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 (ASP)

The function SINUMERIK Integrate Access MyMachine /Ethernet (ASP) provides an overview of the machine's history and permits the uploading and downloading of machine data, files, tachographs and configurable PLC traces. It also supports notification of irregular machine states per SMS or email. SINUMERIK Integrate Access MyMachine /Ethernet (ASP) thus supports cost-effective monitoring of fault conditions and can be fully integrated into the user's existing service and maintenance processes.

Access MyMachine /P2P

Option 6FC5800-0AP30-0YB0 / P30

The function SINUMERIK Integrate Access MyMachine /P2P permits remote access to the SINUMERIK HMI for the purpose of quick diagnosis of the machine's condition. It supports data uploads and downloads, analog and ISDN telephone links as well as access via the Internet. Access MyMachine provides for increased machine availability by virtue of speedy "online presence". Furthermore, it enables users to prepare essential service callouts more effectively.

Access MyTool ID

Option 6FC5800-0AP52-0YB0 / P52

→ Tool identification for loading/unloading tools by means of code carrier

SINUMERIK Integrate Access MyTool ID allows tool identification systems to be linked up via PROFIBUS for SINUMERIK in conjunction with the standard tool management system, and also supports the transfer of tool data. It therefore eliminates the need for error-susceptible manual inputs.

Access protection

Protection level	**Type**	**PLC DB10 DBB 56 bit...**	**User**	**Access to (examples)**
0	Password	–	Siemens	All functions, programs, data
1	Password	–	Machine manufacturer: Development	Defined functions, programs and data (options)
2	Password	–	Machine manufacturer: Commissioning engineers	Defined functions, programs and data (machine data)
3	Password	–	End user: Service	Assigned functions, programs and data
4	Red key Switch position 3	7	End user: Programmer, machine setter	< Protection levels 0 to 3 machine manufacturer/end user
5	Green key Switch position 2	6	End user: Qualified operator who is not a programmer	< Protection levels 0 to 3 end user
6	Black key Switch position 1	5	End user: Trained operator who is not a programmer	Program selection only, tool wear entries and work offset entries
7	Switch position 0	4	End user: Semi-skilled operator	No input and program selection possible, only the machine control panel can be operated

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

These are subdivided into:

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

SINUMERIK controls thus provide a multi-level concept for controlling access rights.

Protection level 0 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 0 to 3 are preprogrammed by default through Siemens.

An entered password takes precedence over a keyswitch position, and machine manufacturers or end users can change access rights for protection levels 4 through 7.

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

Action log

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

SINUMERIK 840D sl Type 1B

Actual-value system for workpiece

The term "actual-value system for workpiece" is used to designate functions, which allow the SINUMERIK user to:

- begin machining in a workpiece coordinate system defined via machine data in JOG and AUTOMATIC mode without any additional manipulations after powering up the control
- retain the valid settings relating to active level, settable frames (G54-G57), kinematic transformations, and active tool offset at the end of the part program for use in the next part program
- toggle between the workpiece coordinate system and the machine coordinate system
- change the workpiece coordinate system (e.g., by changing the settable frames or tool offset)

Advanced Position Control (APC)

Option 6FC5800-0AM13-0YB0 / M13

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 KV factor, improves the surface and therefore increases productivity.

Advanced Surface

Option 6FC5800-0AS07-0YB0 / S07

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 at high workpiece output rates.

Alarms and messages

- Alarms and messages:
  All messages and alarms are output separately on the operator panel in plain text with the date and time and a symbol indicating the cancel criterion. Alarms and messages are displayed separately. 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 give the user information on the current processing status while the program is executing. Message texts may be up to 124 characters long, and are displayed in two 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, you can also set alarms in a CNC program. An alarm always goes hand in hand with a response from the control according to the alarm category.

You will find a list of reactions associated with specific alarms in the Commissioning Manual. The alarm text must be configured. Alarm numbers 65000 to 67999 are reserved for the user.

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

Analog value control

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

Analyze MyCondition

SINUMERIK Integrate Analyze MyCondition provides test cycles for testing synchronized axes and universal axes and for performing circularity tests, and also offers functionality for continuous data acquisition during the production process. It also supports the reporting of parameters about wear and tear of mechatronic components. By employing a condition-oriented maintenance routine, you will keep your machines running longer as well as reduce 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 where the motion sequence is the message to be conveyed

Asynchronous subprograms

→ Interrupt routines with fast retraction from the contour

An asynchronous subprogram (ASUB) is a CNC program, which can be started based on an external event (e.g., a digital input) or from the PLC. Inputs are allocated to 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).

Auto Servo Tuning

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 "adaptation 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.

Auxiliary function output

Auxiliary function output informs the PLC on time 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 retracted 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-like 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 two leading axes (in different coupled-axis groupings) simultaneously.
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 two coupled-axis groupings to machine the two sides of a workpiece.

SINUMERIK 840D sl Type 1B

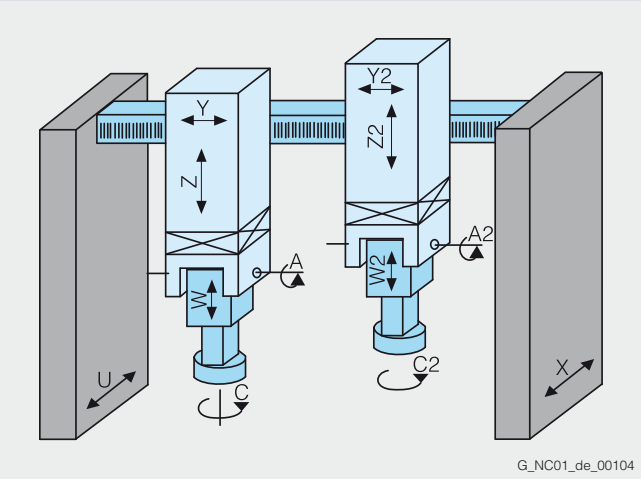
Axes/spindles

Option 6FC5800-0AA00-0YB0 / A01 ... A28

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

Axial coupling in the machine coordinate system MCS

→ Generic couplings



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

The participating axes can be reconfigured following Reset.

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

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

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

The loadable compile cycle PROT can be used to provide collision protection for machining heads.

→ Axis collision protection, PROT

Axis collision protection, PROT

Option 6FC5800-0AN06-0YB0 / N06

The loadable compile cycle supports collision protection of up to 12 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.

Axis data output via PROFIBUS, ADAS

Option 6FC5800-0AN07-0YB0 / N07

The loadable compile cycle enables axis and spindle data to be output to a special PROFIBUS slave 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.

The compile cycle in the NC kernel then sends up to 30 axis data items (of 4 bytes each) to the slave in each PROFIBUS cycle. With the slaves in isochronous mode, the transfer cycle can be equal to the position-control cycle, or a multiple of it.

Axis limitation from the PLC

→ Protection areas

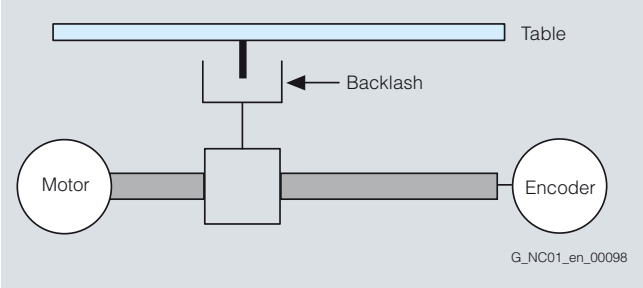
The preactivation of protection areas with specification of a position offset is programmed in the part program. You can put the preactivated protection areas into effect in the PLC user program via the PLC interface. As a result, the relevant protection area is activated, for example, before a tool probe is swiveled into position in the working area, in order to monitor 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.

Axis/spindle interchange

An axis/a spindle is permanently assigned to a specific channel via machine data. The axis/spindle interchange function can be used to release an axis/a spindle (RELEASE) and to assign it to another channel (GET), i.e., to interchange the axis/spindle. The relevant axes/spindles are determined via machine data.

Backlash compensation



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

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

Bidirectional compensation

→ Leadscrew error compensation

SINUMERIK 840D sl Type 1B

Block search

The block search function allows any point in the part program to be selected, at which machining must start or be continued. The function is provided for the purpose of testing part programs or continuing machining after a program abort. Cascaded block search is also possible.

You have a choice of 4 different search options:

- With calculation at 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 you to approach a target position (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 using the method of interpolation valid in the target block.
- Without calculation:
  This method is used for high-speed searches in the main program. No calculations are carried out during the search. The internal control values remain the same as before the block search.
- External block search without calculation:
  You can use the "External without calc." softkey in the "Search position" and "Search pointer" menus to start an accelerated block search in programs that are executed from an external device (local hard disk or network drive).

You can specify the search destination 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

Cartesian PTP travel

For handling and robot-related tasks, two types of movement are required, either in the Cartesian coordinate system (continuous path, CP), or as a point-to-point (PTP) movement. 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 the changing of an arm position during handling.

PTP travel is also possible in JOG 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.

CCG compiler (cam contour grinding, non-circular grinding)

Option 6FC5800-0AP10-0YB0 / P10

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 PCU50 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 control system.

The CCG compiler generates CNC programs in polynomial format to allow the machining of non-circular contours on a cylindrical grinding machine. The lift curves used commonly 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 SINUMERIK system.

Use of the option is conditional upon the availability of the options TRANSMIT (M27) and polynomial interpolation (M18).

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 dimensioning of a drawing is chosen 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.

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, intermediate point and end point.

Clamping monitoring

→ Position monitoring, standstill monitoring

Clamping monitoring is one of SINUMERIK's many extensive monitoring mechanisms for axes.

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 monitor responds, its reactions are the same as those of the standstill monitor.

Clearance control 1D in the IPO cycle

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

Clearance control 1D/3D in position-control cycle, CLC

Option 6FC5800-0AM40-0YB0 / M40

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

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

Restricted functionality of export control versions:

Clearance control 1D in the position-control cycle only, number of interpolating axes restricted to 4.

CNC high-level language

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

System variables

The system variables ($.) can be processed in the CNC program (read, partially write). System variables allow access to, 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 gives you the option of executing all CNC functions and addresses as variables. The names of the variables can be freely defined by the user. Read and write 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 data (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
- Square root
- Absolute value
- Power of 2 (squaring)
- Integer component
- Round to integer
- Natural logarithm
- Exponential function
- Offset
- Rotation
- Scale modification
- Mirroring

Comparison and logic operations

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

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 according to the single instructions.

Control structures

The control 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 program messages

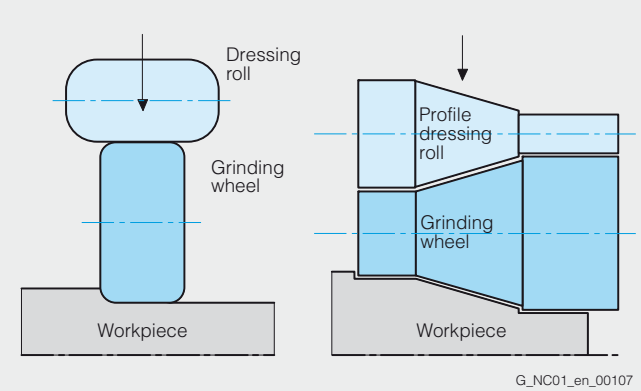
→ Alarms and messages

All messages programmed in the part program and all alarms recognized by the system are displayed on the operator panel in plain text. Alarms and messages are displayed separately. You can program messages in order to provide the operator with the latest information on the current machining situation during the program run.

CNC program transfer

→ Manage MyPrograms

Continuous dressing (parallel dressing)



Parallel dressing

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

CNC user memory

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

CNC user memory, additional

Option 6FC5800-0AD01-0YB0 / D01 ... D06

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

COA interface for compiled OEM cycles, COOC

Option 6FC5800-0AM67-0YB0 / M67

This loadable compile cycle serves as an interface for the OEM's own developments and permits expansions to the program interpreter functionality, i.e. for the purpose of implementing special part program sequences or special calculations in C++.

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.

Communication of Look Ahead data, COLA

Option 6FC5800-0AN61-0YB0 / N61
(on request)

Compensation of a forced mechanical coupling, AXCO

Option 6FC5800-0AM81-0YB0

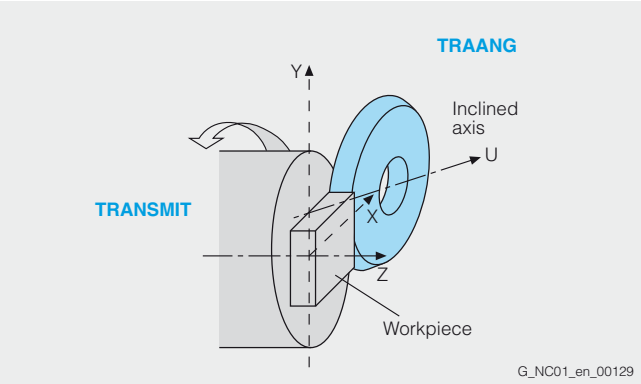
The loadable compile cycle allows an axis motion that occurs due to mechanical coupling of an axis to a following axis, to be compensated such that the axis remains mechanically stationary despite the coupling. The motor of the coupled axis is rotated according to the set coupling ratio.

