

Article No.: 6FC5800-0AN51-0YB0

Composite components are finding wider application, especially in the aerospace domain. The production of these components by fiber placement machines requires highly accurate output of the path-related switching positions required to cut and place fiber strands. SINUMERIK Integrate Run MyCC /XOUT switches up to 128 output signals as a function of the path.

The outputs switch as soon as the machine axis reaches the preprogrammed switching positions. Different modules can be used depending on the level of accuracy required.

- The TMC 2040PN PROFINET module supports switching signal output with micro-second precision
- The ET 200S PROFIBUS module supports switching signal output in the PROFIBUS DP cycle (position control cycle)

It is not possible to combine different modules for use in the same application.

Restricted functionality of export versions:

Not possible.

Glossary

Functions and terms

SINUMERIK 840D sl

Faster braking during grinding, Run MyCC /FABS

Option; order code N81

Article No.: 6FC5800-0AN81-0YB0

SINUMERIK Integrate Run MyCC /FABS uses the edge of an input signal to decelerate the feed axis at a faster rate when grinding if the grinding wheel touches the workpiece. The input signal can be generated by an acoustic emission sensor, for example.

The motion sequence programming must be adapted for use of this function.

Restricted functionality of export versions:
Not possible.

Feedforward control

Axial following errors can be reduced to almost zero with feedforward control. This feedforward control is therefore also called following error compensation. Particularly during acceleration in contour curvatures, e.g. circles and corners, this following error leads to undesirable, velocity-dependent contour violations.

- Velocity-dependent speed feedforward control
In velocity-dependent feedforward control, the following error can be reduced almost to zero at constant velocity.
- Acceleration-dependent torque feedforward control
You can use torque feedforward control to achieve precise contours even when the highest dynamic response is demanded. If the settings are right, you can compensate the following error almost completely, even during acceleration. The result is excellent machining precision even at high path velocities.

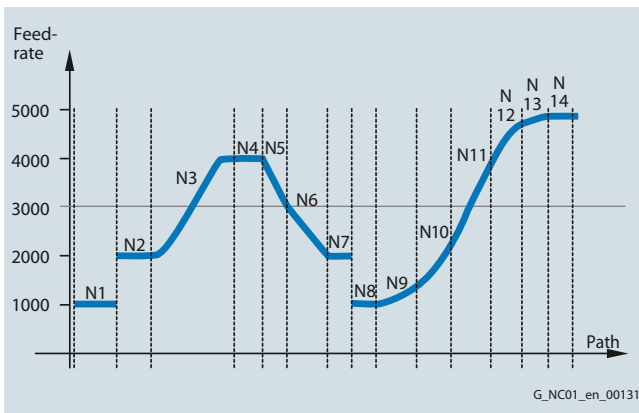
Feedrate interpolation (feed characteristic)

In accordance with DIN 66025, a constant feedrate 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 expanded to include linear and cubic profiles over the path. The cubic profiles can be programmed directly or as an interpolating spline.

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

You can program the following feedrate profiles:

- **FNORM**
Response according to DIN 66025 (default setting). An F value programmed in the CNC block is applied over the entire path of the block, and is subsequently regarded as a fixed modal value.
- **FLIN**
An F value programmed in the block can be traversed linearly (rising or falling) over the path from the actual value at the beginning of the block to the end of the block, and is subsequently regarded as a modal value.
- **FCUB**
The non-modally programmed F values (relative 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**
The feedrate profile can also be programmed directly via a polynomial. The polynomial coefficients are specified analogous to polynomial interpolation.



Programming example: Feedrate interpolation

N1 Constant feedrate profile F1000: FNORM
 N2 Abrupt setpoint velocity change F2000: FNORM
 N3 Feedrate profile via polynomial : F = FPO (4000, 6000, -4000)
 N4 Polynomial feedrate 4000 as modal value
 N5 Linear feedrate profiles F3000: FLIN
 N6 Linear feedrate 2000 as modal value
 N7 Linear feedrate, as modal value
 N8 Constant feedrate 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

→ [Polynomial interpolation](#)

Feedrate override

The programmed velocity is overridden by the actual velocity setting via the machine control panel or by the PLC (0 % to 200 %). In order for the cutting velocity along the contour to be kept constant, the feedrate calculation is referred to the operating point or tool end point.

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

Follow-up mode

In follow-up mode, an axis/a spindle can be moved independently, while the actual value continues to be detected. The traversing paths are updated in the display. Standstill, clamping and positioning monitoring functions are not effective in the follow-up mode. Once follow-up mode is cancelled, reference-point approach of the axis does not have to be repeated.

Glossary

Functions and terms

SINUMERIK 840D sl

Frame concept

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

For SINUMERIK CNCs, the frame in the CNC program transfers from one Cartesian coordinate system to 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
- Settable 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 Cartesian coordinate systems very simply by offsetting, rotating, scaling and mirroring.

The following instructions are used to program these options:

- TRANS programmable work 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 space (for measuring cycles).

The instructions can also be used several times within one program. Existing offsets can either be overwritten or new ones can be added.

Additive frame instructions:

- ATRANS additive programmable work offset
- AROT additive rotation in space or in a plane
- ASCALE scale factor (multiplication)
- AMIRROR repeated mirroring
- AROTS additive rotation referred to the solid angle projected into the planes

If swivel-mounted tools or workpieces are available, machining can be implemented very flexibly, for example:

- By machining several sides of a workpiece by rotating and swiveling the machining plane
- By machining inclined surfaces using tool length and tool radius compensation

Friction compensation, Run MyCC /FRIC

Option; order code N67

Article No.: 6FC5800-0AN67-0YB0

Replaced by:

→ [Optimized friction compensation](#)

Restricted functionality of export versions:

Not possible.

Generator operation

With the generator operation function, brief power outages can be buffered 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, following the same principle as that used by generators.

Generic coupling

We offer 5 different performance levels for generic (general) coupling (CP) of axes/spindles. The functionality is scalable via the number of leading axes to one following axis, via coupling characteristics ranging from simple functionality through to technological innovations and via the simultaneously activatable coupling types.

The options CP-Static, CP-Basic, CP-Comfort and CP-Expert are available. These options can be combined as required. The number of coupled objects actively in use at the same time is monitored, i.e. if multi-edge machining and synchronous spindle are not simultaneously in use, for example, CP-Basic is sufficient. However, if these two functions need to be used simultaneously, CP-Static will be required additionally (or, depending on the number of additional coupling functions, CP-Comfort).

Restricted functionality of export versions:

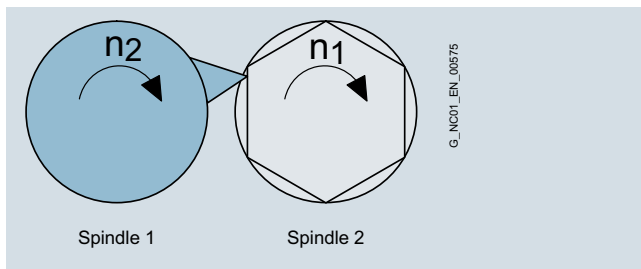
See the functional restrictions for each of the above-mentioned functions.

Generic coupling CP-Basic

Option; order code M72

Article No.: 6FC5800-0AM72-0YB0

- 4 axis pairs in simultaneous coupled motion and
- 1 × synchronous spindles/multi-edge turning and/or master value coupling/curve table interpolation and/or axial coupling in the machine coordinate system



Restricted functionality of export versions:

The number of simultaneously interpolating axes is restricted to 4.

Generic coupling CP-Comfort

Option; order code M73

Article No.: 6FC5800-0AM73-0YB0

- 4 axis pairs in simultaneous coupled motion and
- 4 × synchronous spindles/multi-edge turning and/or master value coupling/curve table interpolation and/or axial coupling in the machine coordinate system

Also:

- 1 × electronic gear for 3 leading axes without curve table interpolation and without cascading

Restricted functionality of export versions:

The number of simultaneously interpolating axes is restricted to 4.

→ Electronic gear

Glossary

Functions and terms

SINUMERIK 840D sl

Generic coupling CP-Expert

Option; order code M74

Article No.: 6FC5800-0AM74-0YB0

- 8 axis pairs in simultaneous coupled motion and
- 8 × synchronous spindles/multi-edge turning and/or master value coupling/curve table interpolation and/or axial coupling in the machine coordinate system

Also:

- 8 × electronic gear for 3 leading axes without curve table interpolation and without cascading
- 5 × electronic gear for 5 leading axes without curve table interpolation and without cascading

Restricted functionality of export versions:

The number of simultaneously interpolating axes is restricted to 4.

→ Electronic gear

Generic coupling CP-Standard

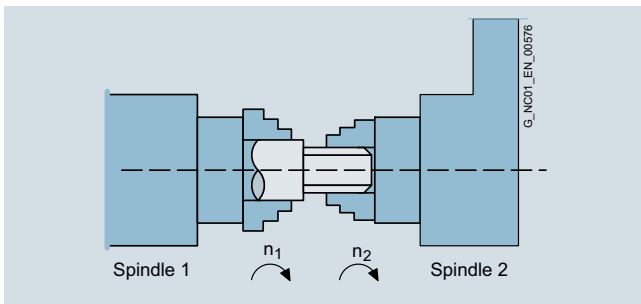
- Up to 4 × simple coupled motion with one leading axis and not used in synchronized actions

Generic coupling CP-Static

Option; order code M75

Article No.: 6FC5800-0AM75-0YB0

One simple synchronous spindle with coupling ratio 1:1, no multi-edge machining



Geometrically redundant axes, Run MyCC /GRED

Option; order code N73

Article No.: 6FC5800-0AN73-0YB0

SINUMERIK Integrate Run MyCC /GRED is a specific function and is not intended for general use.

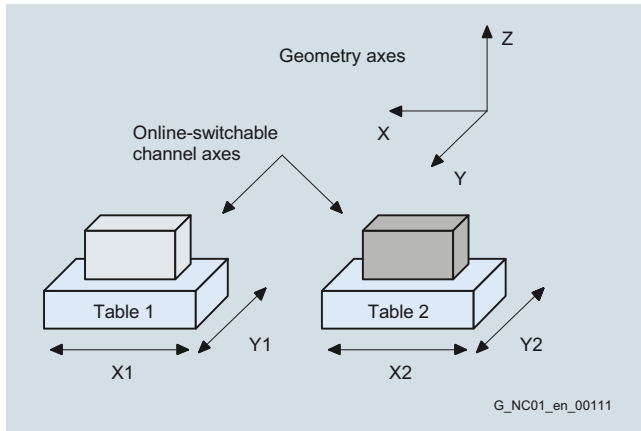
Restricted functionality of export versions:

Not possible.

Geometry axes, switchable online in the CNC program

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

Geometry monitoring interface, Run MyCCI /GPM

Option; order code M83

Article No.: 6FC5800-0AM83-0YB0

SINUMERIK Integrate Run MyCC /GPM is a specific function and is not intended for general use.

Restricted functionality of export versions:
 Not possible.

For general use, refer to:

→ [Collision avoidance in real time](#)

Grinding wheel surface speed, constant

Automatic conversion of the grinding wheel surface speed to a speed of rotation 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.

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

Glossary

Functions and terms

SINUMERIK 840D sl

Handling package

Option; order code S31

Article No.: 6FC5800-0AS31-0YB0

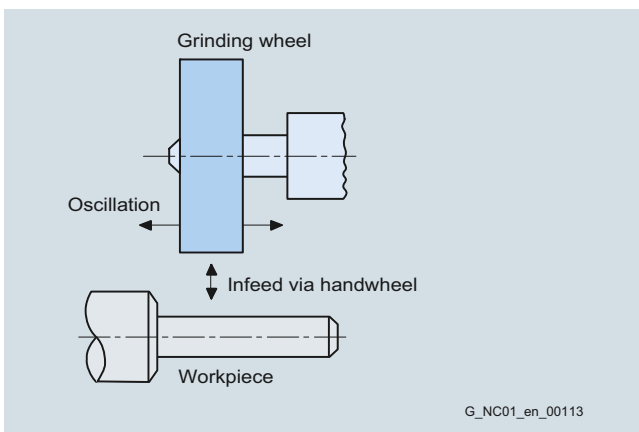
For handling systems, the handling package offers a cost-effective solution:

- 3 additional axes
- 3 additional channels
- Transformation Handling
Requirement: Loadable compile cycle
- Synchronized actions stage 2
- No tool offsets and no spindles

Restricted functionality of export versions:
Not possible

Handwheel override

With the handwheel override function, an axis can be traversed or the velocity of an axis can be overridden. The function is non-modal. At the same time, additional axes can be traversed simultaneously or using interpolation. The actual-value display is continuously updated. Application: Grinding machines



Handwheel override in AUTO CNC operating mode

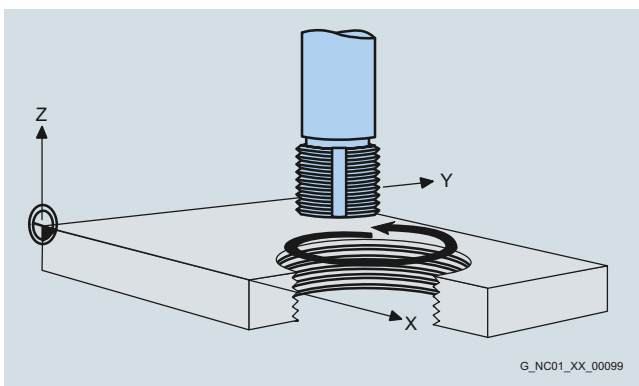
Helical interpolation 2D+6

The helical interpolation function is ideal for machining internal and external threads using form milling tools and for milling lubrication grooves.

The helix comprises 2 motions:

- Circular motion in one plane
- Linear motion perpendicular to this plane

The programmed feedrate F either refers only to the circular motion or to the total path velocity of the 3 CNC axes involved. In addition to the 2 CNC axes performing circular interpolation, other linear motion can be performed synchronously. The programmed feedrate F refers to the axes specially selected in the program.



Helical interpolation: Thread milling with form cutter

Restricted functionality of export control versions:
Not possible.

High-speed CNC inputs/outputs

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

Input signals are possible for the following:

- Several feedrate values in one block (caliper function)
The function allows modification of the feedrate through external signals. 6 digital inputs can be combined with 6 different feedrate values in a CNC block. There is no feed interruption in this case. An additional input can be used to terminate the infeed (starting a dwell time), and another input can be used to start immediate retraction. 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.
- Several auxiliary functions in one block
Several auxiliary functions can be programmed in one CNC block. These functions are transferred to the PLC depending on a comparison operation or an external signal.
- Axis-specific delete distance-to-go
The high-speed inputs cause conditional stopping and delete distance-to-go for the path or positioning axes.
- Program branches
The high-speed inputs make program branches within a user program possible.
- Rapid CNC start
Machining can be enabled conditionally in the CNC program depending on an external input.
- Analog calipers
Various feedrates, a dwell time and a retraction path can be activated depending on an external analog input (threshold values are specified via machine data).
- Safety-related signals such as emergency stop

Output signals are possible for the following:

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

→ [Position switching signals/cam controller](#)

HMI user memory, additional on CF card of NCU

Option; order code P12

Article No.: [6FC5800-0AP12-0YB0](#)

The memory for part programs and expansions to the operator interface can be extended using the additional HMI user memory on the CF card in systems without a SINUMERIK PCU.

With CNC software version 4.5 SP1 or higher (on an 8 GB CF card), the memory is extended by 6 GB.

→ [CNC user memory, additional](#)

Glossary

Functions and terms

SINUMERIK 840D sl

I/O interfacing via PROFIBUS DP and PROFINET

PROFIBUS DP represents the protocol profile for distributed I/Os. It enables high-speed cyclic communication. Advantages of PROFIBUS DP: very short bus cycle times, high degree of availability, data integrity, and standard message frame structure.

Like PROFIBUS, PROFINET as a real-time bus system supports all I/Os.

→ [PROFINET](#)

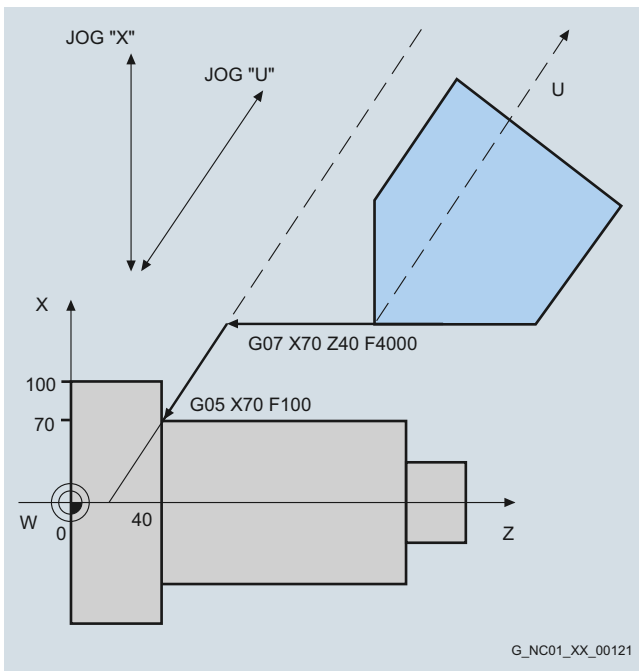
Inclined axis

Option; order code M28

Article No.: 6FC5800-0AM28-0YB0

The inclined axis function is used for fixed-angle interpolation using an oblique infeed axis. This is used primarily in conjunction with cylindrical grinding machines. The axes are programmed and displayed in the Cartesian coordinate system. Tool offsets and work offsets are also entered in the Cartesian system and transformed to the real machine axes.

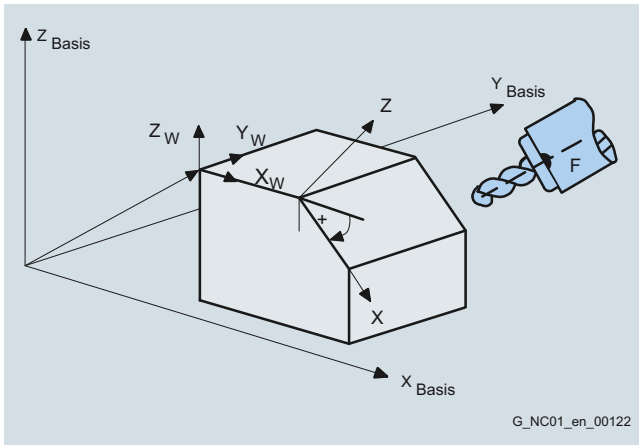
For oblique plunge-cutting with G05, it is necessary to program the start position with G07. In the JOG CNC operating mode, the grinding wheel can be traversed either in the Cartesian coordinate system or in the direction of inclined axis U (selection via the channel DB).



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

Inclined-surface machining with frames

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



Inclined-surface machining with frames

→ [Frame concept](#)

Industrial security

Industrial security includes all measures taken to plan, implement and monitor security in products and networks. The risk of hacking and other security breaches increases as more and more machine networks are implemented via the Internet. Threats of this type need to be warded off by appropriate precautions such as firewalls, encrypted connections, virus scanners, whitelisting and organizational measures/policies.

→ www.siemens.com/industrialsecurity

Integrate CNC into production, Create MyInterface

Article No.: 6FC6000-7AC00-0AA8

With SINUMERIK Integrate Create MyInterface, it is possible to easily integrate SINUMERIK CNCs into the customer's manufacturing network using defined interfaces. All essential machine data (such as status information, piece counts, alarms, messages, CNC programs, tool service life) can be exchanged between a central control system and the CNC, helping to reduce machine downtimes and increase machine capacity utilization.

Integrate screens into the user interface

→ Use HMI applications, Run MyScreens

Integrated process monitoring, Run MyCC /IPM

Option; order code M70

Article No.: 6FC5800-0AM70-0YB0

SINUMERIK Integrate Run MyCC /IPM is a specific function and is not intended for general use.

Restricted functionality of export versions:
Not possible.

Glossary

Functions and terms

SINUMERIK 840D sl

Integrated spindle monitor (S-Monitor)

Option; order code P55

Article No.: 6FC5800-0AP55-0YB0

The integrated spindle monitor (S-Monitor) is designed to allow fast, easy monitoring of additional information about the condition of a WEISS motor spindle from the WEISS Spindle Sensor Module within SINUMERIK Operate:

- Clamping status and diagnostics
- Historical spindle data relating to speed, torque
- Motor temperature
- Bearing temperature
- Additional temperature sensor (KTY)
- Spindle operating time in various control modes
- Number of tool changes
- Logistics data of spindle

Integrated tool monitoring and diagnostics, IMD

The Integrated Monitoring and Diagnostics IMD is a software package with easy access to drive data (drive torque, encoder values, setpoint speed, etc.) and program data (programmed path, programmed velocity, programmed interpolation type, etc.) directly in the real-time part of the control. The function offers interfaces to the part program (language commands), to the PLC (fast I/O) and to the HMI (files, GUD).

Integrated tool monitoring and diagnostics IMD BASE, Run MyCC /IMD

Option; order code N13

Article No.: 6FC5800-0AN13-0YB0

SINUMERIK Integrate Run MyCC /IMD integrated tool monitoring and diagnostics BASE supports the 'missing tool' monitoring function as well as programming of a maximum permissible overload for a tool. The parameters are taught on program setup as for the 'missing tool' monitoring function, and saved in the function-specific global user data (GUD). For all subsequent program runs, the taught signals will be compared with the current ones and tool overload is avoided. Language command in the part program: CC_START_TASK ("Fixed_Overload", ..., ...).

SINUMERIK Integrate Run MyCC /IMD is also the prerequisite (interface) for customized add-ons in the field of process monitoring and diagnostics. The openness in the real-time part of the SINUMERIK 840D sl is utilized, for example, by SINUMERIK Solution Partners.

Integrated tool monitoring and diagnostics IMD LIGHT, Run MyCC IMD-L

Option; order code N12

Article No.: 6FC5800-0AN12-0YB0

SINUMERIK Integrate Run MyCC /IMD-L supports a 'missing tool' monitoring function within the IMD package. For this purpose, cuts (transition from rapid traverse to creep feed) must be selected and parameterized (CC_START_TASK ("MissingTool", ..., ...)) in the part program. For example, select torque of spindle or torque of a specific axis. The torque can be learned in an initial program run (teach-in).

In productive operation, the tool is assumed to be present when the actual signal is equal to the taught signal. If not, an individually parameterized alarm response is triggered (RESET, CANCEL, Stop Spindle, Stop Axes, etc.).

Intermediate blocks for tool radius compensation

Traversing motion with selected tool offset can be interrupted by a limited number of intermediate blocks (block without axis motion in the compensating plane). The permissible number of intermediate blocks can be set using system parameters.

→ [Tool radius compensation](#)

Internal/external drives

Internal drives are connected to an NCU/NX via DRIVE-CLiQ.
External drives are connected via PROFIBUS/PROFINET.

All internal and external drives are supported by SINUMERIK Operate:

- Parameter lists
- Diagnostics (alarms with time synchronization, traffic lights images, drive servicing)
- Series start-up

Third-party drives and SINAMICS G110/G120 are not supported by SINUMERIK Operate.

→ [Axis/spindle](#)

Interrupt routines with fast retraction from the contour

Option; order code M42

Article No.: [6FC5800-0AM42-0YB0](#)

Interrupt routines are special subprograms 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 saved automatically. It is also possible to save such things as the current states of G functions and the current offsets (SAVE mechanism) in buffer storage, making it possible to resume the program at the point of interruption later 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 such as the switching of a high-speed CNC input can trigger a movement via the special subprogram, which allows fast retraction of the tool from the workpiece contour currently being machined.