Computational motion control EMC

Option 6FC5800-0AN47-0YB0 / N47
(on request)

SINUMERIK 840D sl Type 1B

Concatenated transformations



Grinding a TRANSMIT contour with inclined axis

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

Applications:

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

Continue machining at the contour (retrace support)

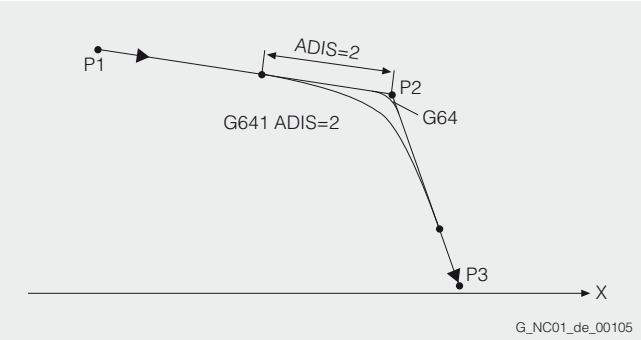
Option 6FC5800-0AM24-0YB0 / M24

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 a view of the workpiece) following an interruption in machining without exact knowledge of the part program in order to continue machining the workpiece from there.

The Continue machining at the countour 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 the reverse travel. Continue machining is used, for example, when the machine operator only notices the failure or interruption a few blocks after the actual interruption. The head has usually already progressed further in the machining, and must, therefore, be appropriately returned for continuation of machining.

Continuous-path mode with programmable rounding clearance



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 on non-tangential transitions. A soft contour transition without a jump in acceleration can be programmed with G641 ADIS=...

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 6FC5800-0AM08-0YB0 / M08

→ Feedrate interpolation

When the contour handwheel function is activated, the handwheel has a velocity-generating effect in AUTOMATIC and MDI 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
  Against the programmed direction of travel (continuation beyond the start of the block is prevented)

Contour monitoring

→ Travel to fixed stop

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.

Contour monitoring with tunnel function

Option 6FC5800-0AM52-0YB0 / M52

With the "contour monitoring with tunnel function" 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.

SINUMERIK 840D sl Type 1B

Control unit management

The 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 with one TCU each. Advantages:

- 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 on 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 (328 ft) (max. distance between two network nodes).

Operation of SINUMERIK via the VNC viewer requires a confirmation on the SINUMERIK operator panel that operation is now permitted via the VNC viewer. When a SINUMERIK operator panel is not used, this option can be used to suppress scanning of the confirmation.

	NCU CNC SW 2.7 SP1 / 4.4 SP1	**PCU SINUMERIK Operate 2.7 SP1**
Control unit management		
2/4 TCUs with interlocking of simultaneous operation by 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		
Downzoom	–	–
Dynamic resolution switchover	•	•
Activate/deactivate MCP		
MCP PROFIBUS	–	–
MCP IE, PN (IE mode)	•	•
MCP PN (PN mode)	–	–
One or several TCUs which can be switched over via several NCUs and PCUs (1 × T : n × M)		
Via actuation of 2 keys	•	•
Via channel menu	•	•

Options for multi-user operation (n × T : M : N)

Legend:
T = Number of TCUs
M = Number of MMCs / HMIs
N = Number of NCUs
– = No
• = Yes

Crank interpolation, CRIP

Option 6FC5800-0AN04-0YB0 / N05

Crank interpolation 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.

Create MyCC

→ Run MyCC

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

Create MyCCI

6FC5863-1YP00-0YB8

SINUMERIK Integrate Create MyCCI supports the development of loadable compile cycles based on customized interfaces without requiring special hardware as a development environment. For this special application, the customer uses software (GNU compiler and linker) in an environment known as "Cygwin software shell" on a Windows PC. The concept allows the OEM to develop real-time applications in C/C++ and to load them as customized compile cycles.

Use of this application is conditional upon installation of the corresponding interface as a loaded compile cycle on the control system.

→ Run MyCCI

Create MyConfig

The engineering software SINUMERIK Integrate Create MyConfig supports the automated commissioning of machines with SINUMERIK. 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 user benefits from 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.

Create MyHMI

→ Create MyHMI /3GL

→ Run MyHMI

→ Run MyHMI /3GL

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.

Create MyHMI /3GL

The SINUMERIK Integrate Create MyHMI /3GL programming package allows SINUMERIK users to design their own user interfaces in order to visualize either machine-manufacturer or end-user functional expansions or simply their own screen form layouts.

User interfaces programmed by Siemens or other machine manufacturers can be modified or replaced. This function is realized in a high-level language development environment based on C++/QT that can generate a platform-independent execution code for Windows XP (SINUMERIK PCU 50) and Linux (NCU).

The screen forms are created platform-independently in the development environment.

The compiled program parts are transferred to the corresponding user directories of the PCU 50 or NCU.

Configuring 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.

You can implement the following functions 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

The SINUMERIK Integrate Create MyHMI /3GL (option P60) runtime license is required to run the programmed user screens.

SINUMERIK 840D sl Type 1B

Create MyInterface

With the SINUMERIK Integrate Create MyInterface option, it is possible to integrate SINUMERIK controls into an existing manufacturing network in an easy process using defined interfaces. All key data of the machine are exchanged between a higher-level control system and the SINUMERIK control, helping to reduce machine downtimes and increase machine capacity utilization.

Cross-mode actions

Option 6FC5800-0AM43-0YB0 / M43

→ Interrupt routines with fast retraction from the contour

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

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

Cycle protection

→ Lock MyCycles

Cycle support

→ SINUMERIK Operate runtime license OA Easy Screen

The technology cycles for drilling, milling and turning and the measuring cycles are supported by cycle screens. Similar input displays are also available for geometric contour programming. Users can also define a number of softkeys, input fields and displays themselves using "SINUMERIK Operate runtime license OA Easy Screen".

Cycles, overview (couplings)

Overview of cycles (couplings)		
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

SINUMERIK 840D sl Type 1B

Cycles, overview (interpolation, measurement)

Overview of cycles (interpolation, measurement)		
Advanced Surface	6FC5800-0AS07-0YB0	CYCLE832 High-speed settings
Measuring cycles	6FC5800-0AP17-0YB0	CYCLE961 Milling measurement: corner – right-angled corner/any corner
		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 – groove/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 cutters, drills)
		CYCLE982 Turning measurement: measure tools – calibrate tool probe
		CYCLE994 Turning measurement: inner/outer diameter
		CYCLE998 Milling measurement: measure 3D – align plane
		CYCLE998 Milling measurement: edge distance – align edge
		CYCLE971 Milling measurement: measure tools
		CYCLE971 Milling measurement: measure tools – calibrate tool probe
		CYCLE997 Milling measurement: measure 3D – 1 sphere / 3 spheres
Measure kinematics	6FC5800-0AP18-0YB0	CYCLE996 Milling measurement: measure 3D – measure kinematics

Cycles, overview (programming language)

Overview of cycles (programming language)		
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

Cycles, overview (programming support)

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: spigot – multi-edge
		CYCLE800 Milling: swivel plane, swivel 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
		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 with 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 with 802D sl: thread undercut form A, B, C, D
		CYCLE97 Turning compatibility with 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 ASUP for supporting special modes
		SLOT1 Milling: slot – longitudinal slot
		SLOT2 Milling: slot – circumferential slot
Pocket milling with free contour definition and islands	Basic scope	CYCLE63 Mill contour pocket: residual material detection
		CYCLE64 Predrill contour pocket: residual material detection
Residual material detection and machining for contour pockets and stock removal	6FC5800-0AP13-0YB0	CYCLE63 Mill contour pocket: residual material detection
		CYCLE64 Predrill contour pocket: residual material detection
		CYCLE952 Stock removal along contour: residual material detection

Overview of cycles (programming support)		
ShopMill/ShopTurn 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 Center contour pocket
		E_CP_CO Broach contour pocket
		E_CP_DR Predrill contour pocket
		E_CP_RE Contour pocket residual material
		E_CR_HEL Helix
		E_DR Drilling
		E_DR_BGF Drill thread milling
		E_DR_BOR Boring
		E_DR_PEC Deep-hole drilling
		E_DR_REA Reaming
		E_DR_SIN Countersinking
		E_DR_TAP Tapping
		E_END End of program
		E_HEAD Program header
		E_MANAGE Management
		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
ShopMill/ShopTurn 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 Center contour pocket
		F_CP_CO Broach contour pocket
		F_CP_DR Predrill 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
		F_DR_REA Reaming
		F_DR_SIN Countersinking
		F_DR_TAP Tapping
		F_DRILL Centric deep-hole drilling
		F_DRILLC Centric centering
		F_DRILLD Centric drilling
		F_END End of program
		F_FR_PEC Deep-hole drilling
		F_GROOV Groove
		F_HEAD Program header
		F_HOME Tool change point
		F_MANAGE Management
		F_MC_L Calibrate workpiece probe: length
		F_MC_MPT Calibrate tool probe: calibrate (tool probe)
		F_MC_R Calibrate workpiece probe: radius
		F_MC_RR Calibrate workpiece probe: radius in ring
		F_MC_RSP Calibrate workpiece probe: radius on sphere
		F_MC_SL Measure workpiece slot
		F_PO_CIR Circular pocket

Overview of cycles (programming support)		
ShopMill/ShopTurn machining step programming (ShopTurn) (continued)	6FC5800-0AP17-0YB0	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_RELEAS Retraction
		F_ROT_C C Translation
		F_ROU_Z Extended stock removal
		F_ROUGH Stock removal corner
		F_S_ASUP Actions after block search
		F_SIM Simulation
		F_SL_CIR Circumferential slot
		F_SL_LON Longitudinal slot
		F_SP_CHA Chamfer
		F_SP_EF Form E+F undercut
		F_SP_INI Initialization
		F_SP_IS Intersection check
		F_SP_RP Approach/retraction
		F_SP_RP2 Calculate approach/retraction
		F_SP_RPT Retraction data
		F_SP_SIM Simulation calculations
		F_SP_TRA Transformation management
		F_SUB_SP Counterspindle
		F_SWIV_H Swiveling
		F_TAP Centric tapping
		F_TCARR Swiveling
		F_TD Manual tool change
		F_TFS Tool change/feedrate/spindle
		F_TR_CON Cutting taper thread
		F_TR_LON Cutting longitudinal thread

Data exchange between machining channels

→ CNC high-level language

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.

Diagnostics functions

For service purposes, a self-diagnostics program and testing aids have been integrated in the controls. The status of the following can be displayed on the operator panel:

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

For testing purposes, signal combinations can be set for the output signals, input signals, and bit memories. All alarms and messages are displayed in plain text on the operator panel along with the corresponding acknowledgement 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

Dimensions metric/inches

Depending on the measuring system used in the production drawing, you can program workpiece-related geometrical data in either metric measure (G71) or inches (G70). The control can be set to a basic system regardless of the programmed dimensional notation. You can enter the following geometrical data directly and let the control convert them into the other measuring 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, you can switch back and forth between inch and metric dimensional notation using a softkey.

Display functions

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

- Block currently being executed
- Previous and following block
- Actual position, distance-to-go
- Current feedrate
- Spindle speed
- G functions
- Auxiliary functions
- Workpiece designation
- 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)

→ Handwheel override

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.

Drive current measurement, KPXT

Option 6FC5800-0AM82-0YB0 / M82

This function is used to optimize the speed controller of rotary axes where these are required to move workpieces of widely varying weights as part of the machining process.

Drive-autonomous extended stop and retract ESR

→ Extended stop and retract

Dynamic preprocessing memory (FIFO)

The traversing blocks are readied prior to execution and stored in a preprocessing memory (FIFO = first in/first out) of specifiable size. In contour sections that are machined at high velocity with short path lengths, blocks can be executed from this preprocessing memory at very high speed. The preprocessing memory 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 STOPRE (stop preprocessor) can be programmed.

Easy Screen

→ Run MyScreens

Electronic gear

→ Generic couplings

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 control versions:

The number of simultaneously traversing axes is restricted to 4.

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 AUTOMATIC and MDI 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, etc.).

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

Electronic Key System

Option 6FC5800-0AP53-0YB0 / P53

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

SINUMERIK 840D sl Type 1B

Electronic transfer CP

Option 6FC5800-0AM76-0YB0 / M76

→ Position switching signals/cam controller

→ Polynomial interpolation

→ Generic couplings

→ Cross-mode actions

→ I/O interfacing via PROFIBUS DP

→ Synchronized actions stage 2

→ Pair of synchronous axes (gantry axes)

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 motions. The "Electronic transfer CP" option makes it possible to control sequences of motion in transfer systems (e.g., gripper or suction lines), depending on a master value, which corresponds to the current ram position of the press. The "Electronic transfer CP" option includes the options:

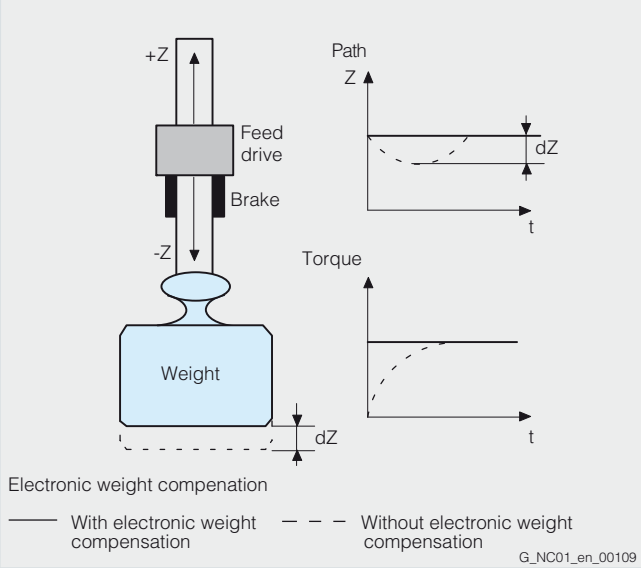
- Position switching signals/cam controller
- Polynomial interpolation
- Generic coupling CP Comfort
- Cross-mode actions
- I/O interfacing via PROFIBUS DP
- Synchronized actions stage 2
- Pairs of synchronous axes (gantry axes)

Combinations of these individual options satisfy all requirements for highly dynamic and accurate transfer controls. When using the "Electronic transfer CP" option, the "Spindle" and "Tool offset" functions cannot be activated.