The retraction angle and the distance retracted can also be parameterized. An interrupt routine can also be executed following the fast retraction.

Inverse-time feedrate

For CNC's, it is possible to program the time required to traverse the path of a block (rpm) instead of programming the feedrate for the axis movement with G93. If the path lengths differ greatly from block to block, 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; order code M21

Article No.: [6FC5800-0AM21-0YB0](#)

Using involute interpolation, it is possible to program a spiral contour with the shape of a circular involute in one CNC 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 shorter machining time. Undesirable facets, which could result from coarse polygon functions, 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.

Glossary

Functions and terms

SINUMERIK 840D sl

Job list

This can be used to create a job list (loading list) for each workpiece.

This job list contains instructions on making the following preparations for executing part programs, even when multiple channels are involved:

- Parallel setup (LOAD/COPY):
Load or copy main programs and subprograms and associated data such as initialization programs INI, R parameters RPA, user data GUD, work offsets UFR, tool/magazine data TOA/TMA, setting data SEA, protection areas PRO, and sag/angularity CEC from the PCU's hard disk into the CNC's work memory.
- Preparations for the CNC Start SELECT:
Select programs in different channels and make initial preparations for processing them
- Parallel clearing, reversed LOAD/COPY:
Remove/unload main programs, subprograms and associated data from the CNC's work memory to the hard disk

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

Laser switching signal, high-speed, Run MyCC /HSLC

Option; order code M38

Article No.: 6FC5800-0AM38-0YB0

For high-speed laser machining, e.g. of aperture plates, the laser is switched on and off automatically and with a very high positional accuracy relating to the path. 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. The laser switching signal can also be coupled to an adjustable G1 feedrate threshold value, if required. To achieve the fastest possible responses, the switching on and off of the digital laser signal is controlled by a hardware timer with microsecond precision depending 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. It is only necessary to program a release with CC_FASTON (DIFF1, DIFF2) for the path section in which the laser is switched on or off. Together with this release, the two offset values, which can offset the switching on and off of the laser by a specific path differential in relation to the position setpoint are entered.

Leadscrew error compensation / measuring system error compensation

On SINUMERIK CNCs, interpolating compensation is divided into two categories:

- Leadscrew error compensation (LEC) or measuring system error compensation (MSEC) 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 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).

Tolerances in ball screw production, however, result in large dimensional deviations to a lesser or greater extent (referred to as leadscrew error). These are complemented by dimensional deviations caused by the measuring system as well as its installation tolerances on the machine (measuring system errors), plus any machine-dependent error sources. Because these dimensional deviations directly affect the accuracy of workpiece machining, they must be compensated for by the relevant position-dependent compensation values.

The compensation values are derived from measured error curves and entered in the CNC in the form of compensation tables during commissioning. The relevant axis is compensated using linear interpolation between the intermediate points.

Leadscrew error compensation, bidirectional

Option; order code M54

Article No.: 6FC5800-0AM54-0YB0

Bidirectional compensation is an expansion of the leadscrew error compensation function (LEC) or the measuring system error compensation function (MSEC). By contrast with the LEC and MSEC, bidirectional compensation acts in both directions.

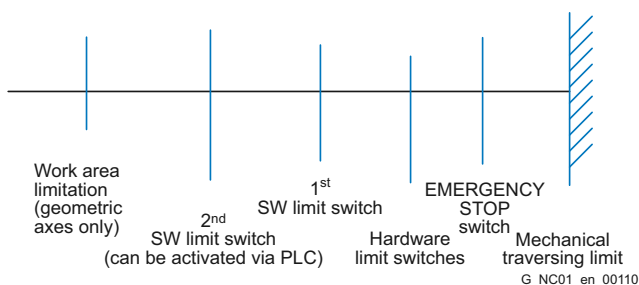
Restricted functionality of export versions:

The correctable tolerance band is restricted to 1 mm (0.039 in).

Limit-switch monitoring

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 realized either as rapid deceleration with setpoint zero or along a braking characteristic. The axes must be retracted in the opposite direction in the JOG CNC operating mode.

Software limit switches precede the hardware limit switches, are not passed, 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.



Glossary

Functions and terms

SINUMERIK 840D sl

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.

Restricted functionality of export control versions:
The number of simultaneously interpolating axes is restricted to 4.

Link control system

→ [Create MyInterface](#)

→ [Master computer interface, Access MyMachine /OPC UA](#)

Lock MyCycles

Option; order code P54

Article No.: 6FC5800-0AP54-0YB0

With SINUMERIK Integrate Lock MyCycles (cycle protection), cycles can be encrypted and then saved in the CNC where they are protected. Execution in the CNC is possible without restrictions, but it is not possible to view the cycle. This protects the internal company know-how.

The cycle can, however, be copied in an encrypted form. It can, therefore, be used on other machines. If use of the cycle on other machines is to be prevented, it can be permanently linked to a particular CNC hardware unit by means of an addition to the program.

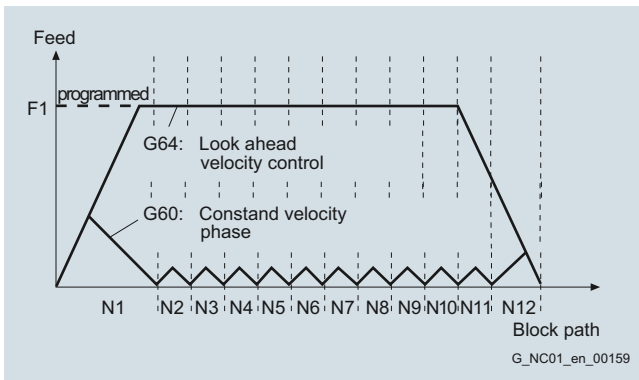
Lock MyPLC

The PLC machine program can be comprehensively protected by the know-how protection functionality provided by SIMATIC STEP 7 software version 5.5 SP3. This software guards technological knowledge against unauthorized access and modification. It prevents the reverse engineering of machines, thus safeguarding the investments you have made.

Look Ahead

During the machining of complex contours, most of the resulting program blocks have very short paths with sharp transitions. If a contour of this type is processed with a fixed programmed path velocity, an optimum result cannot be obtained. In short traversing blocks with tangential block transitions, the drives cannot attain the required final velocity because of the short path distances. Contours are rounded when traveling around corners.

The Look Ahead function is a means of optimizing the machining speed by looking ahead over a parameterizable number of traversing blocks. For tangential block transitions, the axis is accelerated and decelerated beyond block boundaries, so that no drops in velocity occur. For sharp path transitions, rounding of the contour is reduced to a programmable path dimension.



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

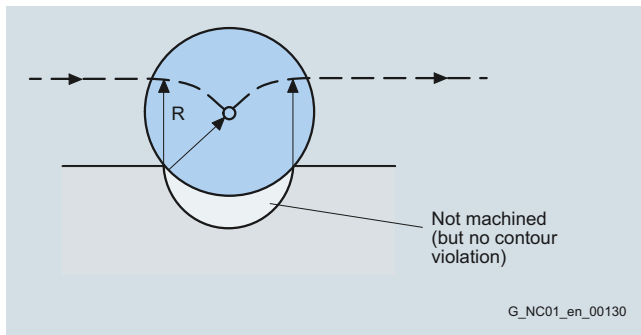
→ [Continuous-path mode with programmable rounding clearance](#)

Look-ahead detection of contour violations

With collision detection CDON on and active tool radius compensation, the CNC monitors tool paths through look-ahead contour calculation. Potential collisions are promptly identified in this way and actively prevented by the CNC.

For example, for an excessively large tool radius, the CNC identifies critical machine situations, which it compensates for by modifying the tool paths.

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



Behavior when tool radius > circle radius

Glossary

Functions and terms

SINUMERIK 840D sl

Machining channel

(Basic configuration: 1 machining channel)

Option; order codes C11 ... C19

Article No.: 6FC5800-0AC10-0YB0

Idle times can be shortened via a channel structure using parallel motion sequences, such as moving a loading gantry during machining. A machining channel must be regarded as a separate CNC 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 with the channel switchover button on the operator panel. Part programs can then be chosen and started for that specific channel. Each possible channel can run in a separate mode group.

With the options with order codes C11 ... C19, the number can be increased to up to 10 machining channels.

→ [Mode group](#)

Machining package 5 axes

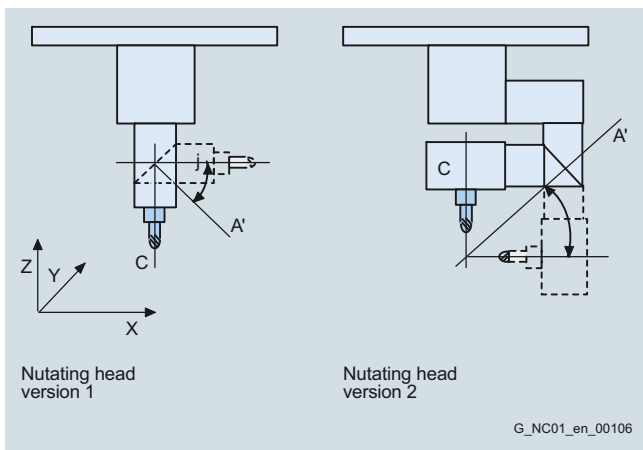
(Basic configuration: 3 axes/spindles)

Option; order code M30

Article No.: 6FC5800-0AM30-0YB0

With the machining package 5 axes, 5-axis machining tasks, such as the milling of free-form surfaces, can be simply implemented. To this end, the machining package 5 axes provides the following functions:

- 5-axis transformation with tool orientation TRAORI
In 5-axis machining, geometric axes X/Y/Z are supplemented by additional axes such as rotary axes for swiveling the tool. The machining task can be completely defined in Cartesian spatial coordinates with Cartesian position and orientation. The path vector is converted in the control into the machine axes, including position and orientation, 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 to carry out defined, oriented tool retraction.
- Tool-oriented RTCP
With the remote tool center point RTCP function, the tool swivel axes can be positioned in the 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.
- Cardan milling head/nutating head
Requirements: Machining package 5 axes with 5-axis transformation.
Using a cardan milling head in conjunction with the Nutating head function, it is possible to machine outside contours of spatially formed parts at high feedrates. To do this, the CNC performs a 5-axis transformation. 3 translatory main axes X/Y/Z determine the tool operating point; 2 rotary axes, one of which is an inclined axis (angle can be set in the machine data), permit virtually any orientation in the working area. Version 1 and version 2 of the cardan milling heads are supported. In the case of version 2, the position of the operating point does not change when the tool is swiveled; the compensating movements required for orientation changes are minimal. Includes multi-axis interpolation (> 4 interpolating axes). The machining package 5 axes does not include any additional axes/spindles which might be required.



Cardan milling head

Restricted functionality of export versions:
Not possible.

→ [Multi-axis interpolation \(> 4 interpolating axes\)](#)

→ [Axis/spindle](#)

Machining package 5 axes, additional function 7th axis

Option; order code S01

Article No.: 6FC5800-0AS01-0YB0

In combination with the machining package 5 axes or SINUMERIK MDynamics 5 axes, this option supports 7-axis interpolation. This means that redundant rotation of a workpiece is possible in the work space, with 5th/6th axis tool kinematics active at the same time. Application example: Fiber placement machines in the aerospace industry.

Restricted functionality of export versions:
Not possible.

Machining with robot, Run MyRobot /Machining

Option; order code P73

Article No.: 6FC5800-0AP73-0YB0

SINUMERIK Integrate Run MyRobot /Machining includes the options:

- Run MyCC /RODI (communication)
- Multi-axis interpolation (> 4 interpolating axes)
- Run MyHMI /3GL (for diagnostics)

SINUMERIK Integrate Run MyRobot /Machining allows a KUKA robot to be utilized, for example, as a milling robot, in combination with a SINUMERIK 840D sl as the CNC. SINUMERIK Integrate Run MyRobot /Machining includes all the options required for this purpose.

→ Communication interface to a robot, Run MyCC /RODI

→ Multi-axis interpolation (> 4 interpolating axes)

→ Use HMI applications, Run MyHMI /3GL

Restricted functionality of export versions:
Not possible.

Magnetic cogging torque compensation, Run MyCC /COCO

Option; order code N46

Article No.: 6FC5800-0AN46-0YB0

This compensation function is used particularly for electric direct drives where it is necessary to reduce the torque ripple at low speeds in order to obtain a better machining quality.

Restricted functionality of export versions:
Not possible.

Main program call from main program and subprogram

If machining operations recur frequently, it is advisable to store them in a subprogram. The subprogram is called from a main program (number of passes ≤ 9999). Eleven subprogram 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 subprogram.

Manage MyPrograms

Article No.: 6FC5864-1AP41-0YB0

SINUMERIK Integrate Manage MyPrograms provides a central system of management and archiving for CNC programs and automatic backup of SINUMERIK CNC data. For example, CNC programs can simply be transferred from Teamcenter. The advantage of this feature is that it supports simple, centralized organization of CNC data into which CNC programmers can be integrated, as well as fast access to CNC programs via networks.

It can be linked into the SINUMERIK Operate user interface without requiring the addition of supplementary external equipment on the CNC.

Manage MyTools

Article No.: 6FC5864-2AP41-0YB0

SINUMERIK Integrate Manage MyTools provides users with a comprehensive overview of actual tool data including statistics and planning for the entire production process. It supports a variety of different code carrier systems.

It offers the advantage of a closed tool data circuit - from procurement to use of the tool (procurement > tool setting > loading > setup). It illuminates potential for rationalization, e.g. achieved by cost cutting, optimization and availability.

Glossary

Functions and terms

SINUMERIK 840D sl

Manual machine

We offer the manual machine function for beginners switching over from conventional machines, but also for experienced CNC machine operators who often only use individual machining steps. The MANUAL basic screen is displayed immediately after booting the machine offering the direct machining options without having to create a part program.

→ [ShopTurn/ShopMill](#)

Master computer interface, Access MyMachine /OPC UA

Option; order code P67

Article No.: 6FC5800-0AP67-0YB0

The SINUMERIK Integrate Access MyMachine /OPC UA function enables the communication channel for the OPC UA server on the SINUMERIK PCU or the SINUMERIK NCU. The OPC UA data access method permits the reading and writing of CNC and PLC variables. OPC Unified Architecture OPC UA is a standardized, industrial communication protocol for the access to control data, e.g. by control systems.

→ www.opcfoundation.org

Master/slave for drives

Option; order code M03

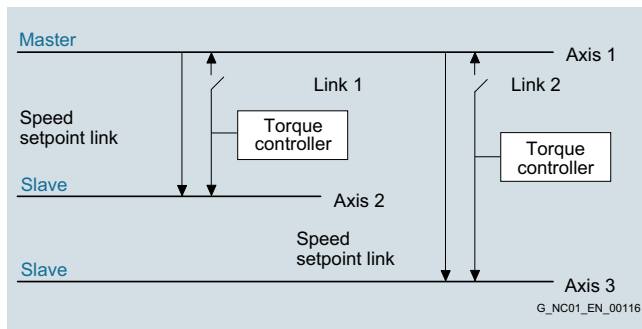
Article No.: 6FC5800-0AM03-0YB0

The master/slave for drives function is required when 2 electrical drives are mechanically linked to an axis. With this link, a torque controller ensures that both drives produce the exact same amount of torque, so that the motors do not work in opposition. In order to attain tensioning between the master and slave drives, a tension torque specifiable via machine data can be applied to the torque controller.

Application examples:

- Increasing the power rating and (occasional) mechanical linking of drives
- Drive with 2 motors that operate on a gear rack
- Remachining of wheel sets for rail vehicles
- Zero backlash reversing of mutually tensioned drives

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



Axis 1 is simultaneously the master axis for axis 2 and axis 3

Master-value coupling and curve table interpolation

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 function has been expanded to include the master value coupling function, which makes it possible for linear guide and following axes to be coupled via curve tables in the CNC program.

Any function relations between axis positions can be approximated. Soft coupling avoids the sudden change in velocity that occurs when the guide axis is activated. Offsets (e.g. 12°), scalings (e.g. 1.00023) and mirroring using frame instructions are possible.

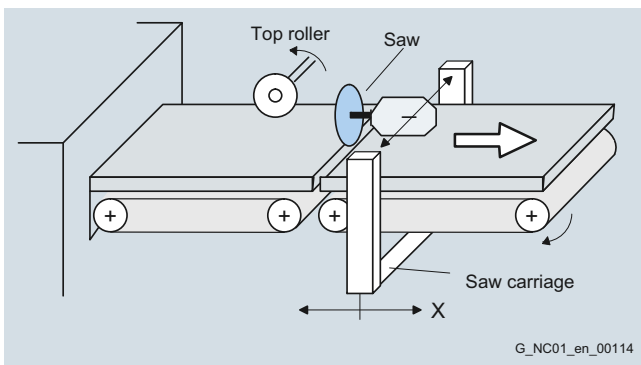
Electronic curve table interpolation replaces the cam disks that were once required for the computerized numerical control of cyclic machines. Complex sequences of motion can be easily defined using familiar CNC language elements. The external reference variable (e.g. "line shaft") is formed by the CNC's master value. The functional relation 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 using mathematical functions (normally through 3rd degree polynomials).

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, woodworking machines, printing machines).

With the SINUMERIK CNC, technological functions such as synchronism, electronic transfer and positioning for cyclic machines can be implemented. Mechanical systems (line shaft, gearing, cam disks, couplings and cams) are replaced by an electronic solution (master value coupling, curve tables, synchronized actions, and electronic cams).

In addition, the electronic functionality permits fast, axis-specific optimization, high-speed phase and path compensation, fast responses to faulty or missing parts, and fast synchronization and resynchronization, as well as decoupling from the leading axis and execution of autonomous movements. Axis cycles and synchronization calculations are carried out in the IPO cycle.

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



Example for cyclic machines: Flying saw

Restricted functionality of export control versions:
The number of simultaneously traversing axes is restricted to 4.

- Generic coupling
- Measuring stage 2
- Synchronous spindles, multi-edge turning

Measure kinematics

Option; order code P18

Article No.: 6FC5800-0AP18-0YB0

The Measure kinematics function supports the calibration of kinematic structures of 5-axis machines.

The SINUMERIK 840D sl is able to determine the parameters of kinematic transformations of the digitally or manually alignable rotary axes quickly and automatically.

The function is ideal for initial startup, because a dimension drawing of the machine is not required. The function can also be used for regular checking of the production process where high precision is required.

Glossary

Functions and terms

SINUMERIK 840D sl

Measuring cycles

Option; order code P28

Article No.: 6FC5800-0AP28-0YB0

In the case of tool measuring, the loaded tool (typically in the revolver on turning machines) is moved towards the probe, which is either in a fixed position or swiveled into the working range by a mechanical device. The automatically derived tool geometry is entered in the relevant tool offset data record.

In the case of workpiece measuring, a probe is moved towards the clamped workpiece like a tool. The flexible structure of the measuring cycles means that it is possible to handle virtually every measuring task associated with turning or milling.

The flying measurement principle is used in SINUMERIK CNCs. The advantage of this principle is that the probe signal is processed directly in the CNC. The measuring parameters and the results of the measurements are output extremely clearly in separate displays, which are either automatically deselected at the end of the cycle or can be acknowledged when starting the CNC.

The result of the workpiece measurement can either include automatic offset of the zero point or a correction of the tool wear by the difference between the actual value and the setpoint. The measurement results can be logged in a file.

Siemens measuring cycles offer a standard log, which can be freely configured by the user.

In order to measure tool and workpiece dimensions, a touch-trigger probe is required that supplies a constant signal (rather than a pulse) when deflected. The probe should switch without any bounce, and mechanical adjustment may be necessary. Multidirectional probes can be used for all tool and workpiece measurements on turning and milling machines. Bidirectional probes are treated like a mono probe for workpiece measurements on milling and machining centers, but are not suitable for tool measurements.

Monodirectional probes can be used on milling machines and machining centers with slight restrictions in workpiece measurements, but are not suitable for tool measurements or for workpiece measurements on turning machines.

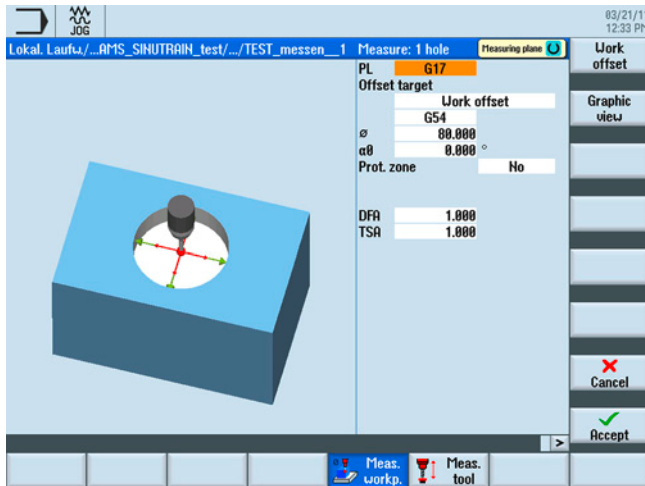
Milling measurement:

- Corner/right-angled corner/any corner
- Calibrate probe: length/radius in ring/radius at edge/calibrate on sphere
- Hole/rectangular pocket
- Hole/inner circle segment
- Edge distance – slot/web
- Edge distance – set edge
- Edge distance – align edge
- Circular/rectangular spigot
- Spigot/outer circle segment
- Measure 3D – align plane
- Measure 3D – 1 sphere/3 spheres
- Measure tools
- Measure tools – calibrate tool probe

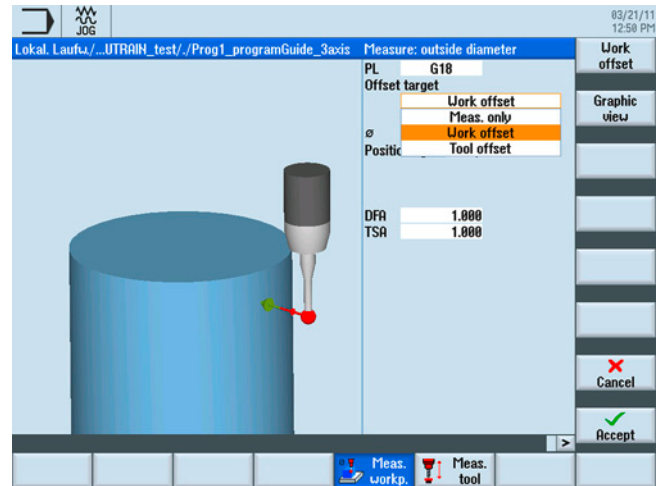
Turning measurement:

- Calibrate probe – length/radius at surface/radius in slot
- Inner/outer diameter
- Measure tools – turning tools/milling tools/drills
- Measure tools – calibrate tool probe

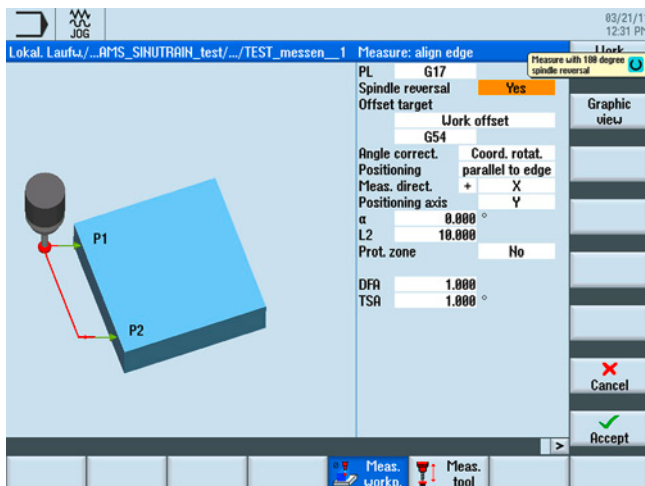
The use of high-precision probes such as those from Renishaw's Rengage range is recommended.



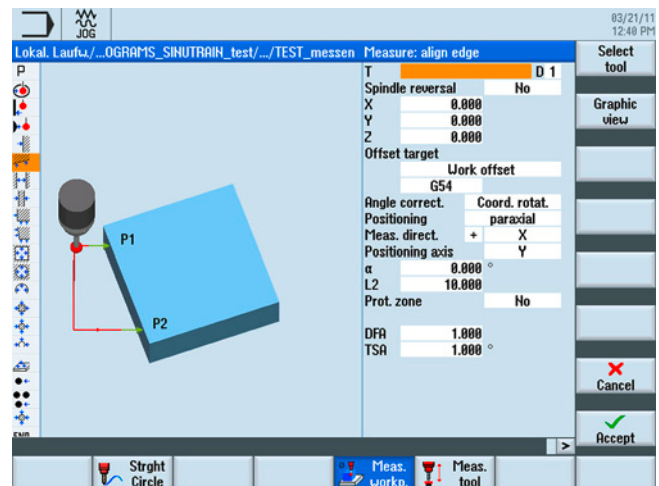
Example: Measuring a hole (programGUIDE)



Example: Measuring an outer diameter (programGUIDE)



Example: Align edge (programGUIDE)



Example: Align edge (ShopMill)

Measuring functions/measuring cycles

Measuring functions can execute either channel-specifically including all axes programmed in the measuring block as well as axially from the part program or from synchronized actions beyond CNC block limits. Measuring cycles are subprograms for implementing specific measuring tasks on tools or workpieces.

They are easy to use and simply require the input of values in predefined parameters.

- Measuring in JOG
- 2 probes can be connected simultaneously
- Measurement in space with frame
- Adaptable measuring modes with/without deletion of distance-to-go
- Display and logging of measuring parameters and results (with measuring cycles)
- Results can be read in the machine or workpiece coordinate system
- Cyclic measurement with synchronized actions parallel to workpiece machining

With the advantages of the measuring functions and measuring cycles, the machining accuracy is assured, unproductive time is reduced, sources of error are eliminated and further production processes are automated.

Glossary

Functions and terms

SINUMERIK 840D sl

Measuring inputs, expanded (16) for axial measurements, Run MyCC /PROX

Option; order code N57

Article No.: 6FC5800-0AN57-0YB0

SINUMERIK Integrate Run MyCC /PROX is a specific function and is not intended for general use.

Restricted functionality of export versions:

Not possible.

Measuring stage 1

Up to 2 switching probes can be connected simultaneously to the CNC. In the case of channel-specific measuring, the measuring process for a CNC channel is always activated from the part program running in the relevant channel. All of the axes programmed in the measuring block take part in the measuring process.

It is possible to program a trigger event (rising or falling edge) and a measuring mode with or without deletion of distance-to-go for each measuring process. The results of measurements can be read in the part program or with synchronized actions in both the machine and the workpiece coordinate system. You can test the deflection of the probe by scanning a variable and outputting it to the PLC interface and deriving responses in the part program.

Measuring stage 2 (option, order code M32) provides expanded functionality (for example for axial measuring, evaluation of up to 4 trigger events, and cyclic measuring).

Measuring stage 2

Option; order code M32

Article No.: 6FC5800-0AM32-0YB0

While the measuring function in motion blocks in the part program is limited to one block, you can activate measuring functions from synchronized actions at any time, independent of the part 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 3 parameters: 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 logged in the form of a file. The measuring cycles (option, order code P28) contain a standard log, which can be configured as the user wishes.

Measuring system error compensation

→ [Leadscrew error compensation/measuring system error compensation](#)

Measuring systems 1 and 2, selectable

For special applications, 2 encoders can be assigned to one axis, such as for a direct measuring system for the machining process with high demands on accuracy, and an indirect measuring system for high-speed positioning tasks. The switchover between measuring systems 1 and 2 is performed via the PLC.

Metal spinning protection area, Run MyCC /MSPZ

Option; order code N42

Article No.: 6FC5800-0AN42-0YB0

SINUMERIK Integrate Run MyCC /MSPZ is a specific function and is not intended for general use.

Restricted functionality of export versions:

Not possible.

Metric/inch dimensions

Depending on the measuring system used in the production drawing, it is possible to program workpiece-related geometrical data in either metric units (G71) or inches (G70). The CNC is set to a basic system regardless of the dimension system that has been programmed.

You can enter the following geometrical data directly and let the CNC convert them into the other dimension system (examples):

- Position data X, Y, Z, etc.
- Interpolation parameters I, J, K and circle radius CR
- Pitch
- Programmable work 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 operating area, the operator can switch back and forth between inch and metric dimensions using a softkey.

Milling technology package SINUMERIK MDynamics 3 axes

Option; order code S32

Article No.: 6FC5800-0AS32-0YB0

SINUMERIK MDynamics 3 axes contains the options:

- ShopTurn/ShopMill
- Residual material detection and machining for contour pockets and stock removal
- 3D simulation 1 (finished part)
- Simultaneous recording (real-time simulation of the current machining operation)
- Advanced Surface
- Spline interpolation (A, B and C splines)
- TRANSMIT/cylinder surface transformation
- Measuring cycles

Milling technology package SINUMERIK MDynamics 5 axes

Option; order code S33

Article No.: 6FC5800-0AS33-0YB0

SINUMERIK MDynamics 5 axes contains the options:

- ShopTurn/ShopMill
- Residual material detection and machining for contour pockets and stock removal
- 3D simulation 1 (finished part)
- Simultaneous recording (real-time simulation of the current machining operation)
- Advanced Surface
- Spline interpolation (A, B and C splines)
- TRANSMIT/cylinder surface transformation
- Measuring cycles
- Machining package 5 axes
- 3D tool radius compensation
- Measure kinematics

Restricted functionality of export versions:
Not possible.

Mode group

(Basic configuration: 1 mode group)

Option; order codes C01 ... C09

Article No.: 6FC5800-0AC00-0YB0

A mode group combines CNC channels with axes and spindles to form a machining unit. A mode group contains channels that must always operate in the same CNC operating 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. The number can be increased to include up to 10 mode groups using the option.

Glossary

Functions and terms

SINUMERIK 840D sl

Monitoring for maximum tool speed/acceleration

Option; order code S08

Article No.: 6FC5800-0AS08-0YB0

Two new parameters within the tool data can be used to specify the maximum speed and the maximum acceleration of a tool. If the tool is in the spindle, the speed or the acceleration of the spindle is limited to these values.

The limits can be separately activated and set. The speed is limited to the permissible value, even if the specified setpoint speed were to be exceeded by > 100 % by using the override switch.

Monitoring functions

The CNCs have integrated monitoring functions that are always active. These monitoring functions detect faults in the CNC, PLC or machine early on to prevent damage to the workpiece, tool or machine. When a fault occurs, machine operation is interrupted and the drives brought to a standstill. The cause of the fault is saved and displayed as an alarm. At the same time, the PLC is notified that a CNC alarm has been triggered.

Monitoring functions exist for the following areas:

- Read in
- Format
- Encoder and drive
- Contour
- Position
- Standstill
- Clamping
- Speed setpoint
- Actual velocity
- Enabling signals
- Voltage
- Temperatures
- Microprocessors
- Serial interfaces
- Transfer between CNC and PLC
- Backup battery voltage
- System memory and user memory

Motion-synchronous actions

→ Synchronized actions

Multi-axis interpolation (> 4 interpolating axes)

Option; order code M15

Article No.: 6FC5800-0AM15-0YB0

The number of interpolating axes can be expanded and is limited by option and machine data as well as by the number of axes available in the channel.

Restricted functionality of export versions:
Not possible.

Multi-axis package

Option; order code M10

Article No.: 6FC5800-0AM10-0YB0

The multi-axis package axes/spindles and channels is a low-cost package for multi-axis machines. It includes all axes/spindles/positioning axes/auxiliary spindles and channels.

Multi-channel display

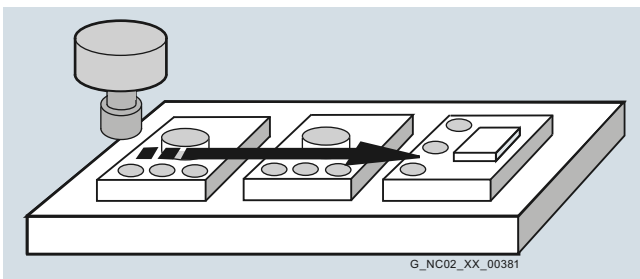
In the machine operating area, the M key can be used to select either single-channel or multi-channel display. In the multi-channel display, only channel information is displayed; the channel can be operated or influenced in the single-channel display. Focus switching, scroll bars and window selection can be operated in the multi-channel display, but it is not possible to change the CNC channel data. The same windows are always displayed together in all channels.

The softkeys for switching the windows, therefore, always affect all the channels that are on display. In the multi-channel display, the axis actual values are displayed in the top window and the selection menus (T/F/S values, program blocks) in the bottom window, depending on which of the softkeys is activated.

Multiple clamping of identical/different workpieces

Several identical workpieces can be clamped onto the machine table. With the multiple clamping function, an entire program is generated from the graphic program of the relevant single machining operation. The machining steps are sorted in this program so that the number of tool changes (and thus idle times) is reduced to a minimum.

This function allows identical and different workpieces to be finished on multiple vises or gripping yokes in a time-saving process.



→ ShopTurn/ShopMill

Multiple feedrates in one block

Depending on external digital and/or analog CNC inputs, this function can be used 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 a 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 time during grinding) lead to deletion of the distance-to-go.

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

Glossary

Functions and terms

SINUMERIK 840D sl

Nodding compensation, Run MyCC /NOCO

Option; order code N63

Article No.: 6FC5800-0AN63-0YB0

SINUMERIK Integrate Run MyCC /NOCO allows workpieces to be machined with greater precision by compensating for mechanical crosstalk effects between axes. SINUMERIK Integrate Run MyCC /NOCO is a useful function for column milling machines, for example, when the acceleration and braking of other axes can lead to "nodding" motion of the tower, causing the tool tip to make marks in the material as it is lowered into the workpiece. The nodding compensation function monitors the other axes and models the nodding behavior. Since the acceleration levels of all axes are known, the effects of nodding motion can be compensated by feedforward control of the other axes. Compensation thus increases machining precision and improves surface quality.

Restricted functionality of export versions:
Not possible.

Number of subprogram passes

In order to execute a subprogram several times in succession, the desired number of program repetitions can be programmed in the block with the subprogram call at address P: value range 1 to 9999. Parameters are transferred only when the program is called or in the first pass.

The parameters remain the same for all repetitions. If you want to change the parameters between passes, you should make the relevant declarations in the subprogram.

Online ISO dialect Interpreter

In general, part programs for SINUMERIK CNCs are programmed according to DIN 66025 and relevant expansions. Part programs created according to the ISO standard (e.g. G codes from other manufacturers) can be read in, edited and executed on SINUMERIK CNCs using the online ISO dialect interpreter.

Operate robot, Run MyRobot

Option; order code P74

Article No.: 6FC5800-0AP74-0YB0

SINUMERIK Integrate Run MyRobot includes the options:

- 1 additional machining channel
- Run MyCC /EXPD
- Run MyHMI /3GL
- Run MyScreens

The runtime license SINUMERIK Integrate Run MyRobot allows integration of KUKA robots into the SINUMERIK 840D sl. Robots can then be operated, programmed and diagnosed on the operator panel of the SINUMERIK 840D sl by users who have no specialist robot knowledge. Machine operating times are increased by this simple automation, or by the simple, flexible setup of machine tool and robot.

You can find an application example on the Internet by entering the search term: Run MyRobot.

→ [Machining channel](#)

→ [Run MyCC /EXPD](#)

→ [Use HMI applications, Run MyHMI /3GL](#)

→ [Use HMI applications, Run MyScreens](#)

→ www.siemens.com/automation/support

Restricted functionality of export versions:
Not possible.

Operating modes

→ [CNC operating modes](#)

Operating software languages

Option; order code N00

Article No.: 6FC5800-0AN00-0YB0

The operating software supports the following languages as standard for user interface display texts: English, French, German, Italian, Simplified Chinese, and Spanish.

Further additional languages can be ordered with the option, order code N00.

Glossary

Functions and terms

SINUMERIK 840D sl

Operator panel management

The SINUMERIK Thin Client Unit TCU for distributed installation allows the spatial separation of SINUMERIK OP/TP operator panel fronts and SINUMERIK PCUs, as well as the connection of up to 4 operator panel fronts to a PCU with one TCU each. To this end, the user interface of a PCU 50.5 is copied to several OPs/TPs with one TCU each.

Benefits:

- Low-vibration installation of the PCU in the control cabinet
- Effective operation of larger machines using up to 5 identical operator panels
- Signal transmission between PCU and operator panel front via Industrial Ethernet
- Operation at the active operator panel with the option to enable the passive operator panel on request
- Mixed operation of operator panel fronts on a TCU or with an integrated TCU and an operator panel front directly connected to the PCU possible
- Distance between PCU and operator panel fronts of up to 100 m (max. distance between two network nodes).

	NCU CNC SW	PCU SINUMERIK Operate
Operator panel management		
2/4 TCUs with interlocking of simultaneous operation using the Veto mode and PLC (4 × T : 1 × M)	✓ (2/4)	✓ (4)
TCU suppression with more than 2/4 TCUs (n × T : 1 × M)	✓	✓
External HMIs (n × M : n × N)	–	✓ (1 × M: 4 × N)
One external HMI which can be switched over via several NCUs (1 × M : n × N)	–	✓ (4 × N)
One internal and one external HMI connected simultaneously to one NCU	✓ TM	✓ TM
Display		
Down-zoom	–	–
Dynamic resolution switchover	✓	✓
Activate/deactivate MCP		
MCP PROFIBUS	–	–
MCP IE (IE mode)	✓	✓
MCP PN (PN mode)	–	–
One or several TCUs, which can be switched over via several NCUs and PCUs (1 × T : n × M)		
Using 2-key operation	✓	✓
Using the channel menu	✓	✓
Options for multi-user operation (n × T : M : N) T = Number of TCUs M = Number of HMIs N = Number of NCUs ✓ = Possible – = Not possible		

Optimized friction compensation

Option; order code S06

Article No.: 6FC5800-0AS06-0YB0

This option is designed to achieve optimum friction compensation. The required setting parameters are calculated automatically by a cycle.

Oscillation functions

Option; order code M34

Article No.: 6FC5800-0AM34-0YB0

With this function, an axis oscillates at the programmed feedrate between 2 reversal points. A possible application is a grinding machine.

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 guide axis for gantry or coupled-motion axes.

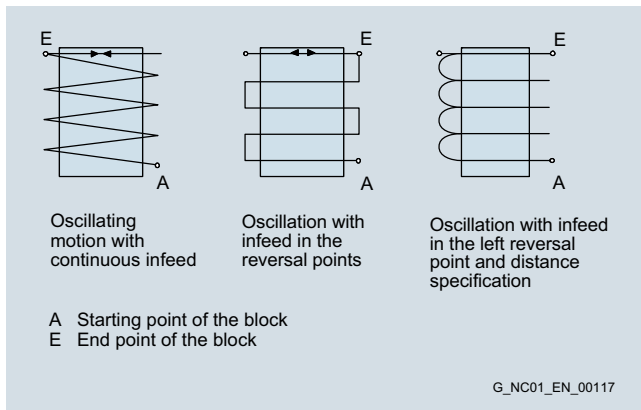
Block-related oscillation

- 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.
- 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 limit (soft reversal)
 - After reaching the programmed position or
 - After reaching the programmed position and expiration of the dwell time
- The following manipulations are possible:
 - Cancellation of the reciprocating movement and infeed by deletion of distance-to-go
 - Modification of the reversal points via CNC program, PLC, handwheel or direction keys
 - Manipulation of the reciprocating axis feedrate via CNC program, PLC or override
 - Control of the reciprocating movement via the PLC

The spindles can also perform reciprocating movement.



Oscillation functions

Glossary

Functions and terms

SINUMERIK 840D sl

Pair of synchronous axes (gantry axes)

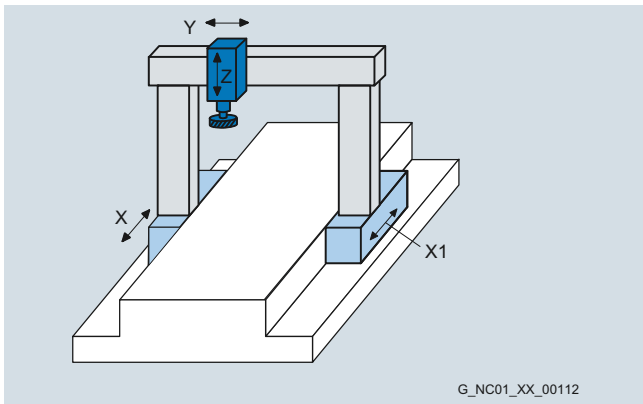
Option; order code M02

Article No.: 6FC5800-0AM02-0YB0

The gantry axes function can be used to traverse the axes of up to 3 pairs of mechanically-coupled axes simultaneously without mechanical offset. The actual values are continuously compared and even the smallest deviations corrected.

During both operation and programming, the axes defined in a gantry grouping are treated like machine axes.

A gantry grouping consists of a guide axis and up to 2 synchronous axes. 2 guide axes can be coupled using curve table interpolation.



Gantry axes (pair of synchronous axes X/X1)

Path length evaluation

Option; order code M53

Article No.: 6FC5800-0AM53-0YB0

With path length evaluation, data in the CNC can be buffered so that conclusions can be drawn with respect to the maintenance state of the machine.

In the first stage, the following data are acquired:

- Total travel path for each axis
- Total travel time for each axis
- Number of traversing actions per axis (stop - traverse - stop)
- Total sum of jerks per axis

This data is stored in the SRAM and is not affected by power on/off. As a consequence, using an external service, consistent data can be achieved for the complete life cycle of a machine. This data can also be read through system variables in the part program and in synchronized actions.

→ Synchronized actions

Path velocity-dependent analog output (laser power control)

Option; order code M37

Article No.: 6FC5800-0AM37-0YB0

Using the path-velocity-dependent analog output, the actual path velocity can be output in the interpolation cycle via a SIMATIC DP ET 200 analog module. The function is programmed via synchronized actions.

One application is laser power control.

Path-related pulse output, Run MyCC /PRIG

Option; order code N76

Article No.: 6FC5800-0AN76-0YB0

SINUMERIK Integrate Run MyCC /PRIG is used in the non-destructive testing of composite components using ultrasound transmission. The number of pulses of an output signal (max. 256 kHz) supplies precise information about the traversed path. It is possible to program the required distance per pulse.

The pulses are output via connector X520 of the SINAMICS TM41 technology module.

Plain text display of user variables

In addition to the predefined variables, programmers can define their own variables and assign values to them.

The variables are displayed in plain text format, e.g.

Definition: DEF INT NUMBER – Display: NUMBER

Definition: DEF REAL DEPTH – Display: DEPTH.

→ [CNC high-level language](#)

Plastics package IME

Option; order code S40

Article No.: 6FC5800-0AS40-0YB0

Plastic parts can be produced quickly and easily with the support of the IME plastics package.

The option includes:

- 3 additional axes as option, order code A03
- Travel to fixed stop as option, order code M01
- Pair of synchronized axes (gantry axes) as option, order code M02
- Master/slave for drives as option, order code M03
- Position switching signals/cam controller as option, order code M07
- Polynomial interpolation as option, order code M18
- Transformation: Handling as option, order code M31
- Synchronized actions stage 2 as option, order code M36
- No spindle function
- No tool offsets

Restricted functionality of export versions:
Not possible.

Glossary

Functions and terms

SINUMERIK 840D sl

PLC

SIMATIC STEP 7

The PLC on the SINUMERIK is programmed using the user-friendly SIMATIC STEP 7 software.

The STEP 7 programming software is based on the Windows operating system and makes it convenient and easy for the user to utilize the full performance of the PLC. The statement list STL, function block diagram FBD and ladder diagram LAD programming languages are available. The user can switch from one to the other using STEP 7 pull-down menus.

The following blocks are available for structured programming:

- Organization blocks OB
- Function blocks FB and function calls FC
- Data blocks DB

In addition, system function blocks SFB and system functions SFC integrated in the operating system can also be called. The STEP 7 software package for SIMATIC S7-300 is a standard component of SIMATIC programming devices, 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 Graph or Structured Control Language SCL.

NCK/PLC interface

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

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

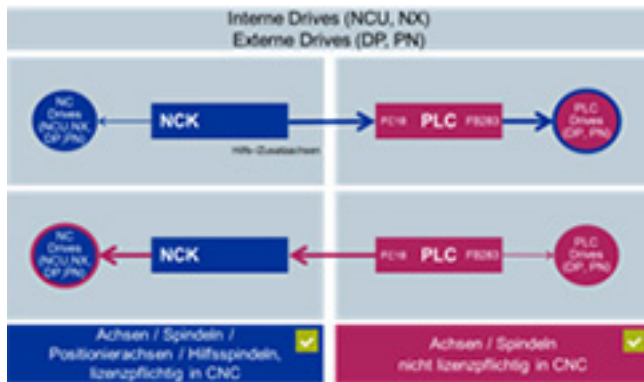
The PLC basic program, which is part of the toolbox, organizes the exchange of signals and data between the PLC user program and the NCK, PCU and machine control panel areas.

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) and NCK status information, e.g. program running. The basic program carries out cyclic signal transfer 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.
- Event-driven signal exchange NCK → PLC
PLC functions that have to be executed as a function of the workpiece program are triggered by 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 workpiece 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, yet reliable. It is therefore alarm and acknowledgment-controlled. The basic program evaluates the signals and data, sends an acknowledgment to the NCK, transfers some of the data to OB40 and the rest to the user interface at the beginning of the cycle. CNC machining is not affected if the data do not require an acknowledgment from the user.
- Event-driven signal exchange PLC → NCK
An event driven signal exchange PLC → NCK takes place whenever the PLC transfers a request to the NCK (e.g. traversing an auxiliary axis). Here again, the data transfer is acknowledgment-controlled. This type of signal transfer is initiated by the user program via an FB or FC. The associated FBs and FCs are provided together with the basic program.
- Messages
User messages are acquired and conditioned by the basic program. The message signals are transferred 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 message events occurs. If an OP is available, the messages are transferred to the OP and displayed on it.

PLC axes

PLC axes/spindles (PLC drives) are controlled by the PLC. **No** license is required for these axes/spindles in the CNC. NCK drives are controlled by the NCK. A license is required for these axes/spindles/positioning axes/auxiliary spindles in the CNC.



- Axis/spindle
- Simulation axis/spindle
- Positioning axis/auxiliary spindle
- Virtual axis
- Internal/external drives

PLC status

The user can check and modify PLC status signals in the Diagnostics operating area via the operator panel. This can be done locally without a programming device:

- Check the input and output signals from the PLC's I/Os
- Carry out limited troubleshooting
- Check the NCK/PLC and PCU/PLC interface signals for diagnostic purposes

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

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

The status of the above signals can be changed for tests. Signal combinations are also possible, and up to 10 addresses 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 segmented according to load memory, work memory and system memory.