Restricted functionality of export control versions:

The number of simultaneously traversing axes is restricted to 4.

Electronic weight counterbalance



Electronic weight counterbalance

With weight-loaded axes without mechanical or hydraulic weight counterbalance, the vertical axis drops when the brake is released and the controller 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.

Evaluation of internal drive variables

Option 6FC5800-0AM41-0YB0 / M41

The evaluation of internal drive variables can be used to control (adaptive control) a second process variable (such as a path-specific or axis-specific feedrate) depending on a measured process variable (such as spindle current).

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

Evaluation of internal drive variables is a prerequisite 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

Extended stop and retract ESR

→ Extended stop and retract

Extended stop and retract ESR, drive-autonomous

Option 6FC5800-0AM60-0YB0 / M60

The function **drive-autonomous** extended stop and retract 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-system-guided ESR function by the autonomous, purely axial stop and retraction motions of the drive which take no account of any NC coupling rules.

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

Extended stop and retract ESR (NC-controlled + drive-autonomous)

Option 6FC5800-0AM61-0YB0 / M61

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

These functions are primarily used for gearing and grinding technologies.

Extrapolated switching signals, XOUT

Option 6FC5800-0AN51-0YB0 / N51

CFK (Carbon Fiber-Enhanced Plastics) is finding wider application, especially in aircraft assembly, and the "fiber placement" machines involved in the process make high demands in terms of switching position precision. The compile cycle XOUT switches up to 64 output signals (8 output bytes). The outputs switch as soon as the machine axis reaches the preprogrammed switching positions.

2 different modules can be used depending on the level of accuracy required.

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

- The TM17 DRIVE-CLiQ module supports switching signal output with micro-second precision.
- The ET 200S PROFIBUS module supports switching signal output that is synchronized with the PROFIBUS DP cycle (position control cycle).

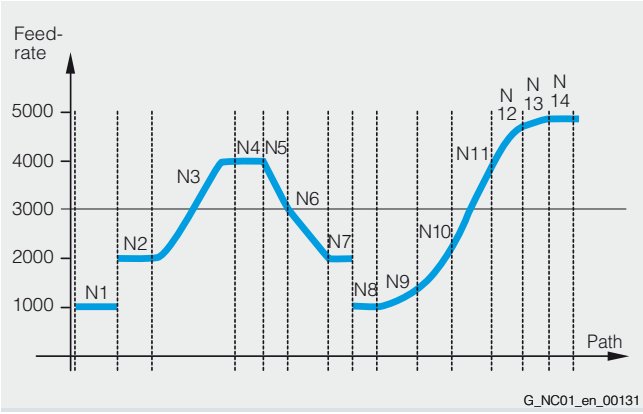
Feedforward control

The 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 (basic version):
  In velocity-dependent feedforward control, the following error can be reduced almost to zero at constant velocity.
- Acceleration-dependent torque feedforward control (option):
  In order to achieve precise contours even when the demand for dynamics is at its highest, you can use torque feedforward control. If the settings are right, you can compensate the following error almost completely, even during acceleration. The result is excellent machining precision even at high tool path feedrates.

Feedrate interpolation (feed characteristic)

→ Polynomial interpolation



Programming example for feedrate interpolation

Legend:
N1 Constant feedrate profile F1000: FNORM
N2 Abrupt set 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

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 extended by 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 as per 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 current 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 by a spline. The spline starts and ends tangentially to the previous or following feedrate setting.
- FPO
  You can also program the feedrate profile directly via a polynomial. The polynomial coefficients are specified analogous to polynomial interpolation.

Feedrate override

The programmed velocity is overridden by the current velocity setting via the machine control panel or by the PLC (0 % to 200 %). In order for the cutting velocity on 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 on 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 traverse paths are updated in the display. Standstill, clamping and positioning monitoring functions are not effective in follow-up mode. Once follow-up mode is cancelled, reference-point approach of the axis does not have to be repeated.

Frame concept

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

On SINUMERIK controls, 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 easily 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 three 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 rotation and swiveling of the machining plane
- By machining of inclined surfaces using tool length and tool radius compensation

Generator operation

With the generator operation function, brief power outages can be bridged or power provided for retraction. To make this possible, the energy stored during spindle rotation or axis movement is fed back into the DC link, following the same principle as that used by generators.

Generic couplings

We offer 5 different performance levels for generic (general) coupling (CP) of axes/spindles. The functionality is scalable via the number of master axes to one slave 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").

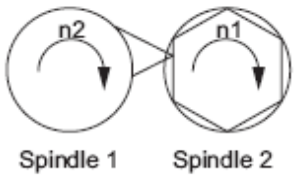
Restricted functionality with SINUMERIK 840DE sl: see the functional limitations for each of the above-mentioned functions and options.

SINUMERIK 840D sl Type 1B

Generic coupling Basic: CP Basic

Option 6FC5800-0AM72-0YB0 / M72

- Up to 4 x simple coupled motion and
- Up to 1 x "Synchronous spindles/multi-edge turning" or "Master value coupling/curve table interpolation" or "Axial coupling in the machine coordinate system".



Generic coupling Comfort: CP Comfort

Option 6FC5800-0AM73-0YB0 / M73

- Up to 4 x simple coupled motion and
- Up to 4 x "Synchronous spindle/multi-edge turning" and/or "Master value coupling/curve table interpolation" and/or "Axial coupling in the machine coordinate system".

Also:

- 1 x → "Electronic gear" for up to 3 leading axes is possible (without curve table interpolation and without cascading).

Generic coupling Expert: CP Expert

Option 6FC5800-0AM74-0YB0 / M74

- Up to 8 x simple coupled motion and
- Up to 8 x "Synchronous spindles/multi-edge turning" and/or "Master value coupling/curve table interpolation" and/or "Axial coupling in the machine coordinate system".

Also:

- Up to 8 x "Electronic gear " for up to 3 leading axes as well as
- Up to 5 x "Electronic gear" for up to 5 leading axes (each with curve table interpolation and with cascading)

The CP expert option is possible in combination with the NCU 720/730.

Generic coupling Standard: CP Standard

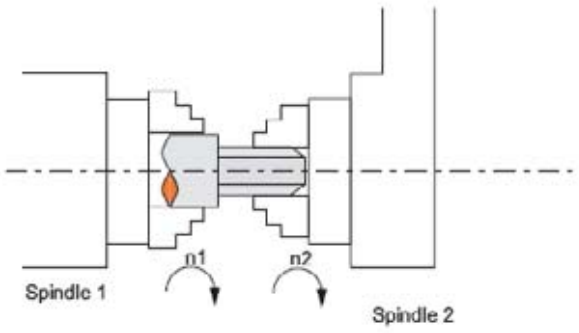
Basic version

- Up to 4 x simple coupled motion (with one leading axis, not used with synchronized actions)

Generic coupling Static: CP Static

Option 6FC5800-0AM75-0YB0 / M75

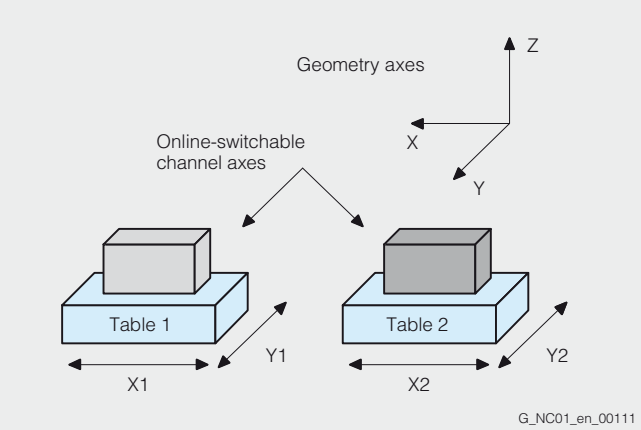
- One simple synchronous spindle (with coupling ratio 1:1, no multi-edge machining)



Geometrically redundant axes, GRED

Option 6FC5800-0AN73-0YB0 / N73
(on request)

Geometry axes, switchable online in the CNC program



Geometry axes, switchable online

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

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 during processing of a part program in AUTOMATIC and MDI modes, but can also be effective immediately after power-up of the control, on reset, and at the end of the part program, and remain in force beyond all mode changes (depending on the machine data).

Handling package

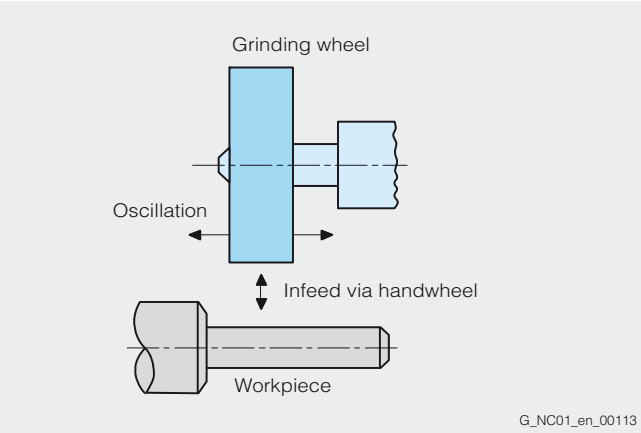
Option 6FC5800-0AS31-0YB0 / S31

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

- 3 additional axes
- 3 additional channels
- Transformation: Handling (option M31) (requires a loadable compile cycle)
- Synchronized actions stage 2 (option M36)
- No tool offsets and no spindles

SINUMERIK 840D sl Type 1B

Handwheel override



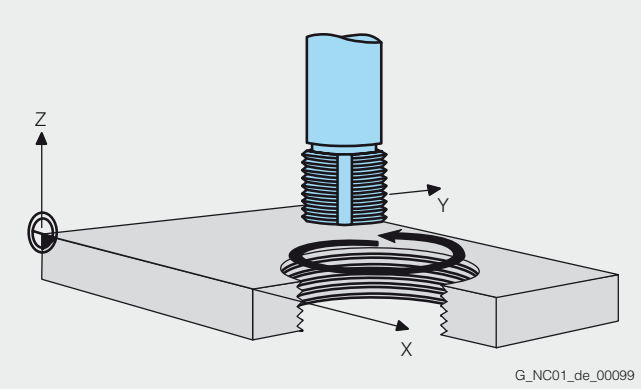
Handwheel override in AUTOMATIC mode

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

Helical interpolation 2D+6



Helical interpolation: Thread milling with form cutter

The helical interpolation function is ideal for machining internal and external threads using form milling cutters and for milling lubrication grooves. The helix comprises two motions:

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

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

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

Restricted functionality of export control versions: not possible.

High-speed CNC inputs/outputs

→ Position switching signals/cam controller

The "high-speed CNC inputs/outputs" function makes it possible to read in or to output signals in the position control/interpolation cycle.

The high-speed CNC inputs/outputs can be used for machines, such as those used for grinding and laser machining, as well as in SINUMERIK Safety Integrated.

Input signals are possible for the following:

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

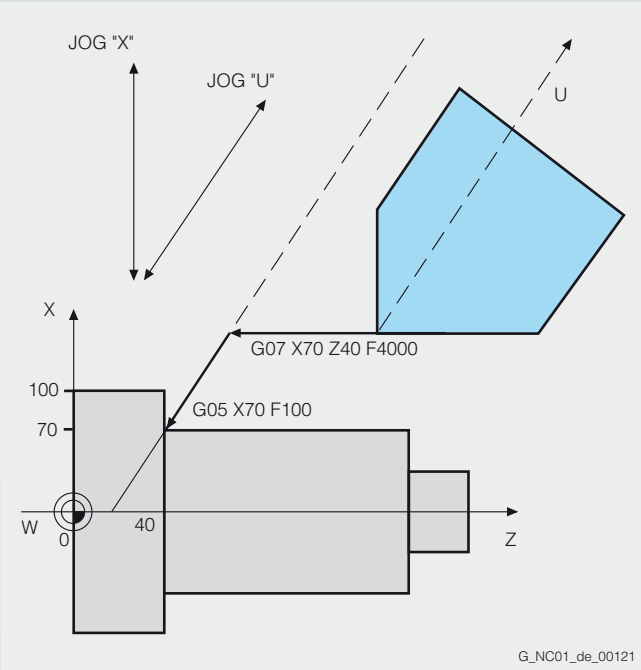
I/O interfacing via PROFIBUS DP

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.

SINUMERIK 840D sl Type 1B

Inclined axis

Option 6FC5800-0AM28-0YB0 / M28



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

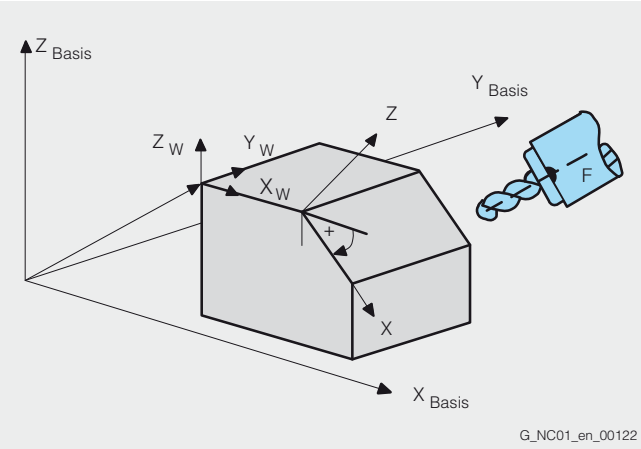
The "Inclined axis" function is used for fixed-angle interpolation using an oblique infeed axis (used primarily in conjunction with cylindrical grinding machines). The axes are programmed and displayed in the Cartesian coordinate system.

Tool offsets and 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 JOG 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).

Inclined-surface machining with frames

→ Frame concept



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.

Integrate screens into the user interface

→ Run MyScreens

Integrated tool monitoring and diagnostics, IMD base

Option 6FC5800-0AN13-0YB0 / N13

This option supports the "Tool missing" monitoring function as well as programming of the maximum permissible overload for a tool.

The parameters are taught on program setup as for the "Tool missing" 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", …, ...)

The compile cycle "IMD base" 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

The compile cycle "Integrated Monitoring & Diagnostics" 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 light

Option 6FC5800-0AN12-0YB0 / N12

This option supports a "Tool missing" monitoring function within the IMD package. For this purpose, cuts (transition from rapid traverse to feedrate) must be selected and parameterized in the part program.

(CC_START_TASK("MissingTool", …, ...). 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

→ Tool radius compensation

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

Interrupt routines with fast retraction from the contour

Option 6FC5800-0AM42-0YB0 / M42

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 (for example, 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

On the SINUMERIK, 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 6FC5800-0AM21-0YB0 / M21

Using involute interpolation, it is possible to program a spiral contour with the shape of a so-called 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 reduction in 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.

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 (MMC's) hard disk into the CNC's working 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 working memory to the hard disk

You can also save your 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.

Languages of operating software

Option 6FC5800-0AN00-0YB0 / N00

The operating software supports the following languages as standard for user interface display texts: English, French, German, Italian, Simplified Chinese, and Spanish. The operator can switch back and forth online between foreground and background languages. Further additional languages can be ordered with option N00.

Laser switching signal, high-speed, HSLC

Option 6FC5800-0AM38-0YB0 / N38

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 the position controller, 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. The overall procedure, however, requires programming of a release (at the beginning of the program) with CC_FASTON (DIFF1, DIFF2). 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. A negative value means an offset before the setpoint (derivative action), a positive value means an offset after the setpoint. If the programmed derivative action value is too high, i.e., if the setpoint had already been exceeded when the edge was detected, the signal is switched immediately.

Leadscrew error compensation: Bidirectional compensation

Option 6FC5800-0AM54-0YB0 / M54

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

Leadscrew error compensation / measuring system error compensation

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

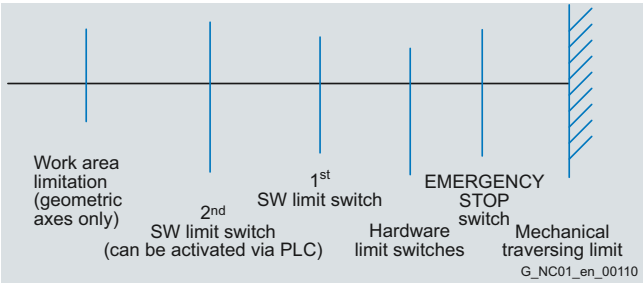
- Leadscrew error compensation (LEC) or measuring system error compensation (MSEC) as axial compensation (basic axis and compensating axis are always identical) and
- Sag error and angularity error compensation as cross-axis compensation (basic axis affects other compensation axis).

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). Added to this are the dimensional deviations caused by the measuring system as well as its installation tolerances on the machine (so-called 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 control in the form of compensation tables during commissioning. The relevant axis is compensated using linear interpolation between the intermediate points.

Limit switch monitoring



Overview of travel limits

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

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

Linear interpolation

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

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

Lock MyCycles

Option 6FC5800-0AP54-0YB0 / P54

With SINUMERIK Integrate Lock MyCycles (cycle protection), cycles can be encrypted and then saved in the controller where they are protected. Execution in the NC 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 encrypted form. It can, therefore, be used on other machines.

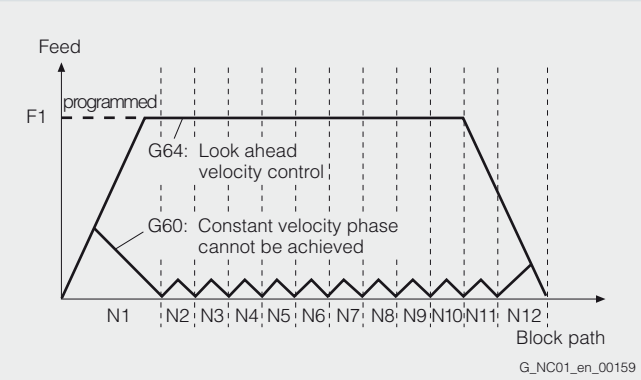
If this should also be prevented, the cycle can be permanently bound to a particular CNC hardware unit by means of an addition to the program.

Lock MyPLC

The "KNOW_HOW_PROTECT" function of SIMATIC STEP 7 provides all-round PLC machine program protection to guard technological knowledge against unauthorized access.

Look Ahead

→ Continuous-path mode with programmable rounding clearance

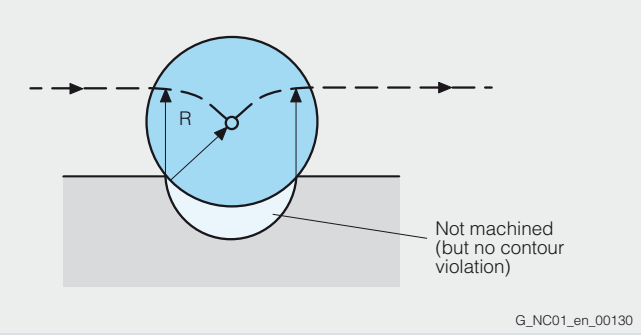


Comparison of velocity response with exact stop G60 and continuous-path mode G64 with look ahead for short displacements.

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

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

Look-ahead detection of contour violations



Behavior when tool radius > circle radius

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

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

- Bottleneck detection:
  As 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 on a transition circle, then continues on the programmed path.
- Tool radius too large for internal machining:
  In such cases, the contours are machined only as much as is possible without causing a contour violation.

Machining channels

Option 6FC5800-0AC10-0YB0 / C11 ... C19

→ Mode group

Idle times can be shortened via a channel structure using parallel sequences of motion, 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. Additional machining channels are optional. One machining channel is available in the basic version. With the options C11 ... C19, the number can be increased to up to 10 machining channels.

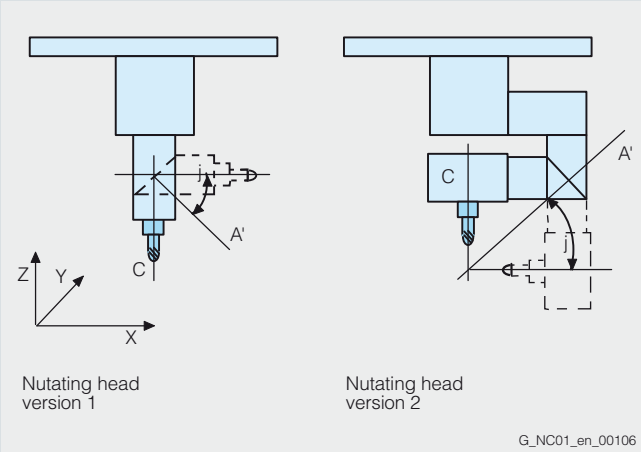
Machining package 5 axes

Option 6FC5800-0AM30-0YB0 / M30

Also contains the option

→ Multi-axis interpolation (6FC5800-0AM15-0YB0 / M15)

With the "machining package 5 axes", 5-axis machining tasks, such as the milling of free-form surfaces, can be solved easily and in a user-friendly manner.



Universal milling head

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

- 5-axis transformation with tool orientation
  In 5-axis machining, geometry axes X, Y and 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 RTCP (remote tool center point) function, the tool swivel axes can be positioned in manual mode, as long as there is compliance with the tool center point marked by the tool tip. The RTCP function simplifies the inclusion of program interpolation points in manual mode with orientation of the tool.
- Universal milling head/nutating head
  Prerequisite: Machining package 5 axes with 5-axis transformation. Using a universal milling head in conjunction with the "Nutating head" function, it is possible to machine outside contours of spatially formed parts at a high feedrate. To do this, the control executes a 5-axis transformation. Three translatory main axes (X, Y, Z) determine the tool operating point; two 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 universal 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.

Note:

The basic version contains 3 axes/spindles. The "machining package 5 axes" does not include any additional axes/spindles which might be required.

→ Axes/spindles Option 6FC5800-0AA00-0YB0 / A01 ... A28

Restricted functionality of export control versions:

Not possible.

Machining package 5 axes, additional function 7th axis

Option 6FC5800-0AS01-0YB0 / S01

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 aircraft assembly.

SINUMERIK 840D sl Type 1B

Magnetic cogging torque compensation, COCO

Option 6FC5800-0AN46-0YB0 / N46

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

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 three levels for interrupt routines) are possible in one main program. A main program can also be called from within another main program or subprogram.

Manage MyPrograms

Option 6FC5800-0AP41-0YB0 / P41

SINUMERIK Integrate Manage MyPrograms provides a central system of management and archiving for NC programs and automatic backup of SINUMERIK NC data. For example, NC programs can simply be transferred from the Teamcenter.

The advantage of this feature is that it supports simple, centralized organization of NC data into which NC programmers can be integrated, as well as fast access to NC programs via networks. It can be linked into the SINUMERIK Operate user interface without requiring the addition of supplementary external equipment on the control.

Manage MyTools

The SINUMERIK Integrate Manage MyTools feature 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.

Manual machine

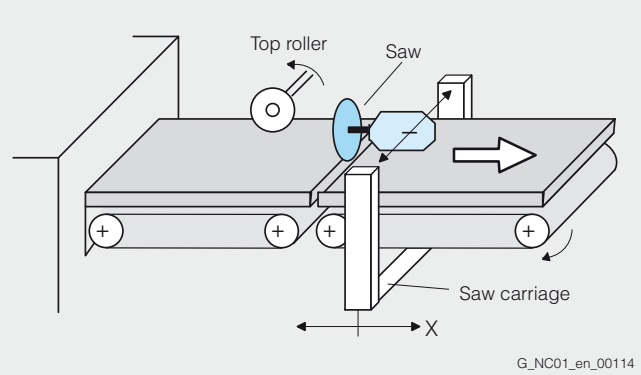
We offer the manual machine function for beginners switching over from conventional machines, but also for experienced CNC machine operators who often only use individual machining steps. The basic screen MANUAL is displayed immediately after booting the machine offering the direct machining options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

Master-value coupling and curve table interpolation

→ Generic couplings

→ Measuring stage 2

→ Synchronous spindles



Example for cyclic machines: Flying saw

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 discs that were once required for the 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 control'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 SINUMERIK, technological functions such as synchronism, electronic transfer and positioning for cyclic machines can be implemented. Mechanics (line shaft, gearing, cam discs, 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 executing 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 pressure marks (on continuous film, for example).

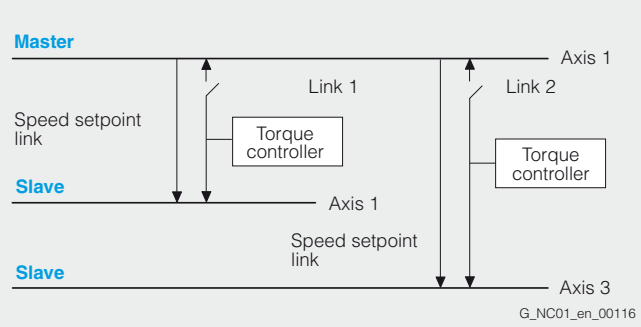
Restricted functionality of export control versions:

The number of simultaneously traversing axes is restricted to 4.

SINUMERIK 840D sl Type 1B

Master/slave for drives

Option 6FC5800-0AM03-0YB0 / M03



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

The "master/slave for drives" function is required when two 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 on the torque controller.

Application examples:

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

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

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.