Load memory is retentive, and takes the form of either integrated RAM or a RAM module (plug-in memory card). It contains data and program and decompiling information. The load memory and the high-speed work memory for execution-relevant program tests provide sufficient space for user programs.

PLC user memory, expanded

Option; order codes D11 ... D18

Article No.: 6FC5800-0AD10-0YB0

The PLC user memory can be expanded by 128 KB in each case with this option.

Polar coordinates

By 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 an absolute dimension or incremental dimension.

Glossary

Functions and terms

SINUMERIK 840D sl

Polynomial interpolation

Option; order code M18

Article No.: 6FC5800-0AM18-0YB0

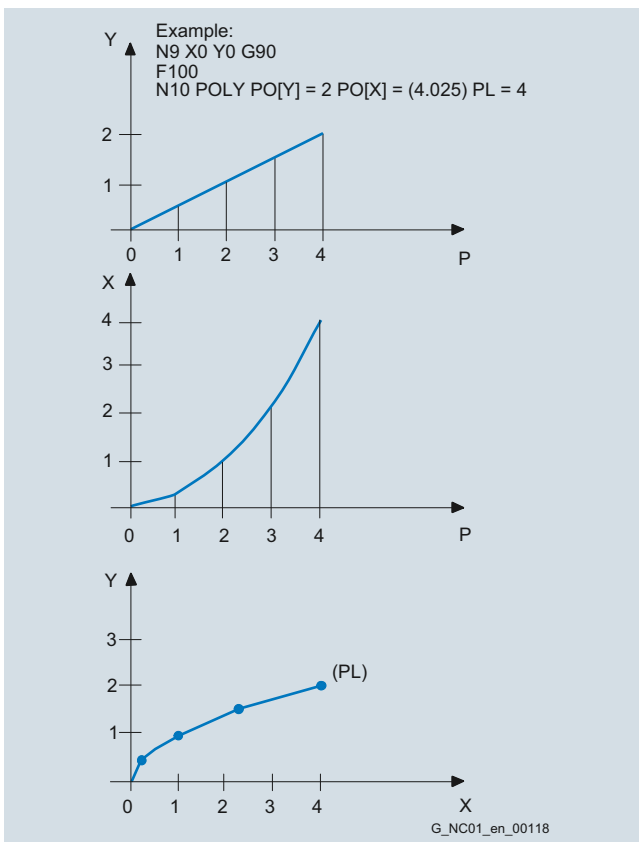
Curves can be interpolated using polynomial interpolation, whereby the CNC axes follow the function:

$$f(p) = a_0 + a_1p + a_2p^2 + a_3p^3 + a_4p^4 + a_5p^5 \text{ (max. 5th degree polynomial)}$$

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 curve characteristics, such as straight line, parabolic and exponential functions.

Polynomial interpolation primarily serves as an interface for programming externally generated spline curves. 5th degree polynomials can be optimally 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. Tool radius compensation can be used as in linear and circular interpolation.



Polynomial interpolation

Position monitoring

SINUMERIK CNCs provide extensive monitoring mechanisms for axis monitoring:

- Motion monitoring functions:
Contour monitoring, position monitoring, standstill monitoring, clamping monitoring, speed setpoint monitoring, actual speed monitoring, encoder monitoring
- Static limit monitoring:
Limit switch monitoring, working area limitation

Position monitoring is always activated after termination of motion blocks according to the setpoint. To ensure that an axis is in position within a specified period of time, the timer configured 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 fell below 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), position monitoring is deactivated and replaced by standstill monitoring. Position monitoring is effective for linear and rotary axes as well as for position-controlled spindles. Position monitoring is not active in the follow-up mode.

Position switching signals/cam controller

Option; order code M07

Article No.: 6FC5800-0AM07-0YB0

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

Although position switching signals are output in the IPO cycle, they can also be output as switching outputs in the position control cycle using the high-speed CNC inputs/outputs function.

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

→ [High-speed CNC inputs/outputs](#)

Positioning axis/auxiliary spindle

Option; order codes B01 ... B28

Article No.: 6FC5800-0AB00-0YB0

Positioning axes can execute movements simultaneously with machining, thus reducing non-productive times considerably. They can be used to advantage when controlling workpiece and tool feeders or tool magazines. They can be programmed with an axis-specific feedrate in the part program.

Axis movement beyond block boundaries is also possible. Positioning axes can also be controlled by the PLC. This means that axis movements can be started independently of the part program without requiring an additional machining channel.

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

Possible functions:

- POS/SPOS/M3, M4, M5 (from CNC block)
- POSA/SPOSA (from CNC block, modally)
- FC18/POS/SPOS/M3, M4, M5 (PLC axes)
- PLC-VDI interface (M3, M4, M5 directly)
- OSCILL (asynchronous oscillation)
- OSCILL (synchronous oscillation)
- do POS/SPOS/M3, M4, M5 (synchronized actions)
- Couplings (TRAIL, gantry and master/slave (positioning axis as slave axis), LEAD, EG, CP, etc.)
- Setpoint and actual value available
- Commissioning with SINUMERIK Operate

Functions that are not possible:

- Path axes/geometry axes/additional path axis/GEOAX()
- Spindles for thread cutting, tapping and thread cutting with compensating chuck
- An additional positioning axis/auxiliary spindle cannot interpolate with other axes.

→ [Axis/spindle](#)

→ [Simulation axis/spindle](#)

→ [Virtual axis](#)

→ [PLC axes](#)

Positioning axes/spindles via synchronized actions

Axes/spindles can be positioned depending on conditions (e.g. the actual values of other axes, high-speed inputs) with a special feedrate or speed to a specific setpoint via synchronized actions. Synchronized actions are executed in the interpolation cycle, are carried out in parallel with the actual workpiece 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 directly from 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 depending on input signals without PLC intervention.

Glossary

Functions and terms

SINUMERIK 840D sl

Preprocessing buffer, dynamic (FIFO)

The traversing blocks are prepared prior to execution and stored in a preprocessing memory (FIFO = first in/first out), whose size can be parameterized. In contour sections that are machined at high velocities with short path lengths, blocks can be executed from this preprocessing memory at very high speed.

The preprocessing buffer is constantly reloaded during execution. Block execution can be interrupted with the STARTFIFO command until the preprocessing memory has been filled, or STOPFIFO (start high-speed machining section) or STOPPRE (stop preprocessor) can be programmed.

Preset

The Preset function can be used to redefine the zero in the machine coordinate system of the CNC. The preset values act on machine axes. Axes do not move when Preset is active, but a new position value is entered for the current axis positions. Once the actual values have been reset, a new reference point approach is required before protection areas and software limit switches can be reactivated.

PROFIBUS tool and process monitoring, Run MyCC /TPM-PB

Option; order code M62

Article No.: 6FC5800-0AM62-0YB0

Identifying errors before they happen. This is the motto for our SINUMERIK CNCs, which allow you to implement tool and process monitoring. Active power monitoring can identify issues such as breakage, wear and missing tools. Precise operating status recognition and process optimization are also possible.

Using the PROFIBUS tool and process monitoring function, the digital drive data for torque, active power and actual current are directly transferred to a special PROFIBUS slave module via the PROFIBUS DP interface.

This hardware and the software for evaluation is offered, for example, by SINUMERIK Solution Partners.

PROFINET

PROFINET helps businesses to operate more successfully. Thanks to its outstanding flexibility, it gives you greater freedom to implement tailored machine and plant concepts. Its high-level efficiency allows you to make optimum use of available resources. High precision and product quality are assured by the unique performance of this Industrial Ethernet standard. You benefit from a sustained increase in your productivity.

PROFINET is the open Industrial Ethernet standard of PROFIBUS International for automation systems. PROFINET is based on Industrial Ethernet and uses TCP/IP and IT standards.

Two versions are available:

- PROFINET CBA (Component Based Automation) for networking distributed plants, component engineering
- PROFINET IO (Input Output) for controlling drives, sensors and actuators using one or several central controllers

PROFINET is supported by PROFIBUS International and has been included in standards IEC 61158 and IEC 61784 since 2003.

PROFINET includes:

- A multi-level real-time concept
- Simple field devices, which operate IOs directly on Ethernet
- Design of modular systems with a high degree of reusability
- Simple integration of existing PROFIBUS or Interbus systems

→ www.siemens.com/profinet

PROFINET CBA

PROFINET CBA is an automation system for plants with distributed intelligence. The key attributes of this model, inspired by standard IEC 61499, include:

- Structuring of logical plant sections into clearly organized subunits and their re-usability
- Clearly defined engineering of the plant
- Seamless integration of existing field bus systems
- Ethernet-based communication

Therefore, a PROFINET CBA system always comprises a variety of intelligent automation devices (components). A component includes all mechanical, electrical and IT variables (PLC program). Each individual component is created with its own manufacturer-specific, standard programming tools. A graphic, non-proprietary component connection editor (iMAP) is available for linking individual components to a higher-level system, i.e. for engineering the system as a whole.

In this context, "engineering" means:

- Configuring the system
- Defining the exchange of data
- Loading configuring data to the components

A standardized PROFINET Component Description (PCD) file is created in XML to describe a component. The component connection editor loads these descriptions and uses them to set up logical connections between individual components.

A PROFINET component always consists of:

- Exactly one Physical Device PDev, with MAC and IP address
- One or several logical devices LDev
- One ACCO per LDEV
- One or several runtime automation objects per LDEV (RT-Auto)

The Active Control Connection Object ACCO functions as a consumer and provider and is the heart of the communication system. The RT-Auto includes the technological functionality, i.e. the executable program.

PROFINET IO

A PROFINET IO system comprises the following devices:

- An IO controller is an automation control system, typically a PLC, CNC, robot control or motion controller. When compared to PROFIBUS, an IO controller is a master.
- An IO device is a distributed field device which is linked via PROFINET IO. It is controlled by an IO controller. An IO device can consist of several modules and submodules. All data to be exchanged are assigned slots and subslots for the purpose of addressing. These are defined in the General Station Description (GSD) file. ET200 distributed I/O or a SINAMICS drive are examples of PROFINET IO devices. When compared to PROFIBUS, an IO device is a slave.
- An IO Supervisor is typically a programming device, a PC or an HMI unit for commissioning or diagnostics. It features an engineering tool, which can be used to parameterize and diagnose individual IO devices. When compared to PROFINET, this would be a class 2 master in terms of function.

PROFINET IO provides protocol definitions for the following functions:

- Cyclical transmission of IO data
- Acyclical transmission of alarms that require acknowledgement
- Acyclical transmission of data (parameters, detailed diagnostic information, commissioning data, I&M data)

An application relation (AR) is formed between an IO controller and an IO device. The communication relationships, diagnostic options and potential useful data exchange are determined by the communication view. Communication relationships (CR) with varying properties are specified for the transfer of parameters, cyclic data communication and alarm handling based on this AR.

Communication channels are set up to handle the data exchange between each IO controller and the IO device. It is possible to form more than one application relationship between different devices. Isochronous drive controls can be implemented with PROFINET IO and the PROFIdrive profile for motion control applications.

In the GSD file, the device manufacturer must exactly describe how the device functions are specifically mapped to the PROFINET IO model, i.e. the properties of the IO device. GSDML (GSD Markup Language), an XML-based language, is used for this purpose. The GSD file is read in by the engineering tool and forms the basis for planning the configuration of a PROFINET IO system.

Glossary

Functions and terms

SINUMERIK 840D sI

Program preprocessing

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

In particular in the case of programs containing sections written in a high-level language and in the case of calculation-intensive programs, e.g. programs containing control structures, motion-synchronous actions or cutting cycles, execution times can be reduced by up to 1/3.

Program screens, operating areas and user interfaces, Create MyHMI

With Create MyHMI we are offering scalable HMI openness for SINUMERIK Operate and the option to add programming or configuring in high-level languages. Create MyHMI is available for the different programming languages currently in use (Qt/C++, .net).

The user benefits from the availability of specific operating screens in SINUMERIK Operate and is therefore able to create customized user interfaces thanks to the unique openness of the SINUMERIK system.

→ [Program screens, operating areas and user interfaces, Create MyHMI /3GL](#)

→ [Use HMI applications, Run MyHMI /3GL](#)

Program screens, operating areas and user interfaces, Create MyHMI /3GL

The SINUMERIK Integrate Create MyHMI /3GL programming package allows machine manufacturers to design their own user interfaces that contain either machine-manufacturer or end-user functional expansions or their own screen form layouts.

This function is realized in a high-level language development environment based on Qt/C++ that can generate a platform-independent execution code for Windows 7 (SINUMERIK PCU 50 or enabled IPC) and Linux (SINUMERIK NCU). The screen forms are created independent of the particular platform in the development environment Visual Studio. The compiled program parts are transferred to the corresponding user directories of the PCU 50 or NCU.

Furthermore, specific user interfaces can be generated on Windows platforms via a .net or C++ communication interface.

Programming examples for new screen forms, which can also be used as the basis for the user's own screen forms, can be found on the product DVD of the SINUMERIK Integrate Create MyHMI /3GL programming package.

Listed below are some of the functions that can be implemented with the SINUMERIK Integrate Create MyHMI /3GL programming package:

- Display screen forms and provide softkeys, variables, tables, texts, help texts, graphics, and help screens
- Start actions when screen forms are displayed and exited, when softkeys are pressed, and values (variables) are entered
- Dynamic restructuring of screen forms, including changing softkeys, designing arrays, displaying, replacing and deleting display texts and graphics
- Read and write variables, combine with mathematical, comparative or logical operators
- Execute subprograms, file functions, program instance services (PI services) or external functions (SINUMERIK Operate)
- Enable data exchange between screen forms

Use of the Create MyHMI /3GL package requires an OEM agreement.

The SINUMERIK Integrate Run MyHMI /3GL runtime license is required to run the programmed user screens.

→ [Use HMI applications, Run MyHMI /3GL](#)

Program screens, operating areas and user interfaces, Run MyHMI /WinCC

Option; order code P61

Article No.: 6FC5800-0AP61-0YB0

SINUMERIK Integrate Run MyHMI /WinCC permits an operating area to be added to SINUMERIK Operate on a PCU/PC. This additional operating area can be simply created using the SIMATIC WinCC ES Advanced engineering system and the add-on SINUMERIK Integrate Create MyHMI /WinCC.

programGUIDE

The programGUIDE, with animated elements and cursor text, provides perfect support for integrating cycles into part programs. Users can define their own softkeys, input fields and displays using SINUMERIK Integrate Run MyScreens.

→ [Use HMI applications, Run MyScreens](#)

Programmable acceleration

With the programmable acceleration function, it is possible to modify the axis acceleration in the program in order to limit mechanical vibration levels in critical program sections. The path or positioning axis is then accelerated at the programmed value. The acceleration value set in the machine data can be exceeded by up to 100 %.

This limitation is active in the 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 (combination of individual instructions).

Programming package

→ [Program screens, operating areas and user interfaces, Create MyHMI /3GL](#)

programSYNC

Option; order code P05

Article No.: 6FC5800-0AP05-0YB0

The programs are split into individual machining steps (blocks) in the dual editor which are then filled with G code or ShopTurn cycles. They provide the basis for multi-channel programming. These program sections can be expanded and collapsed, providing a program structure which is clearly organized.

The programSYNC function is an additional special feature: The individual blocks of the channels are time-synchronized by so-called wait markers. The programSYNC function synchronizes the machining steps with one another and an automatic time evaluation function enables the user to optimize the multi-channel program in the dual editor. In this way it is possible to transfer individual machining steps to other channels where required in order to create a time-optimized program.



Dual editor

Glossary

Functions and terms

SINUMERIK 840D sl

Protection areas 2D/3D

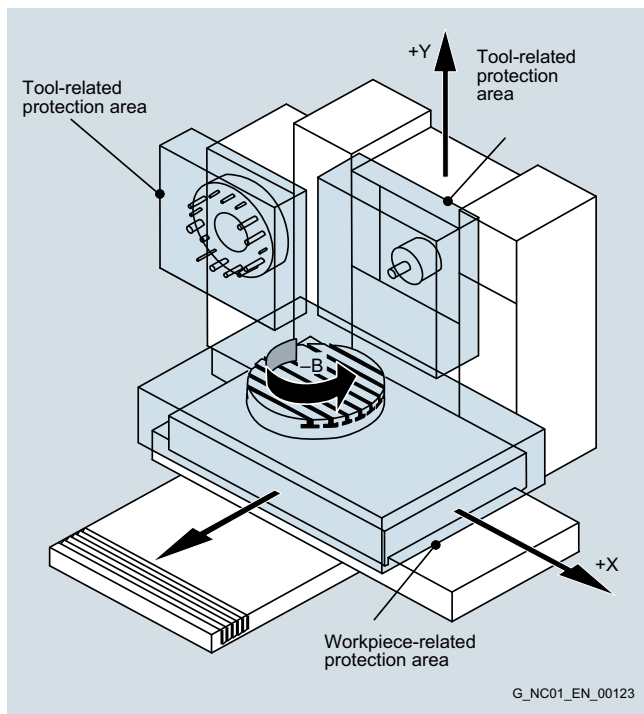
Protection areas can be used to protect various elements on the machine and its equipment, as well as the workpiece to be created, against incorrect movements.

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

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

For the elements to be protected, 2D or 3D protection areas are defined in the part program or via system variables. These protection areas can be activated and deactivated in the part program. Protection areas must always be divided into workpiece-related and tool-related areas. During machining in JOG, MDI or AUTO CNC operating mode, a check is always made to see whether the tool or its protection areas violate the protection areas of the workpiece.

Monitoring of the protection areas is channel-based, that is, all active protection areas for a channel are mutually monitored for collisions (protection areas not channel-specific with NCU system software for 2/6 axes). A maximum of ten protection areas and 10 contour elements are available for describing a protection area.



Protection areas

Punching/nibbling

Option; order code M33

Article No.: 6FC5800-0AM33-0YB0

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

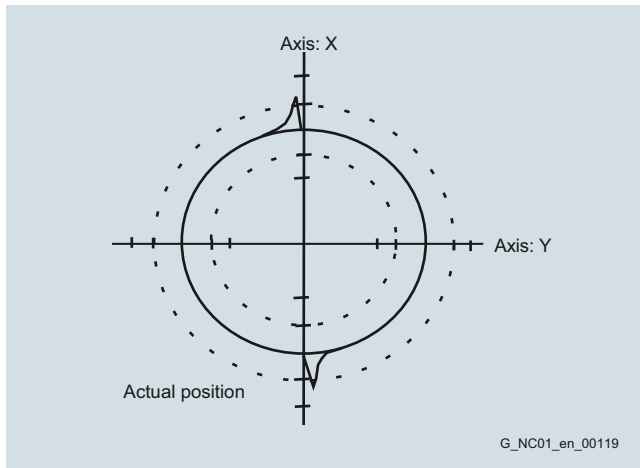
- Language commands
The punching/nibbling functions are activated and deactivated using simple, clear high-level language elements such as PON, SON, PONS, PDELAYON, and so on.
- Stroke control
CNC and punch are synchronized with one another via the high-speed signals that are input and output via the drive bus in the CNC's position control cycle, making it possible to attain high velocities and maximum precision.
- Automatic path segmentation
You can choose whether you want the CNC to break down the machining path automatically by stroke length SPP or stroke rate SPN. With SPP, the travel path is broken down into programmable segments of identical size (modal effect). SPN breaks the travel path down into a programmable number of path sections (non-modal effect).

Quadrant error compensation

Quadrant error compensation (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 path velocity while the second axis is stationary. The different friction conditions can cause contour errors.

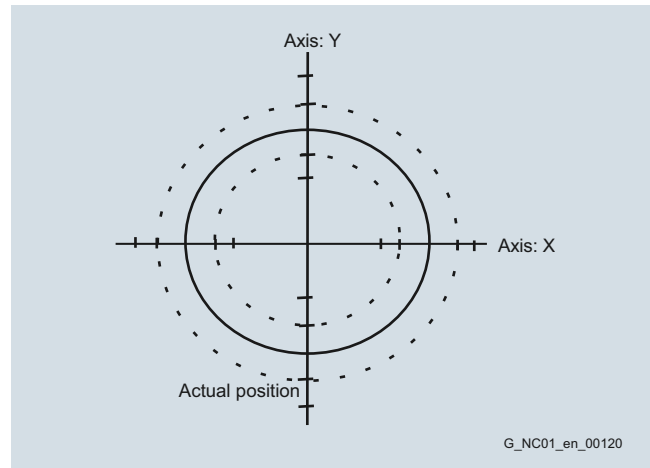
Quadrant error compensation virtually eliminates this problem and produces excellent results, without contour errors, in the very first machining operation. For operator-controlled quadrant error compensation, the intensity of the compensation pulse is set according to an acceleration-based characteristic. This characteristic is determined and parameterized when commissioning 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 recorded by measurement and graphically represented while the circular contour is being traversed.



Quadrant transitions without compensation

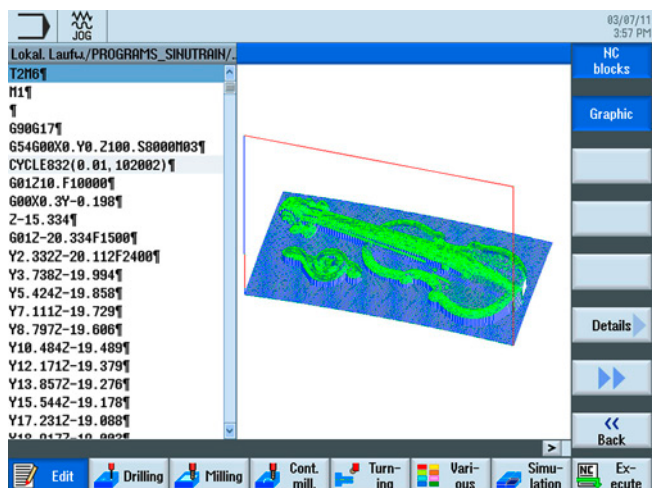
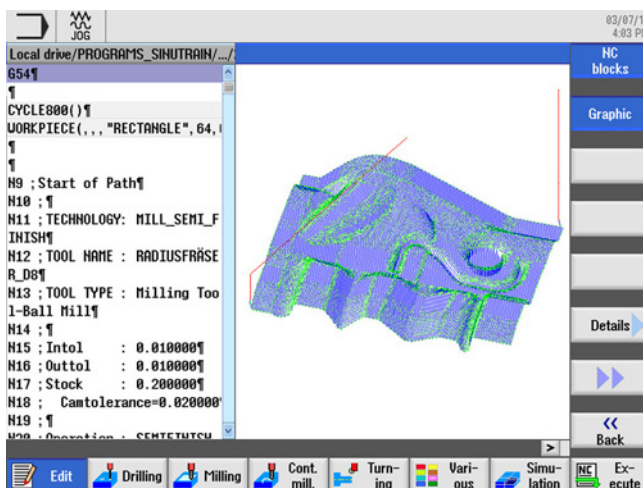
→ [Optimized friction compensation](#)



Quadrant transitions with quadrant error compensation

Quickview for mold-making programs

The high-speed 3D representation of part programs (G1 blocks) gives the user greater confidence in handling mold-making programs. The Quick Viewer for mold-making CNC blocks can be selected in the part program editor.



Quickview

→ [Quickview for mold making programs](#)

Glossary

Functions and terms

SINUMERIK 840D sl

Reading of actual positions correlated with output signal, Run MyCC /COPA

Option; order code N61

Article No.: 6FC5800-0AN61-0YB0

SINUMERIK Integrate Run MyCC /COPA is required to provide an interface to Renishaw's SPRINT measuring system. Renishaw's SPRINT measuring system continuously detects the position values of a touch probe during the path motion and combines these with encoder positions from the NC to supply machine-related measurements.