MDynamics 3 axes

Option 6FC5800-0AS32-0YB0 / S32

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

→ Advanced surface

→ Spline interpolation

→ Transmit and cylinder surface transformation

→ Measuring cycles

MDynamics 5 axes

Option 6FC5800-0AS33-0YB0 / S33

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

→ Advanced Surface

→ Spline interpolation

→ Transmit and cylinder surface transformation

→ Measuring cycles → machining package 5 axes

→ 3D tool radius compensation

→ Measure kinematics

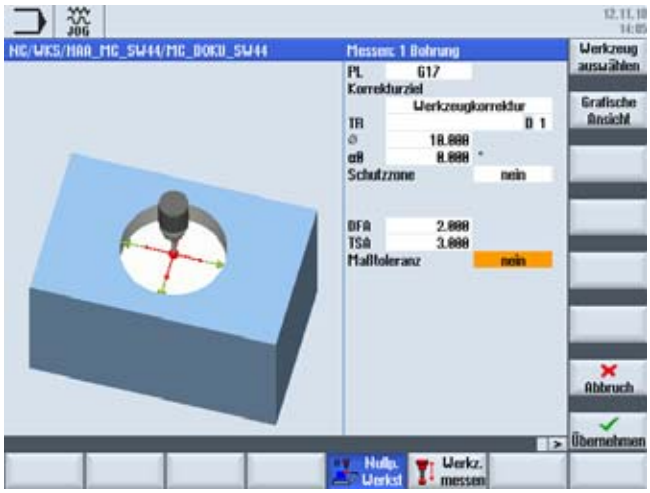
Measure kinematics

Option 6FC5800-0AP18-0YB0 / P18

The "Measure kinematics" function supports the calibration of kinematic structures of 5-axis machines.
The SINUMERIK 840D sl is now 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 dimensioned drawing of the machine is not required. The function can also be used for regular checking of the production process, when high precision is required.

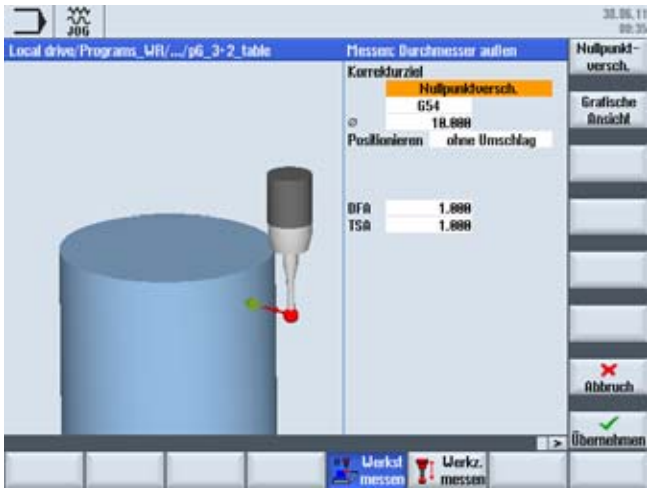
Measuring cycles

Option 6FC5800-0AP28-0YB0 / P28

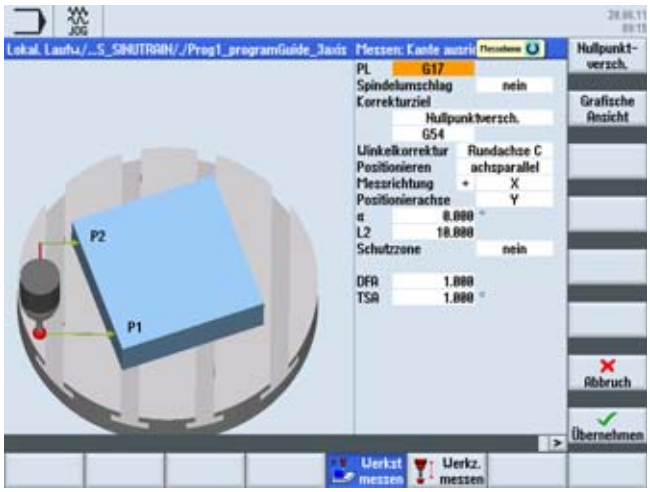


Example: Measuring a hole (programGuide)

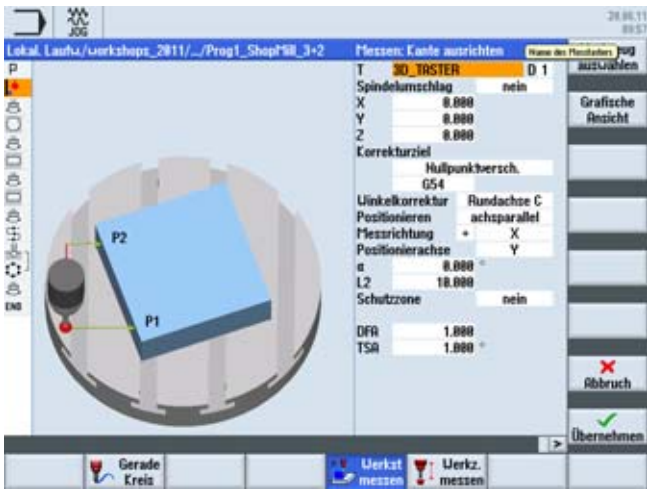
SINUMERIK 840D sl Type 1B



Example: measuring an outer diameter (programGuide)



Example: align edge (programGuide)



Example: align edge (ShopMill)

In the case of tool measuring, the loaded tool (typically in the turret 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 selection of measuring cycles means that you can handle practically every measuring task associated with turning or milling.

The "flying measurement" principle is used in SINUMERIK controls. 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 measured results can be logged in a file.
The Siemens measuring cycles offer a standard log which you can freely configure.

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 bounce-free, 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
- Edge distance – slot/web
- Circular/rectangular spigot
- Edge distance – set edge
- Hole/inner circle segment
- Spigot/outer circle segment
- Measure 3D – align plane
- Edge distance – align edge
- Measure tools
- Measure tools – Calibrate tool probe
- Measure 3D – 1 sphere/3 spheres

Turning measurement:

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

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

Measuring inputs, expanded (16) for axial measurements with TM17, PROX

Option 6FC5800-0AN57-0YB0 / N57
(on request)

Measuring functions/measuring cycles

Measuring functions can execute either channel-specifically including all axes programmed in the measuring set as well as axially from the parts program or from synchronized actions beyond NC 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.

Measuring stage 1

You can connect up to two switching touch probes to the control at the same time. 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.

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

The results of measurements can be read in the part program or with synchronized actions in both the machine and the workpiece coordination system. You can test the deflection of the touch probe by scanning a variable and outputting it to the PLC interface and deriving responses in the part program. Measuring stage 2 (option M32) provides expanded functionality (for example for axial measuring, evaluation of up to 4 trigger events, and cyclic measuring).

Measuring stage 2

Option 6FC5800-0AM32-0YB0 / M32

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 three parameters: touch probe, axis and measuring edge.

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

Measurement results can be logged in the form of a file. The measuring cycles (option P28) contain a standard log which can be freely configured by the user.

Measuring system error compensation

→ Leadscrew error compensation / measuring system error compensation

Measuring systems 1 and 2, selectable

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

Metal spinning protection area MSPZ

Option 6FC5800-0AN42-0YB0 / N42
(on request)

Mode group

Option 6FC5800-0AC01-0YB0 / C01 ... C09

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

A mode group contains channels that must always be in the same mode at the same time during the machining sequence. Within a mode group, every axis can be programmed in every channel. A mode group can be regarded as an independent, multi-channel CNC.

One mode group is available in the basic version. With the option C01 ... C09, the number can be increased to up to 10 mode groups.

Monitoring for maximum tool speed/acceleration

Option 6FC5800-0AS08-0YB0 / S08

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 would be exceeded by > 100 % by the override switch.

Monitoring functions

The controls contain watchdog monitors, which are always active. These monitors detect faults in the CNC, PLC or machine in time to prevent damage to workpiece, tool or machine. When a fault occurs, the 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 6FC5800-0AM15-0YB0 / M15

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 control versions: Not possible.

Multi-axis package

Option 6FC5800-0AM10-0YB0 / M10

The multi-axis package (AXES/SPINDLES AND CHANNELS) is a low-cost package for multi-axis machines. It includes all axes/spindles and channels.

SINUMERIK 840D sl Type 1B

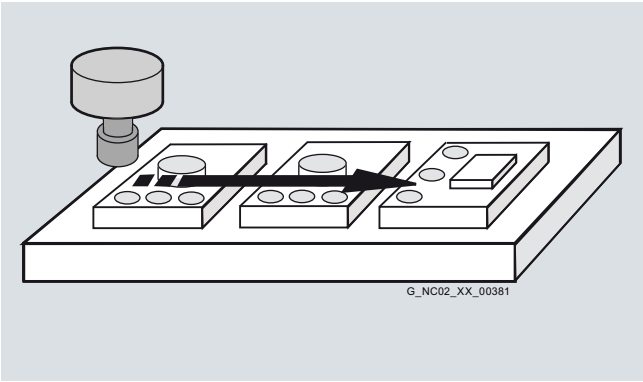
Multi-channel display

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, but it is not possible to change the NC 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 various workpieces

Option 6FC5800-0AP14-0YB0 / P14

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

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.

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 (range of values: 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 controls 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 controls using the online ISO dialect interpreter.

Operating modes

In the "Machine" operating area, you have a choice of three modes:

- JOG
  JOG mode (jogging) is intended for the manual movement of axes and spindles, as well as for setting up the machine. The set-up functions are reference point approach, repositioning, traveling with the handwheel or in the predefined incremental mode, and redefinition of machine zero point (preset/set actual value).
- MDI
  In MDI (Manual Data Input) mode, you can enter individual program blocks or sequences of blocks, then execute them immediately via NC Start. The tested blocks can then be saved in part programs. The Teach In submode allows you to transfer sequences of motion to the MDI program by returning and storing positions.
- AUTO
  In AUTOMATIC mode, the part programs are executed fully automatically once they have been selected from 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 operating modes, you can modify the sequence of a program using the following program control functions:

- SKP Skip block (up to eight 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

Operation without OP

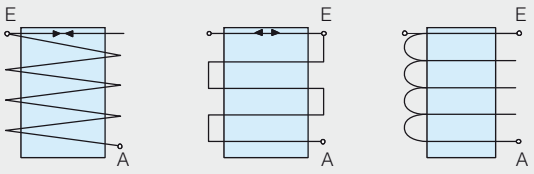
Option 6FC5800-0AP00-0YB0 / P00

Operation of SINUMERIK via the VNC viewer requires activation in the "TCU.ini" and a confirmation on the SINUMERIK operator panel or SIMATIC Thin Client Panel that operation is now permitted via the VNC viewer. When a SINUMERIK operator panel or SIMATIC Thin Client Panel is not installed, this option can be used to activate the setting in the "TCU.ini" and to suppress scanning of the confirmation.

SINUMERIK 840D sl Type 1B

Oscillation functions

Option 6FC5800-0AM34-0YB0 / M34



Oscillating motion with continuous infeed

Oscillation with infeed in the reversal points

Oscillation with infeed in the left reversal point and distance specification

A Starting point of the block
E End point of the block

G_NC01_en_00117

Oscillation functions

With this function, an axis oscillates at the programmed feedrate between two 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:
  - Reciprocating movement and infeed can be terminated by deleting the 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

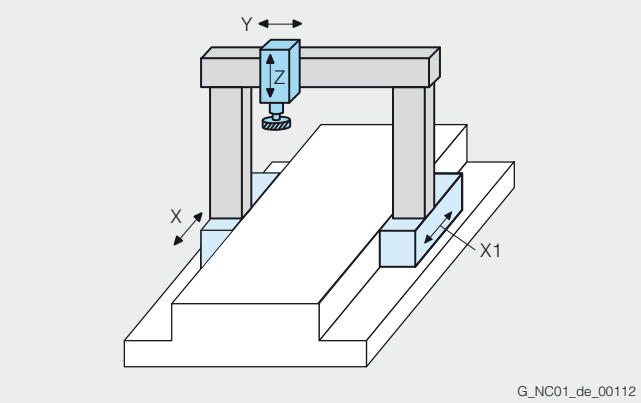
The spindles can also perform reciprocating movement.

Package: Coupling, transformation and sensor technology PCTS

Option 6FC5800-0AN21-0YB0 / N21
(on request)

Pair of synchronous axes (gantry axes)

Option 6FC5800-0AM02-0YB0 / M02



Gantry axes (pair of synchronous axes X/X1)

The "gantry axes" function can be used to traverse the axes of up to three 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.

Path length evaluation

Option 6FC5800-0AM53-0YB0 / M53

→ Synchronized actions

With path length evaluation, data in the control can be buffered, so that conclusions can be drawn in respect of 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

These data are stored in the SRAM and are not affected by power on/off. Using an external utility, consistent data can, therefore, be achieved for the complete life cycle of a machine. These data can also be read through system variables in the part program and in synchronized actions.

Path velocity-dependent analog output

Option 6FC5800-0AM37-0YB0 / M37

Using the path-velocity-dependent analog output, the current 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, PRIG

Option 6FC5800-0AN76-0YB0 / N76

This loadable compile cycle is used in the machining or analysis of workpieces or components in cases where the process requires exact information about the traversed path. It is possible to program the required displacement per pulse. The pulses are output via connector X520 of the SINAMICS TM41 technology module.

SINUMERIK 840D sl Type 1B

Plain text display of user variables

→ CNC high-level language

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 or definition: DEF REAL DEPTH/Display: DEPTH.

Plastics package IME

Option 6FC5800-0AS40-0YB0 / S40

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

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

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 capacity of the PLC. The STL (statement list), FBD (function block diagram), and LAD (ladder diagram) 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 (OBs)
- Function blocks (FBs) and function calls (FCs)
- Data blocks (DBs)

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

The STEP 7 software package (for SIMATIC S7-300) is a standard component of 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 and SCL (Structured Control Language).

PLC/NCK 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 of NCK system variables by the PLC
- Reading and writing of 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, and transfers some of the data to OB40 and the rest to the user interface at the beginning of the cycle. If the data do not require an acknowledgment from the user, CNC machining is not affected.
- 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. traversal of 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 (function blocks) and FCs (function calls) 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 status

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

This allows you to do the following on site 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 divided up into 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 6FC5800-0AD10-0YB0 / D11 ... D18

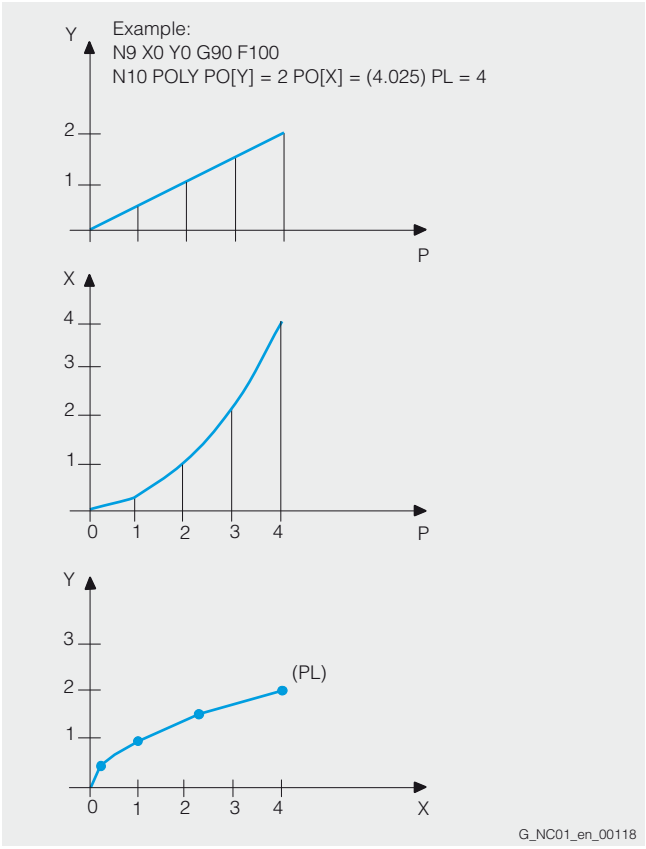
The PLC user memory can be expanded by 128 KB in each case with this option.

Polar coordinates

Programming in polar coordinates, it is possible to define positions with reference to a defined center point by specifying the radius and angle. The center point can be defined by an absolute dimension or incremental dimension.

Polynomial interpolation

Option 6FC5800-0AM18-0YB0 / M18



Polynomial interpolation

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

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

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

With polynomial interpolation, it is possible to generate many different 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 used optimally 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.

Position monitoring

SINUMERIK controls 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 limits 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. In follow-up mode, position monitoring is not active.

Position switching signals/cam controller

Option 6FC5800-0AM07-0YB0 / M07

→ High-speed CNC inputs/outputs

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.

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

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.

Positioning axes/auxiliary spindles

Option 6FC5800-0AB00-0YB0 / B01 ... B28

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 using up an additional machining channel.

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

Overview of possible functions

- POS / SPOS / M3, M4, M5 (from NC block)
- POSA / SPOSA (from NC 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
- Couplings (TRAIL, gantry and master/slave (positioning axis as slave axis), LEAD, EG, CT, ...)

Unavailable functions:

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

Positioning axes/spindles via synchronized actions

You can position axes/spindles depending on conditions (the actual values of other axes, high-speed inputs, etc.) 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 direct 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.

Preset

With the Preset function, you can redefine the machine zero point in the machine coordinate system. 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

Option 6FC5800-0AM62-0YB0 / M62

Spotting errors before they happen. This is the motto for our SINUMERIK controls, which allow you to implement tool and process monitoring. Active power monitoring keeps an eye on such things 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. Up to 2 PROFIBUS slaves can be connected.

PROFINET

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 sensors and actuators using one or several central controllers in production engineering.

PROFINET is supported by PROFIBUS International and the INTERBUS Club 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, highly re-usable systems
- Simple integration of existing PROFIBUS or Interbus systems

PROFINET CBA

PROFINET CBA is an automation system for plants with distributed intelligence. The key characteristics of this model inspired by standard IEC 61499 are therefore:

- 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

A PROFINET CBA system therefore 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 graphical, vendor-independent component connection editor (iMAP) is available for linking individual components to the 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 ACCO (Active Control Connection Object) functions as a consumer and provider and is the heart of the communication system. The RT-Auto provides 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. (An IO controller is a master as compared to PROFIBUS.)
- An IO device is a distributed field device which is linked up 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.
  (An IO device is a slave as compared to PROFIBUS.)
- 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. (As 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 which 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 relations, diagnostic options and potential useful data exchange are determined by the communication view. Communication relations (CR) with varying properties are specified for the transfer of parameters, cyclical 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 relation 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 on 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.

ProgramGUIDE

→ Run MyScreens

The "ProgramGUIDE" with the "Animated Elements" and the "Cursor Text" provides perfect support for integrating the cycles into part programs. Users can define their own softkeys, input fields and displays using "SINUMERIK Integrate Run MyScreens".

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 at "power on" with set machine data. 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 /3GL

Programmable acceleration

With the "programmable acceleration" function, it is possible to modify the axis acceleration in the program in order to limit mechanical vibrations 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 AUTOMATIC mode and in all interpolation modes. As part of intelligent motion control, this function provides a more precise workpiece surface.

Programming language

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

Programming package

→ Create MyHMI /3GL

programSYNC

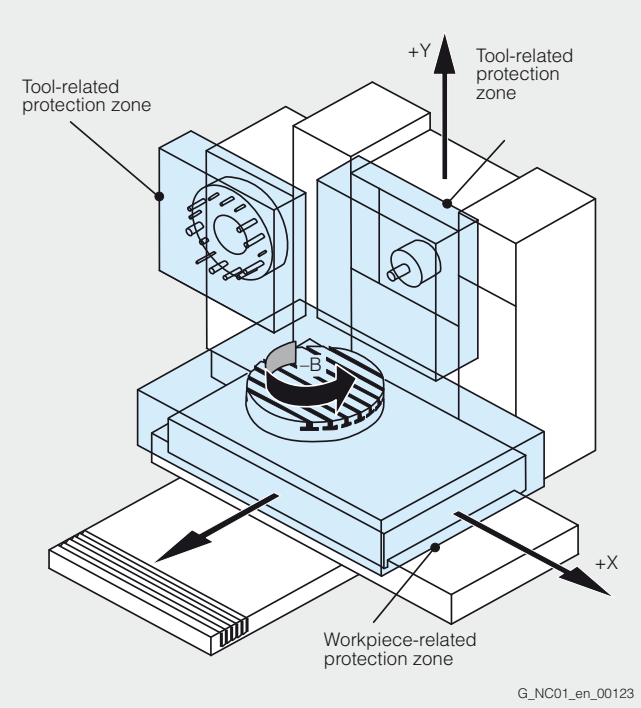
Option 6FC5800-0AP05-0YB0 / P05

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

Protection aeas 2D/3D



Protection areas

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

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

- Fixed machine components and built-on accessories (tool magazines, swiveling touch 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 ares 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 AUTOMATIC 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 ten contour elements, which are available for describing a protection area.

Punching/nibbling

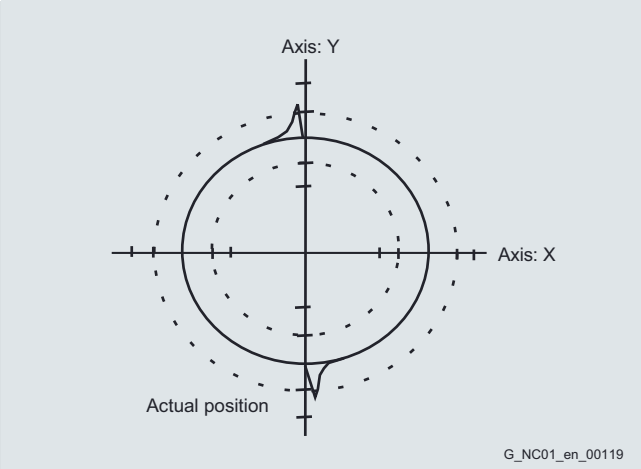
Option 6FC5800-0AM33-0YB0 / M33

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

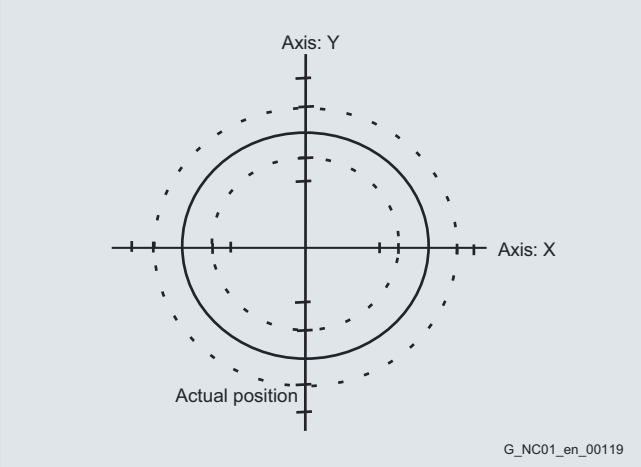
- Language commands
  The punching/nibbling functions are activated and deactivated using simple, clear high-level language elements such as PON, SON, PONS, PDELAYON, and so on.
- Stroke control
  CNC and punch are synchronized to each other by the high-speed signals that are input and output via the drive bus in the control's position-control cycle, making it possible to attain high velocities and maximum precision.
- Automatic path segmentation
  You can choose whether you want the control to break the machining path down automatically by stroke length (SPP) or stroke rate (SPN). With SPP, the 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).

SINUMERIK 840D sl Type 1B

Quadrant error compensation



Quadrant transitions without compensation



Quadrant transitions with quadrant error compensation

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

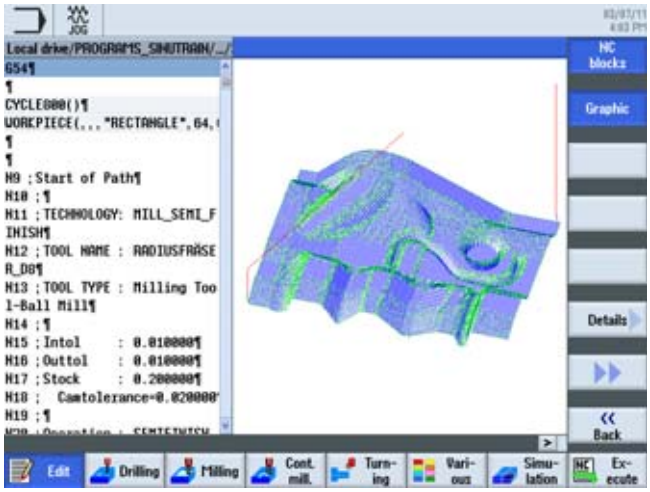
In operator-controlled quadrant error compensation, you set the intensity of the correction pulse as per an acceleration-based characteristic. This characteristic is determined and parameterized on startup with the aid of the circularity test. During the circularity test, deviations of the actual position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

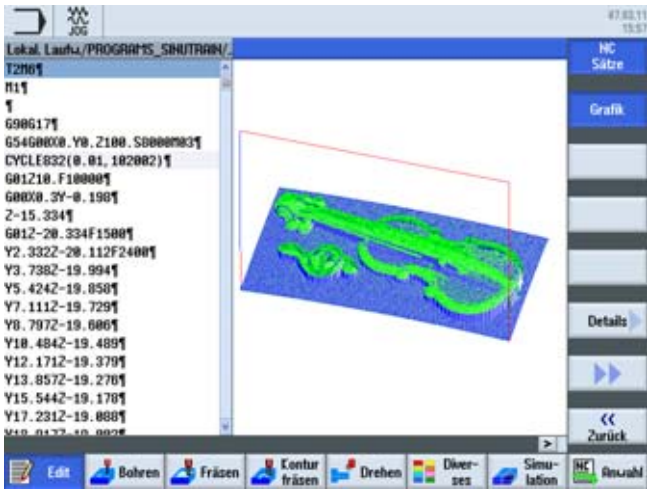
Quickview

→ "Quickview" for mold-making programs

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 NC blocks can be selected by in the part program editor.





Reference point approach

When using a machine axis in program-controlled mode, it is important to ensure that the actual values supplied by the measuring system agree with the machine coordinate values. Reference point approach (limit switch) is performed separately for each axis 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-value encoders is carried out automatically when the control is switched on (without movement of axis), if the corresponding axis is recognized as being calibrated.

Repos

Following a program interruption in AUTOMATIC mode (e.g., to take a measurement on the workpiece and correct the tool wear values or because of tool breakage), the tool can be retracted from the contour manually after changing to JOG mode.

In this case, the control stores the interruption point coordinates and displays the differential travel of the axes in JOG mode in the actual-value window as a Repos (repositioning) offset.

The contour can be reapproached:

- In JOG 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. You can program approach movements as straight lines, in quadrants or in semicircles.