Reference point approach

When using a machine axis in the 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 at a defined velocity either using the direction keys, in a sequence that can be defined in the machine data, or automatically via program command G74.

If length measuring systems with distance-coded reference marks are used, reference point approach is shorter, as it is necessary to approach only the nearest reference mark. Reference point approach of an axis with absolute 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 the automatic mode (e.g. to take a measurement on the workpiece, correct tool wear values or because of tool breakage), the tool can be retracted from the contour manually after changing to the JOG CNC operating mode.

In such cases, the CNC stores the coordinates of the point of interruption and displays the path distances traversed by the axes in the JOG mode as a Repos offset in the actual value window.

The contour can be reapproached:

- In JOG CNC operating mode using the axis and direction keys. It is not possible to overshoot the interruption point; the feedrate override switch is effective.
- By the 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. Approach movements can be programmed as straight lines, in quadrants or in semicircles.

Residual material detection and machining for contour pockets and stock removal

Option; order code P13

Article No.: 6FC5800-0AP13-0YB0

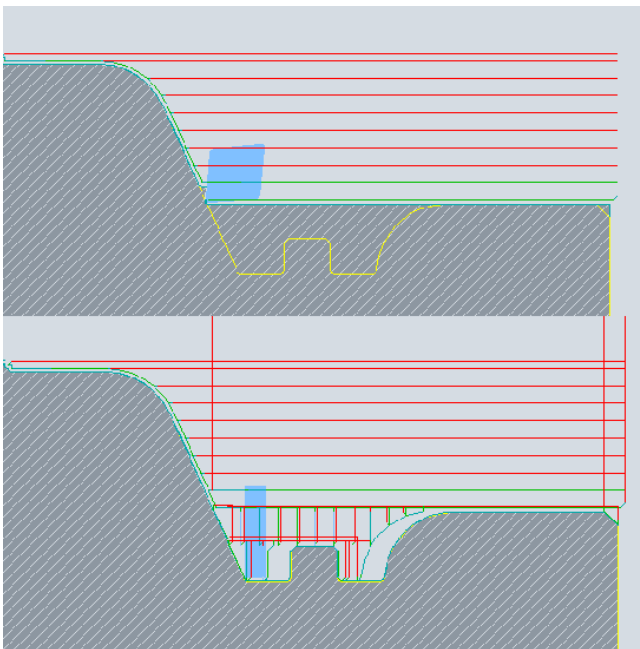
Contour ranges that cannot be machined with large tools are automatically recognized by the cycle for contour pockets or the stock removal cycle. The operator can rework these regions using a smaller tool.

Contour turning offers:

- Contour/axis-parallel cutting with residual material detection
- Contour cutting with residual material detection
- Plunge-turning with residual material detection

Contour milling offers:

- Contour spigot with residual material detection
- Contour pocket with residual material detection
- Machining, e.g. in the steps: centering, predrilling, rough machining and rough machining residual material, smoothing, edge/base, chamfering



Rotary axis, turning endlessly

Depending on the application, the working area of a rotary axis can be limited using software limit switches (e.g. working area between 0° and 60°), to a corresponding number of rotations (e.g. 1000°) - or it can be unlimited (endlessly turning in both directions).

This function can also be used with absolute encoders.

Run MyVNCK

With VNCK we are offering a virtual NC kernel which is implemented at the machine level with the machine OEM directly or in a CAM system. This offers machine OEMs the flexibility they need to implement customized machine applications.

Glossary

Functions and terms

SINUMERIK 840D sl

Safety Integrated

SINUMERIK Safety Integrated provides integrated safety functions that support the implementation of highly effective personnel and machine protection. These safety functions meet the requirements of EN 61508 for use up to and including SIL2 and Category 3, as well as PL d according to EN ISO 13849. This allows not only for simple and economic implementation of the main requirements for functional safety, but also the realization of practical operating and security concepts.

Available functions include, among others:

- Functions for safety monitoring of velocity and standstill
- Functions for establishing safe boundaries in work spaces and protected spaces, and for range recognition
- Direct connection of all safety-related signals and their internal logical linkage
- Safe brake management
- Safe communication via PROFIBUS or PROFINET
- Operator-controlled acceptance test – integrated in SINUMERIK Operate

A distinction is made between Safety Integrated (system-integrated safety functions with safe programmable logic) and Safety Integrated plus (drive-based safety functions and integrated fail-safe PLC).

Safety Integrated – SI axis/spindle, extra for each additional axis/spindle

Option; order codes C71 ... C78

Article No.: [6FC5800-0AC70-0YB0](#)

Enhanced motion monitoring functions for 1 axis/spindle.

Safety Integrated – SI axis/spindle package, 15 additional axes/spindles

Option; order codes C61 ... C62

Article No.: [6FC5800-0AC60-0YB0](#)

Enhanced motion monitoring functions for 15 axes/spindles.

Safety Integrated – SI-Basic

Option; order code M63

Article No.: [6FC5800-0AM63-0YB0](#)

SINUMERIK Safety Integrated for 1 NCU including 1 axis/spindle, 4 SPL inputs and 4 SPL outputs.

Safety Integrated – SI-Comfort

Option; order code M64

Article No.: [6FC5800-0AM64-0YB0](#)

SINUMERIK Safety Integrated for 1 NCU including 1 axis/spindle, 64 SPL inputs and 64 SPL outputs.

Safety Integrated – SI-Connect

Option; order code S67

Article No.: [6FC5800-0AS67-0YB0](#)

This option expands safe communication up to 16 connections.

Safety Integrated – SI-High Feature

Option; order code S68

Article No.: [6FC5800-0AS68-0YB0](#)

Requirement: NCU 720.xB PN/NCU 730.xB PN

SINUMERIK Safety Integrated for 1 NCU including 1 axis/spindle, 192 SPL inputs and 192 SPL outputs.

Safety Integrated plus – SI axis/spindle

Option; order codes K01 ... K09

Article No.: [6FC5800-0AK00-0YB0](#)

Enhanced motion monitoring functions for 1 axis/spindle.

Safety Integrated plus – SI-Logic

Option; order code S60

Article No.: [6FC5800-0AS60-0YB0](#)

This option enables the F-PLC integrated in the SINUMERIK for failsafe programming.

Safety Integrated plus – SI multi-axis package

Option; order code S61

Article No.: [6FC5800-0AS61-0YB0](#)

Enhanced motion monitoring functions for all axes/spindles activated in the SINUMERIK.

Sag compensation, multi-dimensional

Option; order code M55

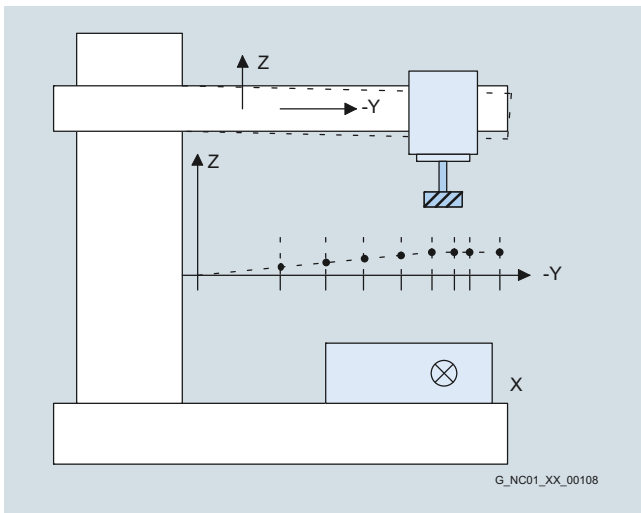
Article No.: 6FC5800-0AM55-0YB0

Multidimensional compensation is also possible for the effects of physical influences and manufacturing tolerances such as sag or leadscrew pitch errors. The compensation tables can be switched over from the PLC. When the reference axis and the compensating axis are identical, leadscrew pitch 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 tolerance band that can be corrected is 10 mm (0.39) for standard versions.



Sag compensation

Restricted functionality of export versions:

The correctable tolerance band is restricted to 1 mm (0.039 in).

Scalable analog setpoint, Run MyCC /SANS

Option; order code N48

Article No.: 6FC5800-0AN48-0YB0

SINUMERIK Integrate Run MyCC /SANS implements isochronous switchover of analysis of an analog setpoint in the position controller via a high-speed digital output of the NCU. This function has been developed for the operation of spindles supplied by Fischer.

Restricted functionality of export versions:

Not possible.

Scratching, determining work offset

A work 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 and entering the desired setpoint position (e.g. 0); the CNC then calculates the work offset.

Screen blanking

When screen blanking is activated, both the screen and backlighting of the operator panel go blank under PLC control or after a programmable period of time has elapsed. This increases the service life of the screens.

Glossary

Functions and terms

SINUMERIK 840D sl

Security

→ Industrial Security

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 (non-modal) for the corner and chamfer contour elements. Feed reduction thus makes it possible to achieve the desired geometrically precise definition of corners and chamfers.

Set actual value

The Set actual value function is provided as an alternative to the Preset function: To use this function, the control must be in the workpiece coordinate system (WCS). With set actual value, the workpiece coordinate 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

Option; order code M05

Article No.: 6FC5800-0AM05-0YB0

Setpoint exchange 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 in the CNC, but are powered by the same motor one after the other.

It is possible to connect up to 4 axes to one motor. The axes, between which the setpoint is switched, can be assigned to different channels or mode groups.

ShopTurn/ShopMill

Option; order code P17

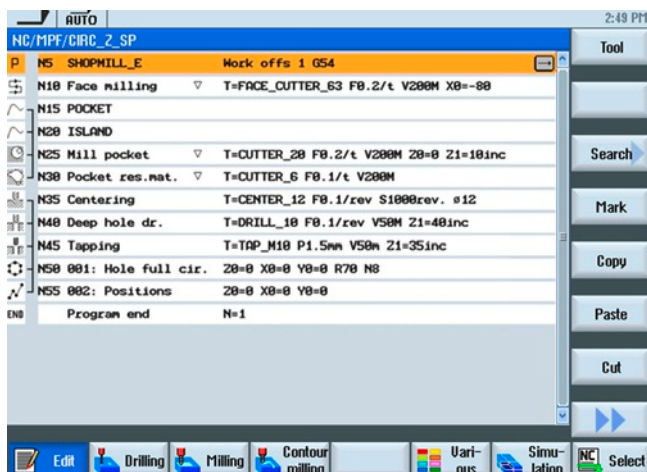
Article No.: 6FC5800-0AP17-0YB0

The ShopTurn/ShopMill option includes the functions:

- Multiple clamping of identical/different workpieces
- Manual machine

Machining step programming

Processes such as drilling, centering, plunging or pocket milling are represented as machining steps in a simple and clear manner. In this way part programs – even for complex machining operations – are very compact and easily read. Associated sequences are automatically interlinked and can be assigned any position patterns. This unique programming convenience allows you to achieve the shortest programming times even for highly demanding machining tasks.



ShopMill: Machining step programming

Simulation axis/spindle

The speed control loop of an axis can be simulated for test purposes. The axis traverses with a following error, similar to a real axis. Setpoint output and actual values are not available. For simulated axes, the program is executed exactly the same as for real axes, i.e. no functional restrictions apply to simulated axes.

Functions:

- POS/SPOS/M3, M4, M5 (from CNC block)
- POSA/SPOSA (from CNC block, modally)
- FC18/POS/SPOS/M3, M4, M5 (PLC axes)
- PLC-VDI interface (M3, M4, M5 directly)
- OSCILL (asynchronous oscillation)
- OSCILL (synchronous oscillation)
- do/POS/SPOS/M3, M4, M5 (synchronized actions)
- Couplings (TRAIL, LEAD, EG, CP, ...)
- Path axes/geometry axes/additional path axes/GEOAX()
- Spindles for thread cutting, tapping and thread cutting with compensating chuck
- Commissioning with SINUMERIK Operate

→ [Axis/spindle](#)

→ [Positioning axis/auxiliary spindle](#)

→ [Virtual axis](#)

→ [PLC axes](#)

Glossary

Functions and terms

SINUMERIK 840D sl

Simulation SINUMERIK Operate

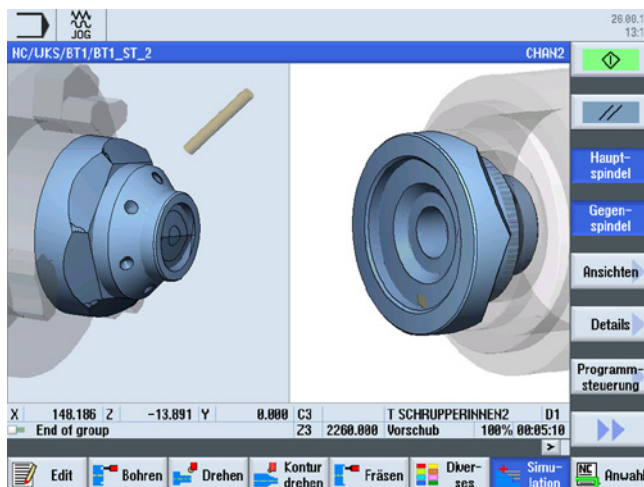
Simulation is supported by an autonomous program interpreter (SNCK) and a separate simulation data environment in SINUMERIK Operate. The SNCK considers the entire syntax of the SINUMERIK CNC family, including the possibility of incorporating special user options on the machine by comparing data with the NCK environment.

The simulation data can be matched statically as required with the NCK environment (initialization data, macros, user data, tool data, machining cycles) or also dynamically when tool data or machining cycles are changed. Machining simulations, with emphasis on the drilling, milling and turning technologies, can be performed in the workpiece coordinate system for certain machine kinematics on the user interface of the CNC.

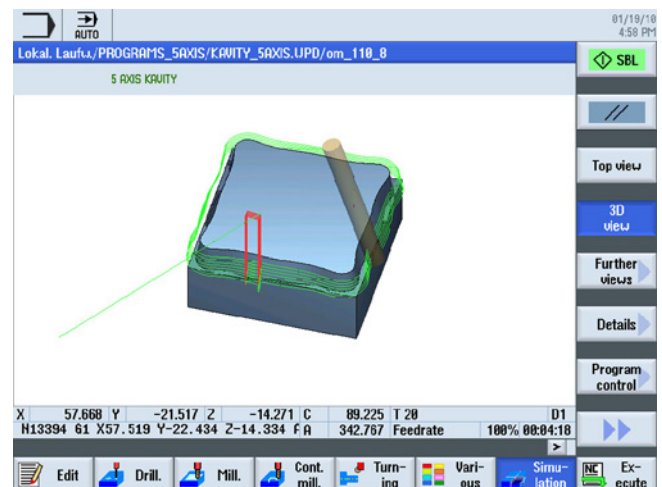
Simulation of the finished part is performed with the real CNC data. The CNC data are automatically matched at each change of the part program.

- The simulation allows a dynamic representation of the machining operation, even when 5-axis transformation TRAORI is active and with swiveled planes.
- Simulation of up to 4 machining channels for turning machines with B axis
- Turning on milling machines
- Simulation of a machining channel for conventional milling machines
- Simulation of up to 12 axes
- Intelligent determination of the block times and the program execution time.
- Very fast graphical representation through continuous refinement of the workpiece
- Optimum resolution for each selected display section
- Any section can be selected
- While one workpiece is being machined, the machining of another workpiece can be simulated in parallel (from NCU 720.x and higher)

The simulation can be extended to 3D representation by means of the 3D simulation 1 (finished part) option. This extension also applies to the simultaneous recording.



Turning/drilling simulation



Milling simulation

Notes:

- Very fast simulation result, but consequently low dynamic resolution
- Fine recording is too slow for complex displays/fine recording fails to run for very complex parts
- Quickview for mold-making part programs
- Simulation of special kinematics not available when compile cycles are used
- Blank CAD data cannot be imported, e.g. for castings
- Couplings cannot be simulated
 - Coupled motion: TRAILON(<following axis>, <leading axis>, <coupling factor>)
 - Axial master-value coupling: LEADON(Faxis, LAxis, curve table n) actual value and setpoint coupling
 - Electronic gear: EGDEF(FA, LA1, coupl.1, LA2, coupl.2, ... LA5, coupl.5), EGON, EGONSYN, EGONSYNE
- Reference point approach G74 is not operative during simulation
- The following is not supported:
 - Axis container
 - Dependencies on part programs and PLC
 - Multiple clamping
 - CNC function synchronized actions for safe programmable logic (SPL) in the CNC
- Different technologies in different channels with different axis coordinate systems are not fully supported.
- Threads with variable pitch are shown with a constant pitch.

→ 3D simulation 1 (finished part)

Simultaneous recording (real-time simulation of the current machining operation)

Option; order code P22

Article No.: [6FC5800-0AP22-0YB0](#)

During machining, the tool paths can be simultaneously recorded on the CNC screen in 3-side view or 3D view. Workpiece graphics and views correspond to the graphic simulation

Note:

Activate simultaneous recording prior to CNC start in order to obtain a complete display.

→ [Simulation SINUMERIK Operate](#)

SINUMERIK extended touch

Option; order code P80

Article No.: [6FC5800-0AP80-0YB0](#)

The SINUMERIK extended touch option expands SINUMERIK Operate to include multi-touch functionality. SINUMERIK Operate then offers state-of-the art gesture-based operation to allow leading-edge user interface designs.

The specific function SINUMERIK extended touch is not intended for general use.

SINUMERIK Grinding Advanced

Option; order code S35

Article No.: [6FC5800-0AS35-0YB0](#)

The SINUMERIK Grinding Advanced option provides an extended scope of grinding functions such as an axis-parallel dressing/pre-profiling cycle, for example.

SINUMERIK MDynamics

Technology packages for 3-axis and 5-axis milling.

The SINUMERIK MDynamics function ensures perfect surfaces through innovative motion control and an optimized CNC data compressor, rapid adaptation to the workpiece, tool and program handling, optimum machining thanks to the flexible programming of programGUIDE and ShopMill and consequently very short programming times.

SINUMERIK Operate /NCU

Option; order code S00

Article No.: [6FC5800-0AS00-0YB0](#)

The option SINUMERIK Operate /NCU permits activation of SINUMERIK Operate in the CNC software for the NCU.

SINUMERIK Operate makes machine tool operation extremely easy. SINUMERIK Operate therefore sets the standard for the efficient operation of machine tools.

→ www.siemens.com/sinumerik-operate

SINUMERIK Operate /PC

Option; order code P87

Article No.: [6FC5800-0AP87-0YB0](#)

The option SINUMERIK Operate /PC permits activation of SINUMERIK Operate in the CNC software for a connected PC.

SINUMERIK Operate makes machine tool operation extremely easy. SINUMERIK Operate therefore sets the standard for the efficient operation of machine tools.

→ www.siemens.com/sinumerik-operate

SINUMERIK Operate /PCU

Option; order code P88

Article No.: [6FC5800-0AP88-0YB0](#)

The option SINUMERIK Operate /PCU permits activation of SINUMERIK Operate in the CNC software for a connected PCU.

SINUMERIK Operate makes machine tool operation extremely easy. SINUMERIK Operate therefore sets the standard for the efficient operation of machine tools.

→ www.siemens.com/sinumerik-operate

Glossary

Functions and terms

SINUMERIK 840D sl

SINUMERIK PCU base software /IPC

Option; order code P86

Article No.: 6FC5800-0AP86-0YB0

The SINUMERIK PCU base software /IPC option provides SINUMERIK PCU functionality on defined SIMATIC IPCs.

TCU support then becomes available for connecting SINUMERIK HT8, SINUMERIK OP and SIMATIC ITC and SINUMERIK Operate /PCU can be used on SIMATIC IPCs.

Skip blocks

CNC blocks that are not to be executed in every program run, e.g. execute a trial program run, can be skipped. Skip blocks are identified by placing a "/" character in front of the block number. The instructions in the skip blocks are not executed and the program resumes with the next block that is not skipped.

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

Spatial compensation for kinematic transformations, Run MyCC /SEC-KT

Option; order code M57

Article No.: 6FC5800-0AM57-0YB0

SINUMERIK Integrate Run MyCC /SEC-KT is a specific function and is not intended for general use.

For Cartesian machines, refer to:

- Spatial compensation VCS-A3
- Spatial compensation VCS-A5
- Spatial compensation VCS A5 plus

Restricted functionality of export versions:
Not possible.

Spatial compensation for rotary axes, Run MyCC /VCS-ROT

Option; order code N31

Article No.: 6FC5800-0AN31-0YB0

This static compensation function enables the user to achieve more accurate tool orientation.

The two swivel axes in the head need to be measured once in the working area. The compensation function then uses the table values to perform a real-time calculation of the offsets for the 3 linear axes referred to the current tool center point.

Restricted functionality of export versions:
Not possible.

Spatial compensation interface, Run MyCCI /VCI

Option; order code N74

Article No.: 6FC5800-0AN74-0YB0

For new applications, use SINUMERIK Integrate Run MyCCI /UCI.

→ Universal spatial compensation interface Run MyCCI /UCI

Restricted functionality of export versions:
Not possible.

Spatial compensation, Run MyCC /VCS-A3

Option; order code N15

Article No.: 6FC5800-0AN15-0YB0

This option implements compensation on Cartesian machines with 3 linear axes in cases where the tool orientation is fixed and the tool lengths are effective in parallel to the coordinates of the linear axes. Gantry axes can be active in parallel to the 3 linear axes.

Restricted functionality of export versions:
Not possible.

Spatial compensation, Run MyCC /VCS-A5

Option; order code N16

Article No. 6FC5800-0AN16-0YB0

This option implements compensation on 5-axis Cartesian machines on which the tool can be oriented relative to the workpiece using 2 rotary axes. Compensation of the measured errors of the linear axes is computed with reference to the tool center point as a function of the current position of the tool.

The machine kinematics can be implemented as pure head kinematics, or as table kinematics with both swivel axes in the table, or as mixed kinematics with one swivel axis in the table – generic transformation types 24, 40 and 56.

Restricted functionality of export versions:

Not possible.

Spatial compensation, Run MyCC /VCS-A5 plus

Option; order code N17

Article No. 6FC5800-0AN17-0YB0

SINUMERIK Integrate Run MyCC /VCS-A5 plus is designed for use on large machines (3-axis and 5-axis gantry milling machines). The requirements for positioning accuracy of these machines over the entire working area (= volumetric accuracy) increases constantly and has become impossible to achieve by mechanical means in many cases.

SINUMERIK Integrate Run MyCC /VCS-A5 plus implements volumetric compensation of all 21 aspects of freedom from errors that the 3 basic axes of a Cartesian machine tool can demonstrate (per axis: linearity, 2 degrees of error, rolling, nodding, yawing as well as deviations from the perpendicular between axes). These geometrical machine errors result in offset of the tool center point and an orientation error in the tool.

On a 5-axis machine, the tool center point and the orientation error of the tool are compensated with Run MyCC /VCS-A5 plus with TRAORI active. In comparison to leadscrew error compensation LEC and circle error compensation, Run MyCC /VCS-A5 plus supports the total compensation of the geometric errors of a Cartesian machine tool.

Run MyCC /VCS-A5 plus can be superimposed on existing LEC and CEC compensations.

The geometric error of a machine tool is measured with external laser-based instruments that determine these errors as effectively as possible. The measured results are supplied to the SINUMERIK CNC as a file in the specified readable VCS format. Measuring machine errors is the responsibility of the OEM or machine operator. Several SINUMERIK Solution Partners offer support and measurement as a service and they can generate the machine-specific VCS files necessary for VCS.

Restricted functionality of export versions:

Not possible.

Glossary

Functions and terms

SINUMERIK 840D sl

Spindle functions

Spindle modes are:

- Open-loop control mode, constant spindle speed S or constant cutting rate G96
- Oscillation mode
- Positioning mode
- Synchronous mode (synchronous spindle)
- Thread cutting/tapping

Functions of the spindle modes:

- Spindle speed with spindle override
- 5 gear stages, specified in the
 - Part program (commands M41 to M45)
 - Automatically via programmed spindle speed M40 or
 - 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
 - Spindle speed, max.
 - Programmable lower (G25) and upper (G26) spindle speed limitation
 - Speed of the gear stage, min./max.
 - Encoder limit frequency, max.
 - End 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 (so that it is easier to engage a new gear stage) and positioning
- Changeover to axis mode:

For machining with a position-controlled spindle (e.g. face machining of turned parts), the main spindle drive can be switched to the axis mode using 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, so there is no longer any need to home the C axis (synchronize C axis on the fly).
- Thread cutting with constant pitch ¹⁾:

The following thread types can be produced with G33: cylindrical, taper and face thread, single-start or multiple-start, 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⁾:

Threads can also be programmed with linearly progressive (G34) or linearly degressive (G35) lead.
- Programmable thread run-in and run-out path:

When thread cutting, DITS/DITE (displacement thread start/end) can be used 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. A prerequisite for rigid tapping (G331/G332) is a position-controlled spindle with position measuring system. The traversing range of the drilling axis is therefore not restricted. By using the method where the spindle, as a rotary axis, and the drilling axis interpolate, threads can be cut to a precise final drilling depth, e.g. for blind hole threads.

¹⁾ Requirement: actual-position sensor (measuring system) with corresponding resolution (mounted directly on the spindle).

Spindle speed limitation

→ Spindle functions

Spline interpolation (A, B and C splines)

Option; order code S16

Article No.: 6FC5800-0AS16-0YB0

Using spline interpolation it is possible to obtain a very smooth curve from just a few defined interpolation points along a set contour. The interpolation points are connected by polynomials. The compressor converts linear movements, 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 interpolation points are placed close together, quite sharp edges can also be programmed. Spline interpolation also considerably reduces the number of program blocks required.

Extremely smooth workpiece surfaces are often very important in mold and tool making, both optically and technologically, e.g. for rubber gaskets. Tool radius compensation is also possible in spline interpolation, just the same as for 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 (interpolation points).
- C spline is true to the tangents and the curvature and runs through the nodes.

With the COMPCAD compressor, smooth curves of this kind can be approximated within the boundaries of compressor tolerance (parallel tool paths) so that surfaces of a high optical quality can also be obtained even in the case of increased tolerances.

Spline interpolation for 3-axis machining is suitable for simple applications and for the JobShop area.

Standstill monitoring

Standstill monitoring represents one of the most comprehensive mechanisms for monitoring axes. This monitoring function checks to see whether the following error has reached the standstill tolerance limit after a parameterizable time has elapsed. After positioning has been completed, standstill monitoring takes over from position monitoring, and checks to see whether the axis moves further from its position than specified in the machine data's standstill tolerance field.

The standstill monitoring function is always active following expiration of the zero speed delay time or when reaching the fine exact stop limit as long as no new traversing command is pending. When the monitoring function responds, an alarm is generated and the relevant axis/spindle brought to standstill with rapid stop along a speed setpoint ramp. Standstill monitoring is effective for linear and rotary axes as well as for position-controlled spindles.

Standstill monitoring is inactive in the follow-up mode.

→ [Position monitoring](#)

Subprogram levels and interrupt routines

Subprograms can be called in the main program as well as in other subprograms. Subprograms can be nested to a maximum depth of 12 levels, including the main program level. That means that a main program may contain as many as 11 nested subprogram calls.

When working with Siemens cycles, 3 levels are required. If such a cycle is to be called from a subprogram, the call can be nested at a depth of no more than 9.

Programs can also be called on the basis of events following reset, part program start or end, or after the CNC has booted. Users can then make the basic function settings or can initialize functions using a part program command. A system variable can be used to scan the event, which activated the associated program.

Glossary

Functions and terms

SINUMERIK 840D sl

Synchronized actions

Even in their basic configuration, SINUMERIK CNCs allow up to 24 actions to be initiated in synchronism with axis and spindle movements. These actions run in parallel with workpiece machining; specific conditions are used to define when they become active. The starting of such motion-synchronized actions – or synchronized actions for short is, therefore, not restricted to CNC block boundaries. Synchronized actions are an excellent programming tool for supporting very quick responses to events in the interpolation cycle.

Synchronized actions are executed in the interpolation cycle. Multiple actions can be processed in the same IPO cycle. Synchronized actions without validity identifier are active non-modally only in the automatic mode. In the automatic mode, synchronized actions with validity identifier ID are modal in the subsequently programmed blocks. Statically effective synchronized actions with the identifier IDS remain active in all CNC operating modes: Cross-mode actions.

Typical applications include:

- Comparison operation-dependent or external signal-dependent transfer of auxiliary functions M and H to the PLC user software and derived machine responses
- Fast, axis-specific deletion of the distance-to-go in response to input signals
- Manipulation of the read-in disable for the CNC block using external signals
- Monitoring of system variables such as velocity, power and torque
- Control of process variables such as velocity, speed and distance

Restricted functionality of export versions:

Only 1 active synchronous function SYNFACT is possible at a time.

The number of axes that can be simultaneously traversed is restricted to 4 (path and positioning axes).

→ Cross-mode actions ASUB and synchronized actions

Synchronized actions stage 2

Option; order code M36

Article No.: 6FC5800-0AM36-0YB0

More than 24 synchronized 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 synchronized actions, stage 2, making it possible, for example, to start axis programs in the same IPO cycle by scanning digital inputs.

Restricted functionality of export control versions:

The number of axes that can be simultaneously traversed is restricted to 4 (path and positioning axes).

Synchronous motion, RUN MyCC /SYMO

Option; order code N66

Article No.: 6FC5800-0AN66-0YB0

SINUMERIK Integrate Run MyCC /SYMO synchronous motion can be used to synchronize the traversing motion of 2 separate machines (e.g. 2 robots for testing components based on ultrasound transmission). In this case, each of the machines is configured in a separate channel. The part program is executed in the master channel. The TCP and orientation of the second machine in the slave channel are coupled in real time to the motion of the machine in the master channel.

Restricted functionality of export versions:

Not possible.

Synchronous spindles, multi-edge turning

Precise angular synchronization between 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 synchronism, the relative angular position of the spindles to one another, e.g. on-the-fly, position-oriented transfer of skewed workpieces, is also specifiable.

On-the-fly transfer:

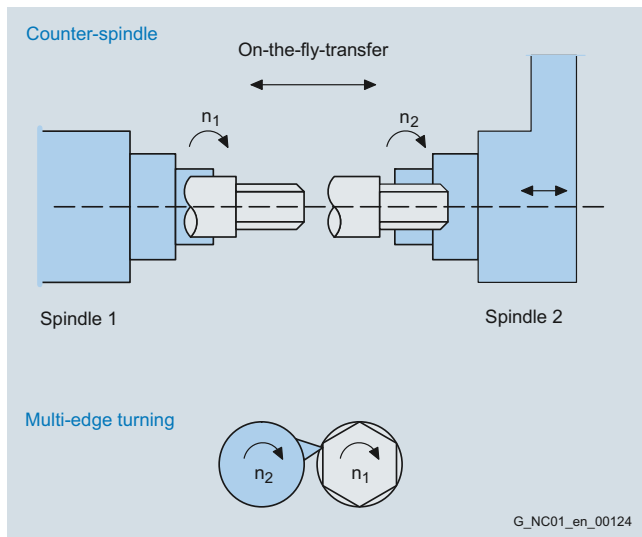
- $n_1 = n_2$
- Angle 1 = angle 2 or
- Angle 2 = angle 1 + angle Δ

Finally, specification of an integer speed ratio between the main spindle and a tool spindle provides the prerequisites for multi-edge machining (polygon turning).

Multi-edge turning:

$$n_2 = T \times n_1$$

Configuring and selection take place either via the CNC program or operator panel. Several pairs of synchronous spindles can be implemented.



Examples for synchronous spindles/multi-edge turning

→ [Generic coupling](#)

Glossary

Functions and terms

SINUMERIK 840D sl

Tangential control

Option; order code M06

Article No.: 6FC5800-0AM06-0YB0

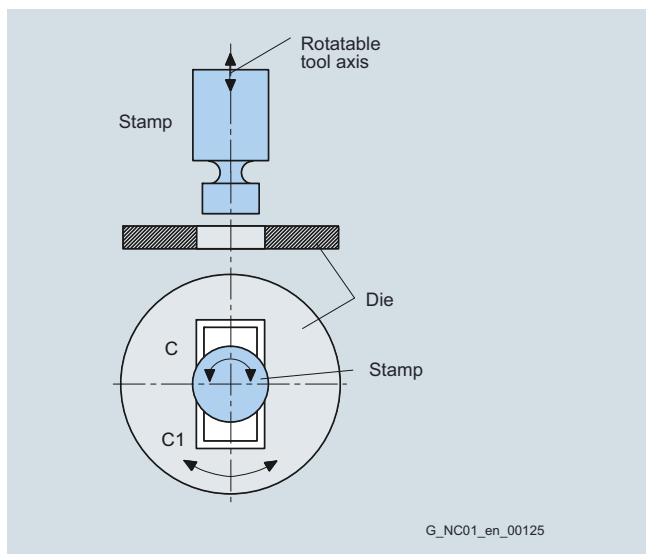
Tangential control makes it possible to correct a rotary axis in the direction of the tangents of two path axes. The two guide axes and the corrected axis lie in the same channel. Tangential control is effective in all interpolation modes.

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 a cutting wheel for cutting glass or paper

On punching and nibbling machines with a rotatable punching tool and associated lower tool, the following functions may be used to ensure that the tool can be no universally used:

- Tangential control
TANGON/TANGOF for vertical rotary axis alignment of the punching tools to the direction vector of the programmed path
- Coupled motion
TRAILON/TRAILOF for synchronous rotation of upper and lower tools (stamp and die)



Representation of a rotatable tool axis and die during punching/nibbling

Tapping with/without compensating chuck

→ Spindle functions

Technological functions with compressor, Run MyCC /COTE

Option; order code N50

Article No.: 6FC5800-0AN50-0YB0

SINUMERIK Integrate Run MyCC /COTE is a special function and has not been approved for general use.

Restricted functionality of export versions:

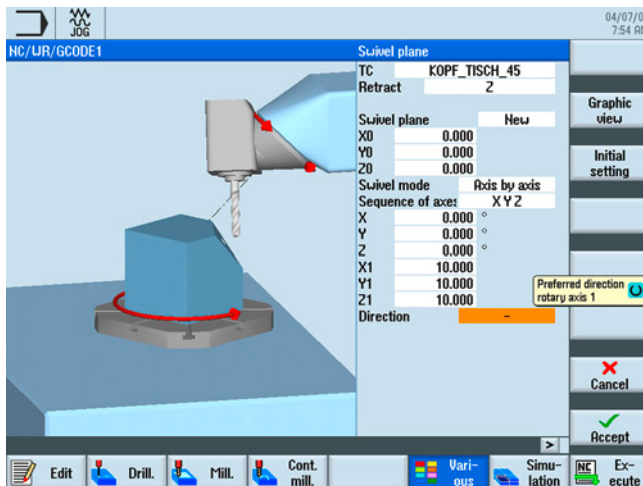
Not possible.

Technology cycles

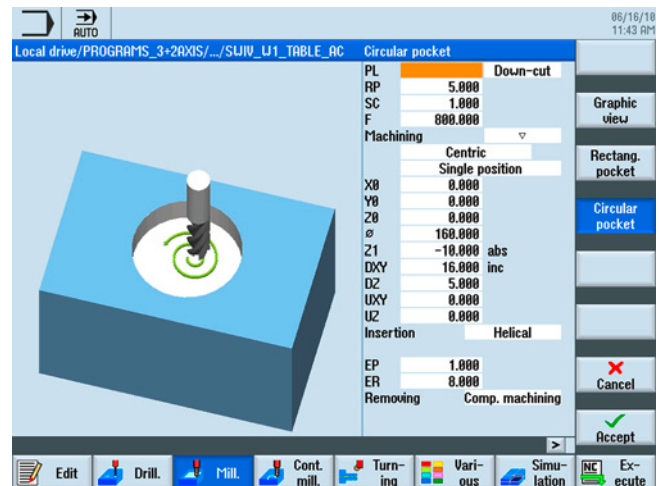
For frequently repeated machining tasks, technology cycles are available for the drilling/milling and turning technologies. Technology cycles are generally applicable technology subprograms, with which specific machining processes can be implemented, such as tapping a thread or milling a pocket.

The cycles are adapted to a concrete machining task using parameters. The parameterization can also be implemented using graphically supported input screens.

- Drilling technology:
Drilling/centering, drilling/counterboring, deep-hole drilling, tapping with and without compensating chuck, boring 1 ... 5, row of holes, circle of holes, grid of holes, machining on inclined surfaces
- Milling technology:
Thread milling, elongated holes in a circle, grooves in a circle, circumferential groove, rectangular/circular pocket, face milling, path milling, rectangular/circular spigot, machining on inclined surfaces, high-speed settings for optimized HSC machining, engraving cycle
- Turning technology:
Groove, undercut, cutting with relief cut, thread undercut, thread cutting, chaining of threads, thread recutting



Example: Swiveling



Example: Milling the circular pocket

Temperature compensation

Heat causes machine parts to expand. This 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 changes can be corrected using temperature compensation.

At a specific temperature, measure the actual-value offset over the positioning range of the axis to obtain the error curve for this temperature value. Error curves for different temperatures can be defined for each axis. In order to ensure proper compensation of thermal expansion at all times, the temperature compensation value, reference position and gradient (angle) parameters must be transferred 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 unnecessarily triggering monitoring functions.

Thread cutting

→ Spindle functions

Glossary

Functions and terms

SINUMERIK 840D sl

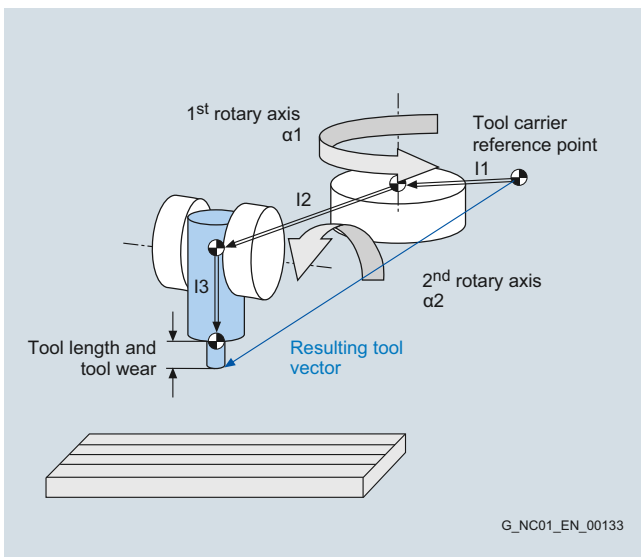
Tool carrier with orientation capability

For machine tools which have tool carriers with settable tool orientation, these kinematics can be freely configured without a 5-axis transformation when the SINUMERIK CNC is used.

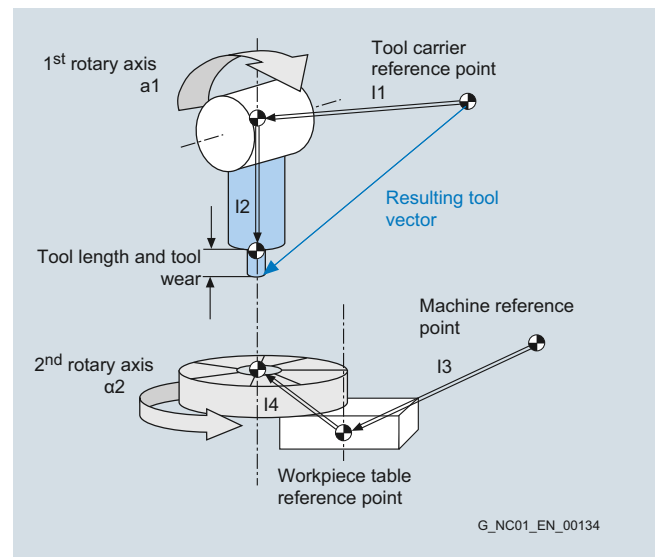
The tool carrier with orientation capability function enables 2½D/3D machining with fixed spatial orientation of the tool/workpiece table. Vectors I1 to I4 represent the geometrical dimensions of the machine. The rotary axes need not move in parallel to the Cartesian axes, but instead can be inclined 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 table.

The following kinematics can be configured flexibly:

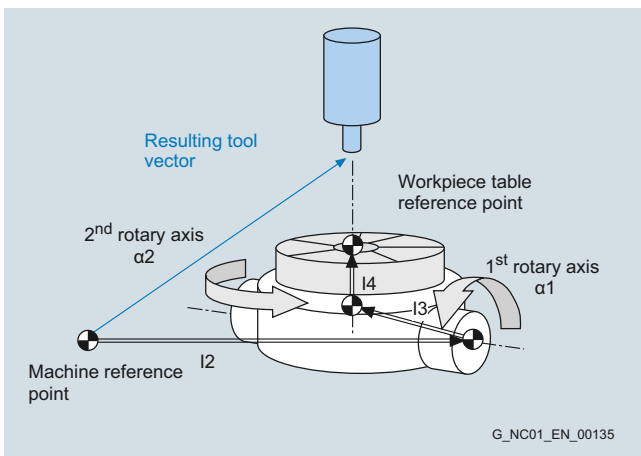
- Rotatable tool: type T (tool)
- Rotatable tool/rotatable workpiece table: type M (mixed)
- Rotatable workpiece table: type P (part)



Kinematics type T



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 2 stages: A T command locates the tool in the magazine, and an M command inserts it in the spindle.

In circular 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 Ident Connection

Option; order code P52

Article No.: 6FC5800-0AP52-0YB0

The Tool Ident Connection function allows tool identification systems such as SIMATIC Ident (see Catalog ID 10) to be linked to SINUMERIK CNCs for operation in conjunction with the standard tool management system, and also supports the transfer of tool data. It therefore eliminates the need for manual entries that are susceptible to errors.

→ [Tool identification systems](#)

Tool identification systems

The tool load and unload dialog of the Siemens tool management system provides a link to an automatic tool identification system, e.g. SIMATIC Ident (see Catalog ID10). This means that you can automatically read and write to the tool code carrier, instead of having to manually enter tool data. When unloading, the data block for the tool is saved; when loading, it is read via the code carrier and entered in the tool management.

In the interim, the tool data can be subsequently edited the same as when the tool is selected from the tool catalog, e.g. offset data. Using an editable description file containing precisely defined tool and cutting data, when loading, the code carrier data is converted into dialog data that can be read by the tool management. When unloading, the dialog data is converted back into code carrier data using the description file.

→ [Tool Ident Connection](#)

Tool management

Tool management ensures that the correct tool is in the correct location on the machine at any given time 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 both scrap by monitoring the tool service life and machine downtimes by using spare tools.

The most important functions of tool management are:

- Tool selection across all magazines and turrets for active tools and spare tools
- Determining a suitable empty location depending 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 at the next D number
- Quantity, tool life and wear monitoring with prewarning limit monitoring function
- Support when using multi-tools

Missing tools can be loaded based on a decision made by the operator. Tools with similar wear characteristics can be combined into wear groups.

Tool management also takes into account tool length compensations for adapters that are permanently mounted at certain magazine locations and fitted with different tools.

Tool management with more than 4 magazines

Option; order code M88

Article No.: 6FC5800-0AM88-0YB0

4 magazines are included in the basic version of SINUMERIK 840D sl:

- Magazine
- Spindle
- Tool buffer
- With the option with order code M88, the number can be increased to a maximum of 64 magazines.

Glossary

Functions and terms

SINUMERIK 840D sl

Tool offsets

You can select the tool by programming a T function (5-digit integer number or identifier) in the block. Every tool can be assigned up to 12 cutting edges (D addresses). The number of tools to be managed in the CNC is set when configuring.

A tool offset block comprises 25 parameters, e.g.:

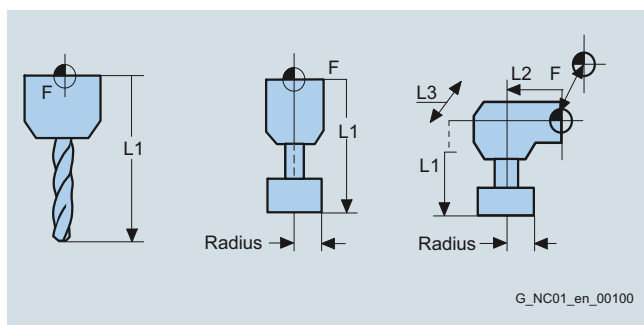
- Tool type
- Up to 3 tool length offset values
- Radius compensation
- Wear dimension for length and radius
- Tool base dimension

The wear and the tool base dimension are added to the corresponding offset. When writing the program, you do not have to take tool dimensions such as cutter diameter, cutter position or tool length into account. You program the workpiece dimensions directly, following the production drawing, for example.

When a workpiece is machined, the tool paths, depending on the relevant tool geometry, are controlled so that the programmed contour can be produced with every tool used. The tool data are entered separately in the tool table of the CNC, and only the required tool with its offset data is called in the program. During program execution, the CNC retrieves the required offset data from the tool files and automatically corrects the tool path for various tools.

Tool offset D always has a reference to tool number T when Siemens tool management is active, e.g. with monitoring functions and management of sister tools.

Up to 32000 D values can be defined per CNC. When programming, D numbers can be freely assigned, checked, renamed, determined with the associated T number, invalidated and activated on a site-dependent basis.



Tool offsets

Tool offsets, grinding-specific

Grinding-specific tool offsets are available for grinding technology:

- Minimum wheel radius
- Maximum speed
- Maximum surface speed

When cutting edges are created for grinding tools (tool type 400 to 499), these are stored automatically for the tool in question.

Tool type	
400	Surface grinding wheel
401	Surface grinding wheel with monitoring
403	Surface grinding wheel with monitoring and without tool base dimensions for grinding wheel surface speed
410	Facing wheel
411	Facing wheel with monitoring
413	Facing wheel with monitoring and without tool base dimensions for grinding wheel surface speed
490 ... 499	Dresser

The TMON command activates geometry and speed monitoring for grinding tools, types 400 to 499 in the CNC part program. Monitoring remains active until deactivated in the part program with TMOF. The actual wheel radius and the actual wheel width are monitored.

The speed setpoint is monitored cyclically in relation to the speed limit value, taking into consideration the spindle override. The speed limit value is the smaller of the values resulting from comparison of the maximum speed with the speed computed from the maximum grinding wheel surface speed and the current wheel radius.

→ [Grinding wheel surface speed](#)

Tool orientation interpolation

Interpolations of tool orientations supplement generic transformation. The tool orientation can be programmed in a plane as large circle interpolation ORIVECT, on the outside surface of a taper in the clockwise or counterclockwise direction ORICONCW/ORICONCCW, or even with free definition of the tool curve orientation ORICURVE.