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

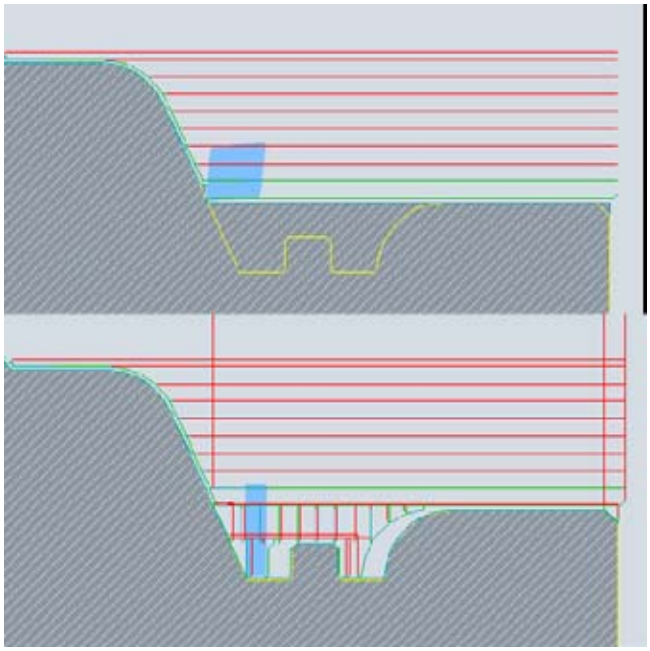
→ Working area limitation; protection areas

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

Residual material detection

Option 6FC5800-0AP13-0YB0 / P13

Contour ranges which 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 via software limit switches (e.g., working area between 0° and 60°) or 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-value encoders.

Run MyCC

Option 6FC5800-0AM04-0YB0 / M04

→ Create MyCC

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

Run MyCC /2RPT

→ Transformation: Rotating workpiece and tool

Run MyCC /2TRA

→ Transformation: DOUBLETRANSMIT

Run MyCC /ADAS

→ Axis data output via PROFIBUS

Run MyCC /AMOV

→ Variable-based axis motion

Run MyCC /AXCO

→ Compensation of a forced mechanical coupling

Run MyCC /CLC

→ Clearance control 1D/3D in position control cycle

Run MyCC /COCO

→ Magnetic cogging torque compensation

Run MyCC /COTE

→ Technological functions with compressor

Run MyCC /CRIP

→ Crank interpolation

Run MyCC /DGEN

→ Transformation: Double GENeric

Run MyCC /DSTT

→ Transformation: Dynamic swivel tripod

Run MyCC /ECCA

→ Transformation: Eccentric axis

SINUMERIK 840D sl Type 1B

Run MyCC /ECCE

→ Transformation: Eccentric

Run MyCC /HSLC

→ Laser switching signal, high-speed

Run MyCC /IMD-L

→ Integrated tool monitoring and diagnostics

Run MyCC /KPXT

→ Drive current measurement

Run MyCC /MSPZ

→ Metal spinning protection area

Run MyCC /PACO

→ Transformation: PARACOP 3 AXES

Run MyCC /PCTS

→ Package: Coupling, transformation and sensor technology

Run MyCC /PIVA

→ Transformation: Swivel axis

Run MyCC /PRIG

→ Path-related pulse output

Run MyCC /PROT

→ Axis collision protection

Run MyCC /PROX

→ Measuring inputs, expanded (16) for axial measurements with TM17

Run MyCC /RCTRA

→ Transformation: Handling

Run MyCC /RDCC

→ Transformation: Redundant axes at workpiece

Run MyCC /RESU

→ Continue machining at the contour (retrace support)

Run MyCC /ROBX

→ Transformation: ROBotic eXtended

Run MyCC /ROTE

→ Transformation: Rotating eccentric

Run MyCC /SANS

→ Scalable analog setpoint

Run MyCC /SCIS

→ Transformation: Pantograph kinematics

Run MyCC /SCRA

→ Transformation: SCARA, 2/3 axes

Run MyCC /SEC-KT

→ Spatial compensation for kinematic transformations

Run MyCC /SKID

→ Transformation: Double slide

Run MyCC /SW2A

→ Transformation: Swivel by 2 linear axes

Run MyCC /THYK

→ Transformation: Tripod hybrid kinematics

Run MyCC /TPM-PB

→ PROFIBUS tool and process monitoring

Run MyCC /VCS-A3

→ Spatial compensation VCS-A3

Run MyCC /VCS-A5 PLUS

→ Spatial compensation VCS-A5 PLUS

Run MyCC /VCS-A5

→ Spatial compensation VCS-A5

Run MyCC /VCS-ROT

→ Spatial compensation for 2 rotary axes

Run MyCC /VIBX

→ Vibration extinction

Run MyCC /XOUT

→ Extrapolated switching signals

SINUMERIK 840D sl Type 1B

Run MyCCI /COOC

→ COA interface for compiled OEM cycles

Run MyCCI /IMD

→ Integrated tool monitoring and diagnostics

Run MyCCI /UCI

→ Universal compensation interface

Run MyCCI /VCI

→ Spatial compensation interface

Run MyCCI /xy

→ Create MyCCI

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 for SINUMERIK 840D sl. For this special application, the customer uses software (GNU compiler and linker) in an environment known as "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.

→ Run MyCCI /IMD

→ Run MyCCI /UCI

→ Run MyCCI /COOC

Run MyHMI

→ Create MyHMI

SINUMERIK Integrate Run MyHMI is used to execute programmed HMI applications.
The applications are programmed with either QT/ C++, Visual Basic .net or C#. The programming package offers complete flexibility and a very wide range of functions for generating customized operating screens. The user is given very broad scope for expanding the SINUMERIK user interface, from the simple operation of integrating individual user screens in SINUMERIK Operate to the more complex process of creating customized user interfaces.

Run MyHMI /3GL (.NET)

Option 6FC5800-0AP66-0YB0 / P66

→ Create MyHMI /3GL (.NET)

The runtime license SINUMERIK Integrate Run MyHMI /3GL (.NET) allows users to execute programmed HMI applications on the PCU 50 under the Windows operating system.

This option will be required by users who want to create their own operating areas with the .NET Framework.

This option is also required for background functions with data communication (application without HMI components).

Run MyHMI /3GL

Option 6FC5800-0AP60-0YB0 / P60

→ Create MyHMI /3GL

The runtime license SINUMERIK Integrate Run MyHMI /3GL allows users to execute programmed HMI applications on the PCU 50 or the NCU.

The software option will be needed by users who want to integrate individual screens in SINUMERIK Operate or to create their own operating areas.

This option is also required for background functions with data communication (application with HMI components)

Run MyHMI /PRO

Option 6FC5800-0AP47-0YB0 / P47

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 by variable layout.

Run MyScreens

Option 6FC5800-0AP64-0YB0 / P64

The SINUMERIK Integrate Run MyScreens functionality allows SINUMERIK users to design their own user interfaces in order to visualize either machine-manufacturer or end-user functional expansions or simply their own screen form layouts.

User interfaces configured by Siemens or other machine manufacturers can be modified or replaced. 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 control itself. A graphic tool is required to create graphics and pictures. Part programs can be processed with newly created user interfaces.

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.

You can implement the following functions 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
- Execute subprograms, file functions, program instance services (PI services) or external functions (HMI Advanced)
- 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.

With the integrated editor, even the basic version of the user interface can be expanded at predefined softkeys by up to 5 screens (more than 5 screens with SINUMERIK Integrate Run MyScreens (option P64).

Run MyVNCK

With the VNCK we are offering a virtual NC kernel which is implemented at 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.

Runtime license OA Programming

→ Run MyHMI /3GL

Safety Integrated

SINUMERIK Safety Integrated are integrated safety functions that support the implementation of highly effective personnel and machine protection. The safety functions meet the requirements of DIN EN 61508 for use up to and including SIL2 and Category 3, and PL d according to DIN EN ISO 13849. This allows the main requirements for functional safety to be implemented easily and cost-effectively. 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

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

Option 6FC5800-0AC60-0YB0 / C61 ... C62

Further additional 15 SINUMERIK Safety Integrated SI axes/spindles)

Safety Integrated axis/spindle, additional axis/spindle

Option 6FC5800-0AC70-0YB0 / C71 ... C78

One additional SINUMERIK Safety Integrated SI axis/spindle

SINUMERIK 840D sl Type 1B

Safety Integrated, SI-Basic

Option 6FC5800-0AM63-0YB0 / M63

SINUMERIK Safety Integrated SI-Basic for one NCU incl. 1 axis/spindle, 4 SPL inputs and 4 SPL outputs

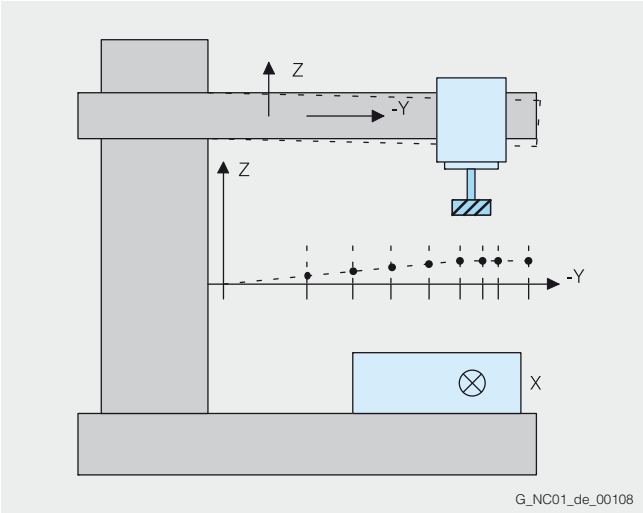
Safety Integrated, SI-Comfort

Option 6FC5800-0AM64-0YB0 / M64

SINUMERIK Safety Integrated SI-Comfort for one NCU incl. 1 axis/spindle, 64 SPL inputs and 64 SPL outputs

Sag compensation, multi-dimensional

Option 6FC5800-0AM55-0YB0 / M55



Example: Sag compensation

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 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

Restricted functionality of export control versions:

The tolerance band that can be corrected is limited to 1 mm (0.039 in) (in standard control versions: 10 mm (0.39 in).

Scalable analog setpoint SANS

Option 6FC5800-0AN48-0YB0 / N48
(on request)

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, entering the desired setpoint position (e.g. "0"), and the controller 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.

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.

Series startup

Files called series startup files can be generated to enable transfer of a particular configuration, in its entirety, to other controls that use the exact same software version, for example, controls that are to be used for the same machines. Series startup thus means bringing a series of controls to the same initial state as regards their data. You can archive/read selected CNC, PLC and PCU data for series startup. Compensation data can be optionally saved. The drive data are stored as binary data, and cannot be modified.

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

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 coordination 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 6FC5800-0AM05-0YB0 / M05

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 control, but are traversed only in succession by one motor.

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

SINUMERIK 840D sl Type 1B

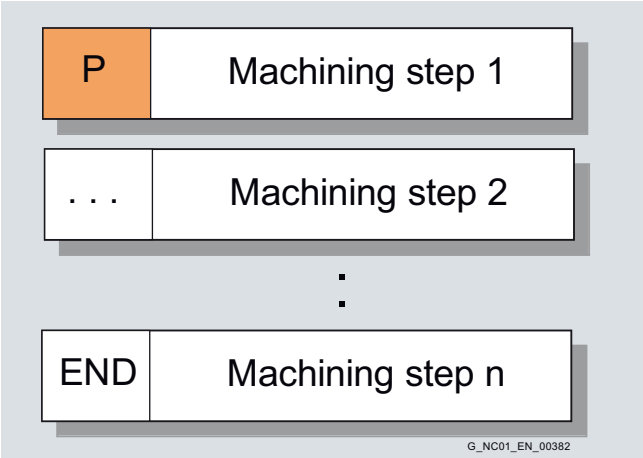
ShopMill/ShopTurn

Option 6FC5800-0AP17-0YB0 / P17

- The "ShopMill/ShopTurn" option includes the functions:

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.



Machining step programming

Simulation 1, 3D

Option 6FC5800-0AP25-0YB0 / P25

The simulation can be extended to 3D representation by means of the "3D simulation" option. This extension also applies to the simultaneous recording.

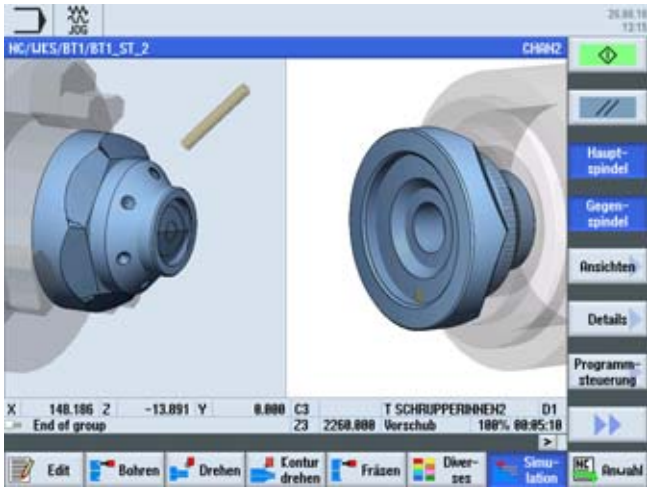
Simulation (SINUMERIK Operate)

→ Simulation 1, 3D

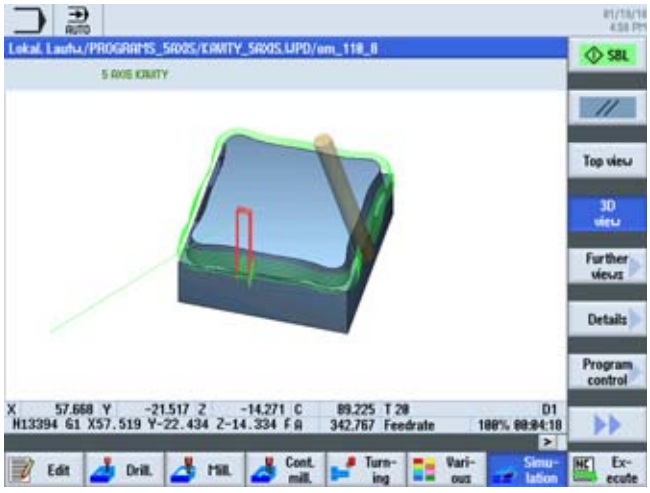
Simulation is supported by an autonomous program interpreter (SNCK) and a separate simulation-data environment in SINUMERIK Operate. The SNCK considers the entire syntax of the SINUMERIK controller family, including the possibility of incorporating special user options on the machine by comparing data 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 controller:
Simulation of the finished part is performed with the real NC data. The NC data is automatically compared 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
- 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 picture area.
- Any section can be selected
- While one workpiece is being machined, the machining of another workpiece can be simulated in parallel (as of NCU 720.x)
- The simulation can be extended to 3D representation by means of the "3D simulation 1" 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 yet available when compile cycles are used
- Blank CAD data cannot be imported (for example, for castings)
- Couplings cannot be simulated (coupled motion, axial master value coupling, electronic gear)
  - 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
- Synchronization with axis replacement (GET, GETD) cannot be simulated. This means a time estimate is also not possible!
- If parallel processing is performed in channel 1 and channel 2 (revolutional feed rate with spindle from 2nd channel), the program execution in the simulation does not match that of the automatic program (simultaneous recording)
- The representation of the workpiece is incorrect for swiveled planes (cycle 800).