→ Transformation, generic

Tool radius compensation

When tool radius compensation is active, the CNC automatically calculates the equidistant tool paths for different tools. To do so, it requires the tool number T, the tool offset number D (with cutting edge number), the machining direction G41/G42, and the relevant working plane G17 to G19. The path is corrected in the programmed level depending on the selected tool radius.

The approach and retract paths can be adapted to the required contour profile or unmachined part (blank) forms, for example:

- **NORM**
The tool travels in a straight line directly to the contour and is positioned perpendicular to the path tangent at the starting point.
- **KONT**
If the starting point is behind the contour, the corner point P1 of the contour is bypassed. If the starting point is in front of the contour, the normal position at the starting point P1 is approached in the same way as with NORM.

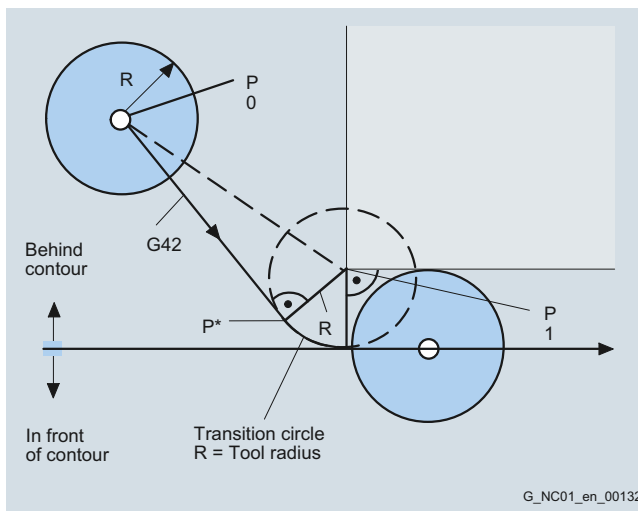
In the part program it is also possible to select the strategy with which the outside corners of the contour are to be bypassed:

- Transition radii: Circle or ellipse
- Intersection of equidistant paths

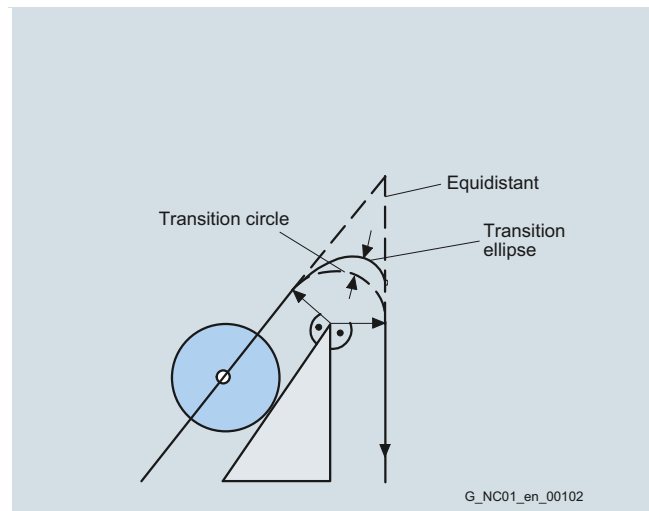
You can implement various strategies spatially or in the plane to achieve soft approach to/retraction from the contour, i.e. tangential approach and retraction irrespective of the position of the starting point:

- Approach and retraction from left or right
- Approach and retraction along a straight line
- Approach and retraction along a quadrant or semicircle

The CNC can also automatically insert a circle or a straight line in the block with the tool radius compensation when no intersection with the previous block is possible. The offset process of tool radius compensation may only be interrupted by a certain number of successive blocks or M commands containing no motion commands or positional data in the compensating plane. The number of successive blocks or M commands can be set using machine data (standard 3, max. 5).



KONT for selection behind the contour



Bypassing the outside corners with transition circle/transition ellipse

3D tool radius compensation

Option; order code M48

Article No.: 6FC5800-0AM48-0YB0

Inclined surfaces can be machined with tool radius compensation in 3D representation or tool offset in space. This function supports circumferential milling and face milling with a defined path. The inclined tool clamping position on the machine can be entered and compensated.

The CNC automatically computes the resulting positions and movements. The radius of a cylindrical milling cutter at the tool insertion point is included in the calculation. The insertion depth of a cylindrical milling cutter can be programmed. The milling cutter can be rotated not only in the X, Y and Z planes, but also around the lead or camber angle and the side angle.

Glossary

Functions and terms

SINUMERIK 840D sl

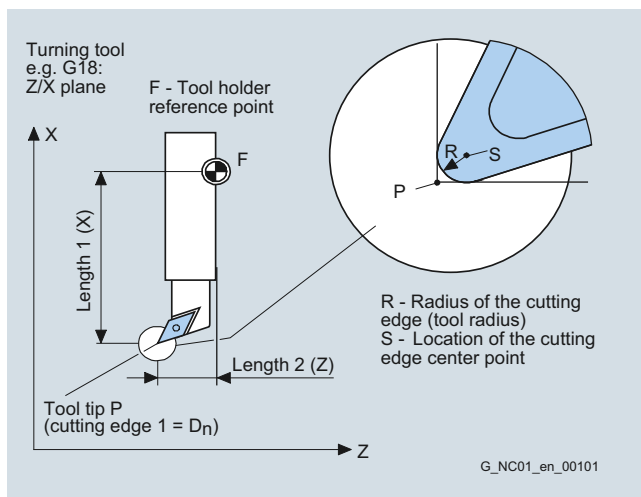
Tool types

The tool type determines the geometry specifications 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 CNC combines these individual components to produce a result variable, e.g. total length, total radius.

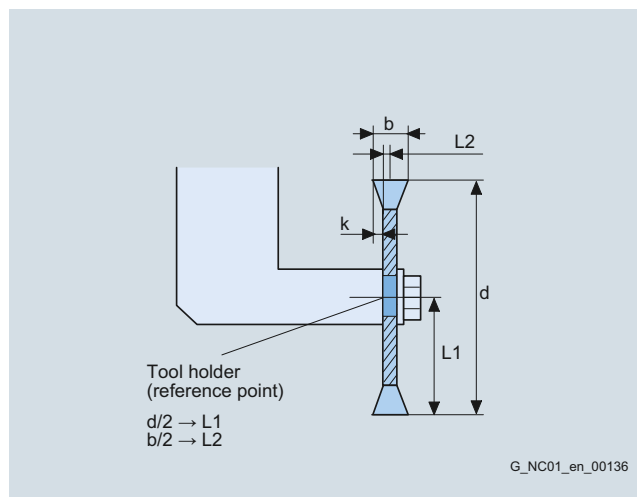
The relevant overall dimension goes into effect when the offset memory is activated. The use of these values in the axes is determined by the tool type and current machining plane G17, G18 or G19.

Tool types, parameterizable		
Group 1xy	Milling tools	From spherical head cutters to bevel cutters
Group 2xy	Drills	From twist drills to reamers
Group 4xy	Grinding tools	From surface grinding wheels to dressers
Group 5xy	Turning tools	From roughing tools to threading tools
Group 700	Slotting saw	Wood technology
Group 900	Special tools	

Input screen forms can be used to save all of the tool offsets.



Geometry of turning tool



Geometry of slotting saw

Tooltips

Tooltips are provided as a simple user help in all screens – also displayed in red when input errors are made.

Top Surface

Option; order code S17

Article No.: 6FC5800-0AS17-0YB0

The function Top Surface with its innovated Compressor COMPSURF assures improved workpiece surfaces. Top Surface copes significantly better with demanding CAD/CAM programs. The workpiece surfaces are perfect, especially for part programs with inhomogeneous distribution of points, with oblique milling finishing programs and bidirectional milling.

Top Surface provides further benefits both for the programmer and the machine operator: Preset standard values for contour and orientation tolerance, which are valid for nearly all NC programs, ensure independence from the tolerance settings used in CAD/CAM, which is often not known in production.

Transformation, generic

Generic transformation is used to define any tool orientation in space with the initial state of the axes, and not just according to the Z direction. The transformation can then be used much more flexibly and universally. It is then also possible to control machine kinematics by the CNC, where the orientation of the rotary axes is not precisely in parallel to the linear axes.

Generic 5-axis transformation is an extension of the 3-axis and/or 4-axis transformation, i.e. it can also be used for machines with only one rotary axis, e.g. rotatable tool or workpiece.

Restricted functionality of export versions:
Not possible.

Transformation: Additional function 6th axis, Run MyCC /TR-6A

Option; order code M66

Article No.: 6FC5800-0AM66-0YB0

SINUMERIK Integrate Run MyCC /TR-6A is a specific function and is not intended for general use.

Restricted functionality of export versions:

Not possible.

Transformation: Double Generic, Run MyCC /DGEN

Option; order code N34

Article No.: 6FC5800-0AN34-0YB0

With the SINUMERIK Integrate Run MyCC /DGEN double generic transformation it is possible to couple two 5-axis kinematics for a machining task.

The axes of both 5-axis machines must be programmed in one channel. Programming coupled 5-axis kinematics is subject to certain restrictions that do not apply to normal 5-axis programming. The application is used in 5-axis milling operations on thin structures (e.g. outer shells of aircraft) that require simultaneous movement of 5-axis kinematics as a counter-support immediately beneath the tool center point.

Restricted functionality of export versions:

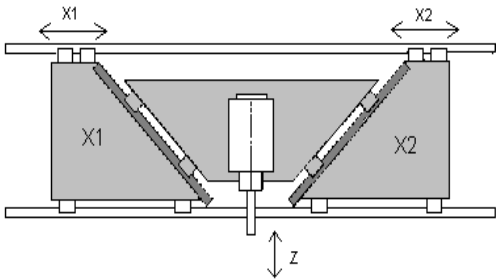
Not possible.

Transformation: Double slide, Run MyCC /SKID

Option; order code M80

Article No.: 6FC5800-0AM80-0YB0

The purpose of the SINUMERIK Integrate Run MyCC /SKID transformation is to traverse a platform that can move in the Z and X directions using two slides that can move only in the X direction. The platform that supports the spindle is mechanically coupled to the two X slides using slideways arranged at an oblique angle.



The specific function SINUMERIK Integrate Run MyCC /SKID is not intended for general use.

Restricted functionality of export versions:

Not possible.

Glossary

Functions and terms

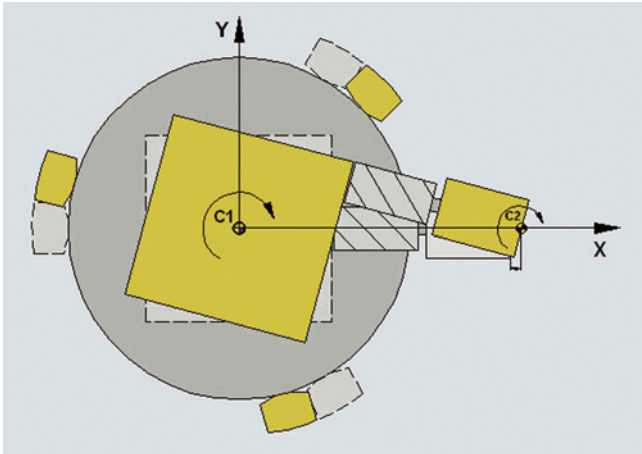
SINUMERIK 840D sl

Transformation: DOUBLETRANSMIT, Run MyCC /2TRA

Option; order code M25

Article No.: 6FC5800-0AM25-0YB0

The SINUMERIK Integrate Run MyCC /2TRA DOUBLETRANSMIT transformation supports machining on the end face or cylinder surface of a workpiece in a lathe with a driven milling tool that is always held vertically with respect to the contour by means of an appropriate swivel movement and compensation movement in X.



Transformation: DOUBLETRANSMIT Run MyCC /2TRA

Restricted functionality of export versions:
Not possible.

Transformation: Dynamic swivel TRIPOD, Run MyCC /DSTT

Option; order code M84

Article No.: 6FC5800-0AM84-0YB0

The kinematic transformation Dynamic Swivel TRIPOD SINUMERIK Integrate Run MyCC /DSTT is a 5- or 6-axis transformation with serial-parallel kinematics. It therefore allows an axially symmetrical tool (milling cutter, laser beam) to be oriented to the workpiece in the machining space. The restriction to axially symmetrical tools no longer applies with 6 axes. The transformation is programmed in the Cartesian coordinate system including orientation via TRAORI. The machining programs are therefore independent of the special kinematics.

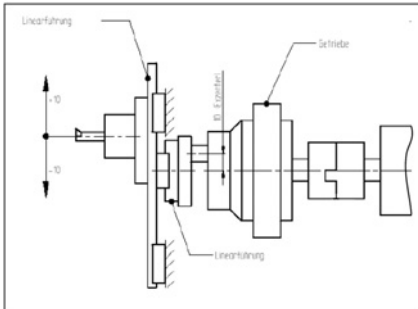
Restricted functionality of export versions:
Not possible.

Transformation: Eccentric, Run MyCC /ECCE

Option; order code N41

Article No.: 6FC5800-0AN41-0YB0

SINUMERIK Integrate Run MyCC /ECCE transformation eccentric uses an eccentric gear to convert the rotation of an axis into a one-dimensional linear slide motion. The motion component perpendicular to the direction of the moving slide is absorbed by a journal bearing.



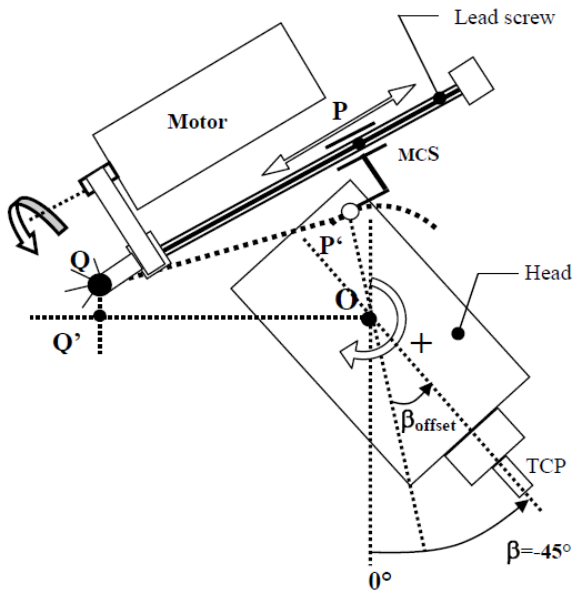
Restricted functionality of export versions:
Not possible.

Transformation: Eccentric axis, Run MyCC /ECCA

Option; order code N44

Article No.: 6FC5800-0AN44-0YB0

SINUMERIK Integrate Run MyCC /ECCA supplies adjustments to the position controller that are needed for the high-precision dynamic traversal of a rotary axis which is driven eccentrically by linear motion (threaded rod) in the range < 180 degrees.



Eccentric axis kinematics ECCA

Restricted functionality of export versions:
Not possible.

Transformation: GRINDING, Run MyCC /GRIN

Option; order code N40

Article No.: 6FC5800-0AN40-0YB0

SINUMERIK Integrate Run MyCC /GRIN is a specific function and is not intended for general use.

Restricted functionality of export versions:
Not possible.

Glossary

Functions and terms

SINUMERIK 840D sl

Transformation: Handling, Run MyCC /RCTRA

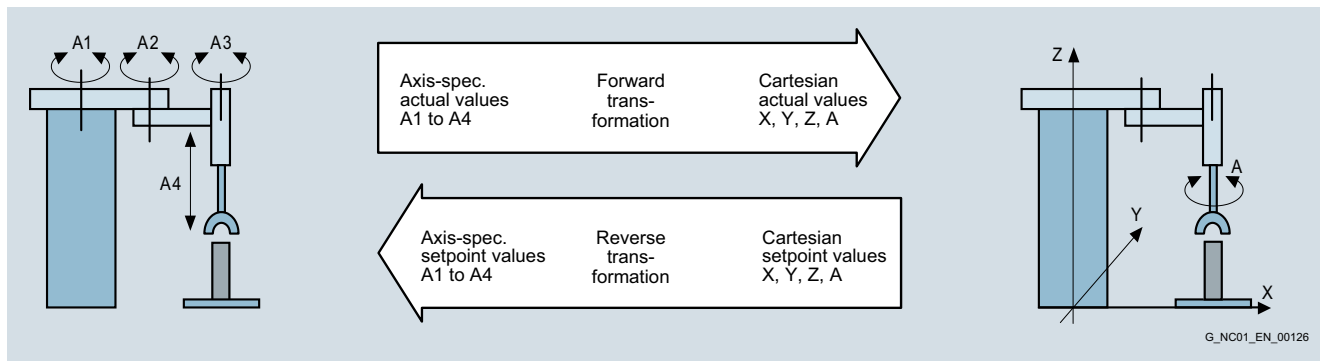
Option; order code M31

Article No.: 6FC5800-0AM31-0YB0

The handling transformation SINUMERIK Integrate Run MyCC /RCTRA contains the so-called standard transformation block, with whose help typical 2-axis to 4-axis handling devices such as gantries or Scara kinematics can be implemented. This coordinate transformation converts programmed Cartesian setpoints (e.g. X, Y, Z, A) into axis positions (e.g. A1 to A4) for the handling device.

Thanks to this coordinate transformation, it becomes significantly easier and more convenient to move the handling device. The handling device can be set up, that is, manually traversed not only in the axis-specific coordinate system, but also in the Cartesian coordinate system, using, for example, the jog keys on the handheld programming unit. Adaptation of the respective kinematics is carried out via machine data.

The function Run MyCC /ROBX is available for applications with more than 4 axes.



Transformation: Handling

Restricted functionality of export versions:
Not possible.

Transformation: HEXAPOD, Run MyCC /HEX

Option; order code M71

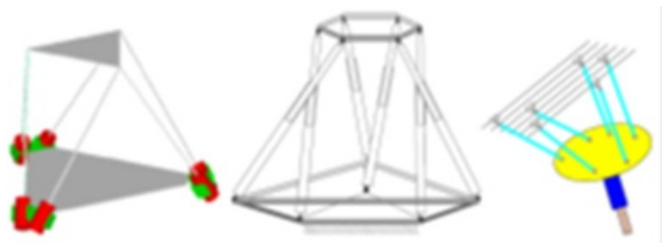
Article No.: 6FC5800-0AM71-0YB0

SINUMERIK Integrate Run MyCC /HEX is a 6-axis transformation with parallel kinematics. The tool platform is moved in all 6 spatial degrees of freedom.

Two versions of the transformation are available:

- 6 variable-length, parallel struts, or
- 6 linear guides, which move the base points of 6 articulated rods.

It therefore allows a tool (e.g. milling cutter, laser beam) to be oriented to the workpiece in the machining space. The path and path velocity are programmed in the same way as for 3-axis systems. The tool orientation is programmed additionally in the traversing blocks. The transformation SINUMERIK Integrate Run MyCC /HEX calculates the resulting motion of all 6 axes. The generated machining programs are therefore not machine-specific. CNC functions such as tool radius and tool length compensation, and the selection of coordinate systems, are available without any restriction.



Restricted functionality of export versions:
Not possible.

Transformation: Machining with virtual Y axis, Run MyCC /MAVY

Option; order code N56

Article No.: 6FC5800-0AN56-0YB0

SINUMERIK Integrate Run MyCC /MAVY transformation machining with virtual Y axis allows users to perform milling and drilling operations on a lathe. This transformation replaces the missing Y axis of a lathe so that milling and drilling operations can be performed on the peripheral surface of turned parts. The user can use a Cartesian coordinate system, machining operations are performed in the X-Y plane (G17).

Restricted functionality of export versions:
Not possible.

Transformation: Pantograph kinematics, Run MyCC /SCIS

Option; order code M51

Article No.: 6FC5800-0AM51-0YB0

The SINUMERIK Integrate Run MyCC /SCIS transformation Pantograph kinematics is a type of 2/4-axis transformation with parallel kinematics. The transformation can work with variable-length rods, or with movable base points of fixed-length rods.

The SINUMERIK CNC calculates the required motion of the machine axes online. Therefore, the programmer can create part programs in the same way as on a conventional machine, and does not have to take the special kinematics of the machine into account. Workpieces can be programmed in the usual way in Cartesian coordinates.

Restricted functionality of export versions:
Not possible.

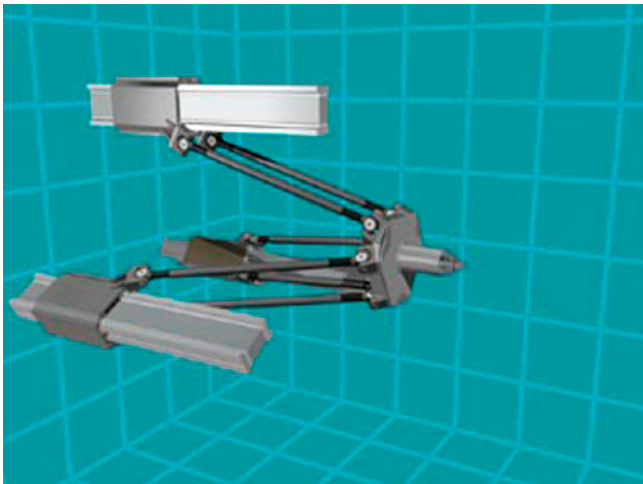
Transformation: PARACOP 3 axes, Run MyCC /PACO

Option; order code M44

Article No.: 6FC5800-0AM44-0YB0

PARACOP kinematic transformations and pantograph kinematics are used on parallel-kinematics machines PKM. Parallel kinematics means that the drive forces engage on the spindle head (Stewart platform) simultaneously (in parallel). PARACOP machines are tripod systems on which the Stewart platform is moved by 3 actuators. Design measures are used to ensure that the Stewart platform cannot move in an undefined manner on these tripod types.

On PARACOP machines, 2 parallel rods run on a slide for each actuator. These machines are suitable for 3-axis machining. When using kinematic transformations, workpieces can be programmed in Cartesian coordinates as usual. The SINUMERIK CNC calculates the required motion of the machine axes online. Therefore, the programmer can create part programs in the same way as on a conventional machine, and does not have to take the special kinematics of the machine into account.



PARACOP

Restricted functionality of export versions:
Not possible.

→ Transformation: Pantograph kinematics, Run MyCC /SCIS

Glossary

Functions and terms

SINUMERIK 840D sl

Transformation: Redundant axes at workpiece, Run MyCC /RDCC

Option; order code N26

Article No.: 6FC5800-0AN26-0YB0

The SINUMERIK Integrate Run MyCC /RDCC kinematic transformation Redundant axes at workpiece is a multi-axis transformation. This function is used particularly for the purpose of assembling large aircraft components which need to be positioned and orientated in a working area. The component is aligned by programming 3 Cartesian coordinates and 3 orientation angles. The function can move up to 31 axes in such a way that the contact points on the component, to each of which up to 3 axes are assigned, do not change their reciprocal positions relative to the moving part.

In combination with the additional function Run MyCC /AMOV, axes of another NCU coupled by means of NCU-Link can also be included in the interpolation with the result that the motion of more than 31 axes in total can be synchronized.

Restricted functionality of export versions:
Not possible.

→ Variable-based axis movement, Run MyCC /AMOV

Transformation: Robotic extended, Run MyCC /ROBX

Option; order code N54

Article No.: 6FC5800-0AN54-0YB0

The SINUMERIK Integrate Run MyCC /ROBX transformation supports robot kinematics with between 4 and 6 axes.

A 6-axis robot can orientate the tool in any desired relationship to the workpiece in every point of the machining space. In order to improve accessibility to machining points on large workpieces, up to 3 additional linear axes, which move the robot or the workpiece, can be included in the transformation. The workpiece can be turned in space by an additional 3 rotary axes that are also included in the transformation. Tool motion and orientation are still programmed with reference to the workpiece.

Restricted functionality of export versions:
Not possible.

Transformation: Rotating eccentric, Run MyCC /ROTE

Option; order code N37

Article No.: 6FC5800-0AN37-0YB0

SINUMERIK Integrate Run MyCC /ROTE transformation rotating eccentric is a 4-axis transformation. The path and path velocity are programmed in the same way as for 3-axis systems. The SINUMERIK Integrate Run MyCC /ROTE transformation calculates the resultant motion of the 4 machine axes. The generated machining programs are therefore not machine-specific. The Y coordinate is converted to a corresponding position of axis 4. Compensating motion of axes 1 and 2 are then calculated as a function of axis 3.

Restricted functionality of export versions:
Not possible.

Transformation: Rotating workpiece and tool, Run MyCC /2RPT

Option; order code N43

Article No.: 6FC5800-0AN43-0YB0

The SINUMERIK Integrate Run MyCC /2RPT transformation rotating workpiece and tool is a special 3-axis transformation. The path and path velocity are programmed in the same way as for 3-axis systems. The SINUMERIK Integrate Run MyCC /2RPT transformation calculates the resultant motion of the 3 machine axes. The generated machining programs are therefore not machine-specific.

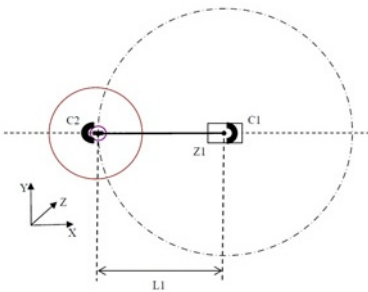
With these kinematics, the tool is swiveled by a rotary axis (C1) that rotates precisely about the Z direction of the basic coordinate system. The workpiece is turned by the second rotary axis (C2) which travels in parallel to the first.

At the zero position of the first rotary axis, the toolholder is positioned exactly above the center of rotation of the second rotary axis. The tool unit also has a serially arranged Z axis (Z1).

The basic coordinate system is at the center of rotation of the second rotary axis.

Length L1 describes the distance between the two rotary axes.

The X coordinate of the basic coordinate system points towards the first rotary axis.



Restricted functionality of export versions:
Not possible.

Transformation: SCARA, 2/3 axes, Run MyCC /SCRA

Option; order code M68

Article No.: 6FC5800-0AM68-0YB0

SINUMERIK Integrate Run MyCC /SCRA is a customized transformation. SINUMERIK Integrate Run MyCC /RCTRA represents a general solution.

→ Transformation: Handling, Run MyCC /RCTRA

Restricted functionality of export versions:
Not possible.

Transformation: Swivel axis, Run MyCC /PIVA

Option; order code N52

Article No.: 6FC5800-0AN52-0YB0

The SINUMERIK Integrate Run MyCC /PIVA transformation swivel axis allows the user to program in a Cartesian coordinate system with X, Y and Z axes, while at the machine, axes X1 and Z1 plus a rotary axis AY are moved. The rotary axis moves a rotatable arm with driven tools about the X axis in the Y-Z plane. The tools are always positioned in parallel to the X axis.

Restricted functionality of export versions:
Not possible.

Transformation: Swivel by 2 linear axes, Run MyCC /SW2A

Option; order code N45

Article No.: 6FC5800-0AN45-0YB0

Transformation: Swivel by 2 linear axes SINUMERIK Integrate Run MyCC /SW2A is a customized transformation. The general solution is SINUMERIK Integrate Run MyCC /PACO.

Restricted functionality of export versions:
Not possible.

→ Transformation: PARACOP 3 axes, Run MyCC /PACO

Glossary

Functions and terms

SINUMERIK 840D sl

Transformation: TRICEPT 5-axis transformation, Run MyCC /TRIC

Option; order code M46

Article No.: 6FC5800-0AM46-0YB0

SINUMERIK Integrate Run MyCC /TRIC is a kinematic transformation for a TRICEPT (tripod with center tube). TRICEPT kinematic transformations are used on parallel-kinematic machines PKM. Parallel kinematics means that the drive forces engage on the spindle head (Stewart platform) simultaneously (virtually in parallel). TRICEPT machines are TRIPOD systems on which the Stewart platform is moved by 3 actuators. Design measures are used to ensure that the Stewart platform cannot move in an undefined manner. On TRICEPT, an additional passive telescope (center tube) is used.

On TRICEPT, 2 additional rotary axes are required to define the tool orientation in space. These axes can be arranged like the fork head of a 5-axis machine, for example, thus allowing the machine to carry out 5-axis machining.

The SINUMERIK CNC calculates the required motion of the machine axes online. Therefore, the programmer can create part programs in the same way as on a conventional machine, and does not have to take the special kinematics of the machine into account.



TRICEPT

Restricted functionality of export versions:
Not possible.

Transformation: TRIPOD hybrid kinematics, Run MyCC /THYK

Option; order code N36

Article No.: 6FC5800-0AN36-0YB0

The kinematic transformation TRIPOD hybrid kinematics THYK is a 5-axis transformation for 3 parallel linear axes and 2 rotary axes. The tool movements are programmed complete with their orientation in the same manner as the Cartesian 5-axis machine. The real-time transformation calculates the necessary paths and speeds for the real machine axes.



Parallel kinematic type Exechon

Restricted functionality of export versions:
Not possible.

Transformation: W-Z Tracking, Run MyCC /WZTR

Option; order code N22

Article No.: 6FC5800-0AN22-0YB0

SINUMERIK Integrate Run MyCC /WZTR is a specific function and is not intended for general use.

Restricted functionality of export versions:

Not possible.

TRANSMIT/cylinder surface transformation

Option; order code M27

Article No.: 6FC5800-0AM27-0YB0

The TRANSMIT function is used for milling external contours on turned parts, e.g. square parts – linear axis with rotary axis. As a result, the programming process becomes much more simple, and complete machining increases machine efficiency: Turning and milling can be performed on one machine without rechucking.

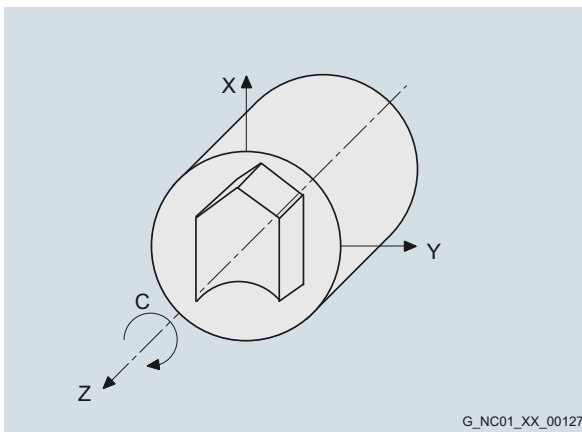
3D interpolation with 2 linear axes and one rotary axis is possible. The two linear axes are mutually perpendicular and the rotary axis lies at right angles to one of the linear axes. TRANSMIT can be called up in different channels simultaneously. The function can be selected and deselected with a preparatory function (straight line, helix, polynomial and activating tool radius compensation) in the part program or MDI.

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.

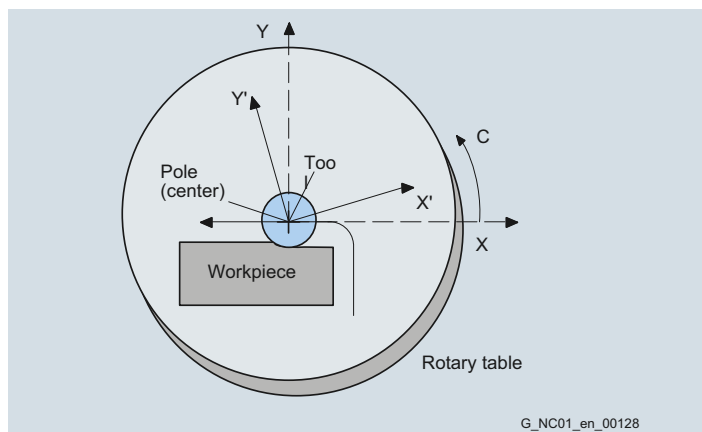
TRANSMIT through the pole is implemented in different ways:

- When traveling through the pole, the rotary axis is turned automatically through 180° when the turning center is reached and the remaining block is then executed.
- When traversing close by the pole, the CNC automatically reduces the feedrate 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.

Cylinder surface transformation is used on turning machines and milling machines, and enables cylinder surface machining, e.g. for turned parts. The cylinder surface transformation or TRACYL cylinder surface transformation can be used to manufacture grooves of any shape on the surface of cylindrical bodies with or without groove side offset. The shape of the grooves is programmed in reference to the unwound plane cylinder surface.



Face machining with TRANSMIT



Tool-center-point path through the pole

Glossary

Functions and terms

SINUMERIK 840D sl

Travel to fixed stop

With this function, tailstocks or sleeves, for example, can be traversed to a fixed stop in order to clamp workpieces. The pressure applied can be defined in the part program.

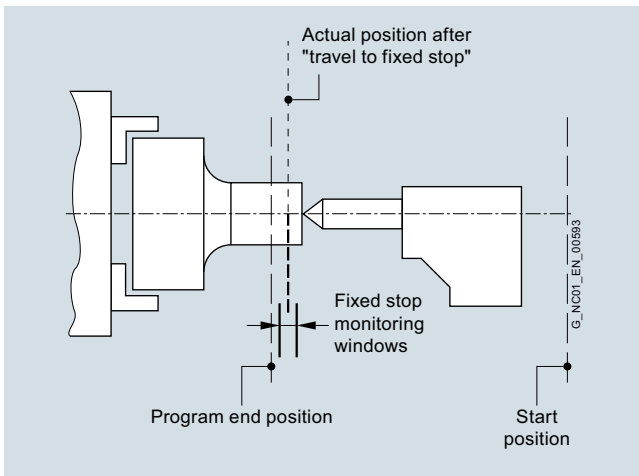
Several axes can be traversed to a fixed stop simultaneously and while other axes are traversing.

Travel to fixed stop with Force Control

Option; order code M01

Article No.: 6FC5800-0AM01-0YB0

The extended travel to fixed stop function can be used to adapt torque or force on a modal or non-modal basis; travel with limited torque/limited force (force control, FOC) can be initiated, or synchronized 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. When the default value is specified in the machine data field "Computational resolution for linear or angular position" (1000 increments per mm or degree), the ranges of values specified in the table can be programmed with this resolution:

	G70 [inches, degrees]	G71 [inches, degrees]
Linear axes X, Y, Z, ...	± 399999,999	± 999999,999
Rotary axes A, B, C, ...	± 999999,999	± 999999,999
Interpolation parameters I, J, K	± 399999,999	± 999999,999

If the computational resolution is increased/decreased by a factor of 10, then the value ranges change accordingly. The traversing range can be restricted by software limit switches and working areas.

Universal interpolator NURBS

Internal motion control and path interpolation are performed on the basis of non uniform rational B splines NURBS. 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 interpolation
- Circular interpolation
- Helical interpolation
- Involute interpolation
- Splines (A, B, C)
- Polynomial

Universal spatial compensation interface, Run MyCCI /UCI

Option; order code N75

Article No.: 6FC5800-0AN75-0YB0

SINUMERIK Integrate RunMy CCI /UCI provides an interface for customized compensation algorithms.

Restricted functionality of export versions:
Not possible.

Use HMI applications, Run MyHMI /3GL

Option; order code P60

Article No.: 6FC5800-0AP60-0YB0

SINUMERIK Integrate Run MyHMI /3GL is used to execute programmed HMI applications. The applications are programmed with either Qt/C++, .net (C#, VB) or C++.

This option is also needed if third-party software or background functions with data communication (application without HMI components) are to be integrated in SINUMERIK Operate.

→ [Program screens, operating areas and user interfaces, Create MyHMI /3GL](#)

Use HMI applications, Run MyScreens

Option; order code P64

Article No.: 6FC5800-0AP64-0YB0

The SINUMERIK Integrate Run MyScreens functionality allows SINUMERIK users to design their own screen forms in order to visualize either machine-manufacturer or end-user functional expansions.

This function is implemented via an integrated interpreter and via configuring files containing the description of the user interface. The screen forms can be designed directly on the CNC itself. Users can also write their own cycle forms that can also be called from the part program editor.

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 supplied toolbox.

The following functions can be implemented with the SINUMERIK Integrate Run MyScreens option:

- Display screen forms and provide softkeys, variables, tables, texts, help texts, graphics, and help screens
- Start actions when screen forms are displayed and exited, when softkeys are pressed, and values (variables) are entered
- Dynamic restructuring of screen forms, including changing softkeys, designing arrays, displaying, replacing and deleting display texts and graphics
- Read and write variables, combine with mathematical, comparative or logical operators
- Subprograms, file functions or program instance services (PI services)
- Enable data exchange between screen forms

SINUMERIK Integrate Run MyScreens is configured using ASCII files that can be stored on the PCU 50 or the NCU. Files that contain ASCII descriptions for the layout of interactive screen forms, softkey functions and display texts and graphics are interpreted. These configuring files are created with the ASCII editor, taking into account certain special rules of syntax.

The user interface can be expanded even in the basic version by up to 5 screen forms using predefined softkeys with the integrated editor. The option with order code P64 is required to add more than 5 screen forms with SINUMERIK Integrate Run MyScreens.

Glossary

Functions and terms

SINUMERIK 840D sl

Use standardized and freely configurable operating screens, Run MyHMI /PRO

Option; order code P47

Article No.: 6FC5800-0AP47-0YB0

The runtime license SINUMERIK Integrate Run MyHMI /PRO allows users to run configurations on the PCU 50 or the NCU that have been created with the HMI PRO CS configuration software. The software option includes the standardized operating screens within HMI PRO as well as the freely configurable user screens that have been created using the variable layout.

User interface

The user interface has a clear layout with 8 horizontal and 8 vertical softkeys. The specific use of windows-type technology permits simple and user-friendly machine operation.

The user interface is subdivided into 6 operating areas:

- Machine
- Parameters
- Program
- Program Manager
- Diagnosis
- Commissioning

This means that in parallel with part machining, for example, another part program can be created. The last active menu is always stored when changing the operating area.

User interface on SIMATIC Panel, Run MyHMI /SIMATIC OP for SIMATIC Comfort and Mobile Panels

Option; order code P03

Article No.: 6FC5800-0AP03-0YB0

SINUMERIK Integrate Run MyHMI /SIMATIC OP allows operation of the SINUMERIK NCU with its own user interface on SIMATIC Comfort Panels. The user interface required on the SIMATIC Comfort Panel can be created simply using the SIMATIC WinCC ES Advanced engineering system and the add-on SINUMERIK Integrate Create MyHMI /WinCC.

User machine data

The NC kernel makes machine data available for configuring the PLC user program. These user machine data are stored in the NCK-PLC interface during CNC power-up, prior to PLC power-up.

The PLC basic program reads these data from the NCK-PLC interface during its initialization phase. This means that specific machine configurations, machine expansions and user options can be activated.

Utility programs for the servo, Run MyCC /SUTI

Option; order code N10

Article No.: 6FC5800-0AN10-0YB0

SINUMERIK Integrate Run MyCC /SUTI includes a number of special functions in the position controller: Position controller gain factor in real time influences the position of the slave encoder taking into account the master-slave speed coupling.

Variable-based axis movement, Run MyCC /AMOV

Option; order code N62

Article No.: 6FC5800-0AN62-0YB0

SINUMERIK Integrate Run MyCC /AMOV is a specific function and is not intended for general use.

Restricted functionality of export versions:

Not possible

Variables and arithmetic parameters

Using variables instead of constant values allows flexible programs to be developed. Variables make it possible to respond to signals, e.g. measured values. If variables are used as a setpoint value, the same program can be used for different geometries.

The CNC uses 3 sorts of variables:

- User-defined variables
Variables defined by the user with name and type, e.g. arithmetic parameters
- Arithmetic parameters
Special, predefined arithmetic variables whose address is R plus a number. The predefined arithmetic variables are of the REAL type
- System variables
Variables provided by the CNC that can be processed in the program (write, read). System variables enable access to work offsets, tool offsets, actual values, measured axis values, CNC conditions, etc.

Variable type	Meaning	Value range
INT	Integers with sign	$\pm(2^{31} - 1)$
REAL	Real numbers (fractions with decimal point, LONG REAL in acc. with IEEE)	$\pm(10^{-300} \dots 10^{+300})$
BOOL	Boolean values: TRUE (1) and FALSE (0)	1.0
CHAR	ASCII character specified by the code	0 ... 255
STRING	Character string, number of characters in [...], maximum of 200 characters	Sequence of values with 0 ... 255
AXIS	Axis names (axis addresses) only	All axis identifiers in the channel
FRAME	Geometrical parameters for moving, rotation, scaling, and mirroring	

Velocity

The maximum path and axis velocity 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 IPO cycle is always limited to the maximum velocity or, in the case of short path lengths, reduced 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 velocities 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).

Velocity adaptation for water jet cutting, Run MyCC /VADA

Option; order code N05

Article No.: 6FC5800-0AN05-0YB0

Velocity adaptation for water jet cutting SINUMERIK Integrate Run MyCC /VADA

For 2D water jet cutting applications, it is possible to reduce the path velocity at corners and along circular contours as a function of the corner angle or circle radius. NC commands in the part program activate and deactivate the function and parameterize the velocity reduction.

Restricted functionality of export versions:

Not possible.

Glossary

Functions and terms

SINUMERIK 840D sl

Vibration extinction, Run MyCC /VIBX

Option; order code N11

Article No.: 6FC5800-0AN11-0YB0

The SINUMERIK Integrated Run MyCC /MIBX vibration extinction function supports axis-specific damping of machine vibrations. Up to 8 axes can be parameterized in the CNC, each with 2 machine data for the filter frequency and the required damping factor.

The function significantly reduces disturbing oscillations that follow positioning.

Restricted functionality of export versions:

Not possible.

Virtual axis

A virtual axis is one that is interpolated in the follow-up mode. Virtual axes are used in master value couplings. Typical examples are electronic transfer CP, master value coupling with curve tables or flying shears.

→ Axis/spindle

→ Positioning axis/auxiliary spindle

→ Simulation axis/spindle

Work offsets

According to DIN 66217, clockwise, rectangular (Cartesian) coordinate systems are used in machine tools.

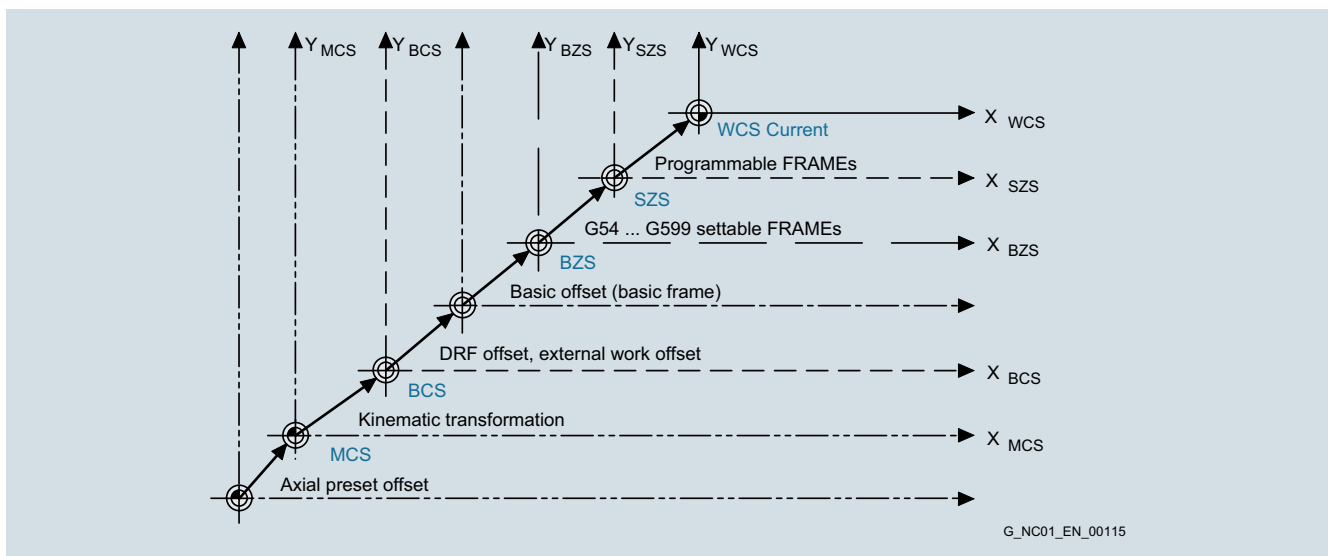
The following coordinate systems are defined:

- Machine coordinate system MCS
The machine coordinate system is made up of all physically available machine axes.
- Basic coordinate system BCS
The basic coordinate system consists of 3 Cartesian axes (geometry axes), as well as other non-geometric axes (special axes).
- BCS and MCS are always in conformance when the BCS can be mapped to the MCS without kinematic transformation (e.g. TRANSMIT/interfacial transformation, 5-axis transformation and max. 3 machine axes).
- Basic zero system BZS
DRF offsets, external work offsets and basic frames map the BCS on the BZS.
- Settable zero system SZS
An activated settable work offset G54 to G599 transfers the BZS to the SZS.
- Workpiece coordinate system (WCS)
The programmable frame determines the WCS, which is the basis for programming.

Work offsets are therefore used to transform the machine zero point into the workpiece zero point in order to simplify programming.

The following work offsets are possible:

- Settable work offsets:
It is possible to enter up to 100 work offsets (G54 to G57, G505 to G599), offset coordinates, angles and scaling factors in order to call zero points program-wide for various fixtures or clamping operations, for example. The work offsets can be suppressed block-by-block.
- Programmable work offsets:
Work offsets can be programmed with TRANS (substitution function, basis G54 to G599) or ATRANS (additive function). This allows you, for example, to work with different work offsets for repetitive machining operations at different positions on the workpiece. G58/G59 make previously programmed work offsets axially replaceable.
- External work offsets:
Axis-related linear work offsets can also be activated via the PLC user software (function blocks) with assignment of system variable \$AA_ETRANS [axis].



Coordinate system

→ [Frame concept](#)

Working area limitation

Working area limitations define the area in which machining is permitted. These limitations refer to the basic coordinate system BCS. The tool tip is monitored to determine whether it is inside the protected working area (taking into account the tool radius).

One value pair per axis (\pm) 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 traversing range of the axes in addition to the limit switches. Protection areas in which tool motion is inhibited and which protect equipment such as tool turrets, measuring stations, etc., against damage, are thus set up in the machine's working area.

→ [Work offsets](#)

Glossary

Functions and terms

SINUMERIK 840D sl

Working plane

When specifying the working plane in which the desired contour is to be machined, the following functions are defined at the same time:

- Plane for the tool radius compensation
- Infeed direction for the tool length compensation depending on the type of tool
- Plane for the circle interpolation

When calling the tool path correction G41/G42, the working plane must be defined so that the CNC 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.

→ [Tool radius compensation](#)

Workpiece-related actual value system

The workpiece-related actual value system of the SINUMERIK CNC enables the user to do the following:

- After the CNC has powered up, start machining in a workpiece coordinate system defined via machine data in JOG and AUTO CNC operating modes without any additional operator actions
- Retain the valid settings relating to the active plane, settable frames (G54 ... G57), kinematic transformations, and active tool offset at the end of the part program for use in the next part program
- Toggle between the workpiece coordinate system WCS and the machine coordinate system MCS
- Change the workpiece coordinate system, e.g. by changing the settable frames or the tool offset.