Simultaneous recording with SINUMERIK Operate

Option 6FC5800-0AP22-0YB0 / P22

→ Simulation

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

Note:

- Activate simultaneous recording prior to NC start in order to avoid incomplete displays.

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 eight skip levels (/0 to /7) may be programmed. The individual skip levels are activated via a data block in the PLC interface.

Spatial compensation for 2 rotary axes, VCS ROT

Option 6FC5800-0AN31-0YB0 / N31

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 an online calculation of the offsets for the 3 linear axes referred to the current tool center point.

Spatial compensation for kinematic transformations

Option 6FC5800-0AM57-0YB0 / M57

Space error compensation (SEC) is a method for compensating static position errors at the tool center point (TCP). In this way, these position errors can be compensated in the three directions in space (x, y and z) simultaneously. Measuring instruments are required for determining the error that allow the three coordinates of a measuring point to be recorded simultaneously at multiple points distributed throughout the working area (e.g. a 3D laser tracker). The SINUMERIK control can then use the resulting 3D error table to compensate each positioning window in real time.

For Cartesian machines, refer to: Spatial compensation, VCS

Spatial compensation interface, VCI

Option 6FC5800-0AN74-0YB0 / N74
(on request)

Spatial compensation, VCS A3

Option 6FC5800-0AN15-0YB0 / N15

This option implements compensation on 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 also be active in parallel to the 3 linear axes. The compile cycle supports a maximum of 3 gantry groups per channel; the compensation value of the guide axis is automatically copied to the following axis.

Spatial compensation, VCS A5 plus

Option 6FC5800-0AN17-0YB0 / N17

VCS plus is designed for use on large machines (3-axis and 5-axis gantry milling machines). The requirements for positioning accuracy of these machines in the complete work space (= volumetric accuracy) increases constantly and is specified in individual cases as < 50 µm.

The compile cycle "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, pitching, swerving as well as deviations from the perpendicular between axes). These geometrical machine errors cause offsetting 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 VCS A5 plus with TRAORI active.

In comparison to LEC (leadscrew error compensation) and CEC (circle error compensation), VCS A5 plus supports the total compensation of the geometric errors of a Cartesian machine tool. The existing LEC and CEC settings can be superimposed on VCS A5 plus.

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 as a file in the specified readable VCS format.

Measurement of the machine error 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.

Spatial compensation, VCS A5

Option 6FC5800-0AN16-0YB0 / N16

This option implements compensation on 5-axis machines in cases where the tool can be "freely" 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 orientation error of the tool is not compensated. 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).

SINUMERIK 840D sl Type 1B

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 set range
  - Maximum spindle speed
  - Programmable lower (G25) and upper (G26) spindle speed limitation
  - Min./max. speed for gear stage
  - Max. encoder limit frequency
  - 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 (for easier engaging of 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 axis mode with a program command. A common encoder can be used for both axis and spindle modes. The zero mark of the spindle is also the reference mark of the C axis, so there is no longer any need to home the C axis (synchronize C axis on the fly).
- Thread cutting with constant pitch:[1)]
  With G33 you can produce the following thread types: 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 pitch:[1)]
  Threads can also be programmed with linearly progressive (G34) or linearly degressive (G35) pitch.
- Programmable thread run-in and run-out path:
  When thread cutting, you can use DITS/DITE (displacement thread start/end) to program the path ramp for the acceleration or deceleration process as a displacement. 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 whereby 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)] Prerequisite: 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 6FC5800-0AS16-0YB0 / S16

Using spline interpolation it is possible to obtain a very smooth curve from just a few defined interpolation points along a set contour. The intermediate 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 intermediate 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, as it is in linear or circular interpolation.

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

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

With the COMPCAD compressor, "smooth" curves can be approximated within the boundaries of compressor tolerance (parallel tool paths) and surfaces of a high optical quality can also be obtained in the case of large tolerances.

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

Standstill monitoring

→ Position monitoring

Standstill monitoring represents one of the most comprehensive mechanisms for monitoring axes. The monitor checks to see whether the following error has reached the standstill tolerance limit following the elapse of a programmable time period. Upon termination of a positioning action, standstill monitoring takes over from position monitoring, and checks to see whether the axis moves further from its position than stipulated in the machine data's standstill tolerance field. The standstill monitoring function is always active following expiration of the zero speed delay time or upon reaching the fine exact stop limit as long as no new traversing command is pending.

When the monitor responds, an alarm is generated and the relevant axis/spindle brought to standstill with rapid stop via 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 follow-up mode.

Subprogram levels and interrupt routines

Subprograms can be called not only in the main program, but also in other subprograms. Subroutines can be nested to a depth of 12 levels, including the main program level. This means that a main program may contain as many as 11 nested subroutine calls. When working with Siemens cycles, three 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 resetting of the part program start or end, or following booting of the control. Users can then make the basic function settings or carry out initializations using a part program command.

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

SINUMERIK 840D sl Type 1B

Synchronized actions

→ Cross-mode actions

Even in its basic configuration, SINUMERIK allows you to initiate up to 24 actions synchronous to the axis and spindle movements. These actions run in parallel with workpiece machining, and their inception can be determined on the basis of conditions. The starting of such motion-synchronous actions (or synchronized actions for short) is, therefore, not restricted to CNC block boundaries.

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

Synchronized actions without validity identifier are non-modal only in AUTOMATIC mode. Synchronized actions with validity identifier ID are modal in the subsequently programmed blocks in AUTOMATIC mode. Statically effective synchronized actions with the identifier IDS remain active in all modes (see "Cross-mode actions").

Synchronized actions provide you with an excellent programming tool to respond very quickly to events in the interpolation cycle. Here are some typical applications:

- Comparison operation-dependent or external signal-dependent transfer of auxiliary functions M and H to the PLC user software and derived machine responses
- Fast, axis-specific deletion of the distance-to-go caused by input signals
- External signal-controlled read-in disable for the CNC block
- Monitoring of system variables such as velocity, power and torque
- Controlling process variables (velocity, speed, distance, etc.)

Restricted functionality of export control versions:
Only 1 active synchronous function (SYNFCT) is possible at a time. The number of simultaneously traversed axes is limited to 4 (path and positioning axes).

Synchronized actions stage 2

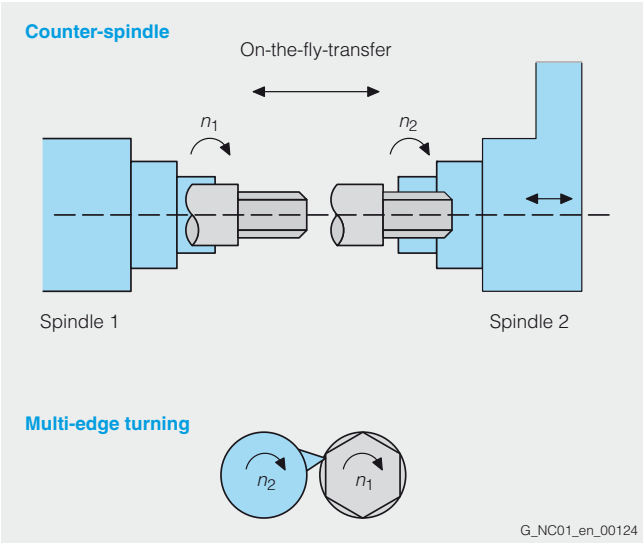
Option 6FC5800-0AM36-0YB0 / M36

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 simultaneously traversed axes is limited to 4 (path and positioning axes).

Synchronous spindles, multi-edge turning

→ Generic couplings



Examples for synchronous spindles/multi-edge turning

True-to-angle synchronization of one leading and one or more following spindles enables on-the-fly workpiece transfer, particularly for turning machines, from spindle 1 to spindle 2, for example for the purpose of finishing, without experiencing the non-productive times normally associated with rechucking.

In addition to the speed synchronism, the relative angular position of the spindles to one another, e.g., on-the-fly, position-oriented transfer of edged workpieces, is also specifiable.

On-the-fly transfer:

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

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

Multi-edge turning:

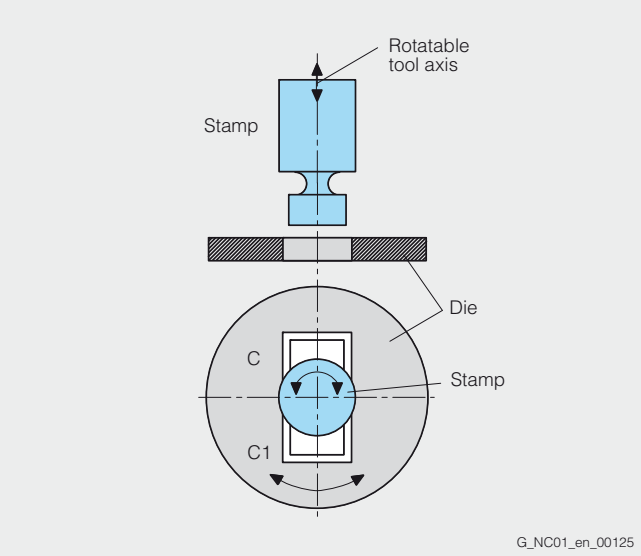
$n_2 = T \cdot n_1$

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

SINUMERIK 840D sl Type 1B

Tangential control

Option 6FC5800-0AM06-0YB0 / M06



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

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.

Applications:

- Tangential setting of a rotatable tool during punching/nibbling
- Correction of the workpiece alignment for a belt saw
- Setting of a dressing tool on a grinding wheel
- Tangential feed of a wire for 5-axis welding
- Setting of a cutting wheel for machining glass or paper

Tangential control is effective in all interpolation modes.

On punching and nibbling machines with a rotatable punching tool and associated lower tool, the following functions may be used to ensure universality of the tool:

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

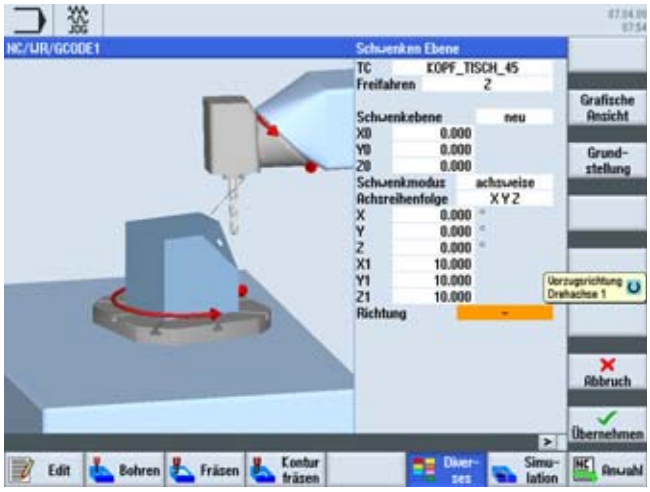
Tapping with/without compensating chuck

→ Spindle functions

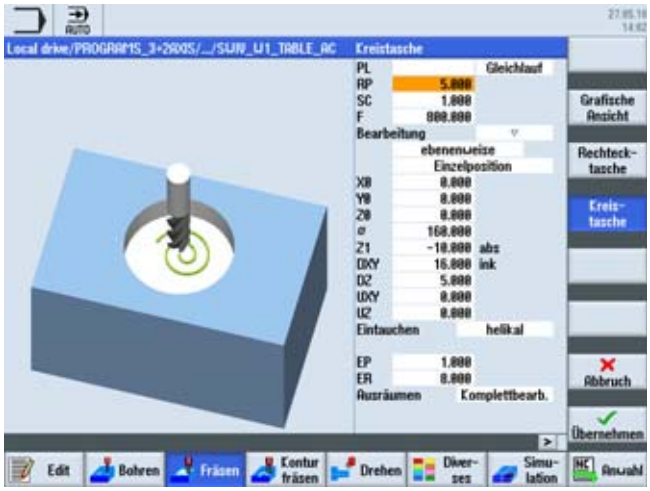
Technological functions with compressor, COTE

Option 6FC5800-0AN50-0YB0 / N50
(on request)

Technology cycles



Example: Swiveling



Example: Milling the circular pocket

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 by means of 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

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 modifications can be corrected using temperature compensation.

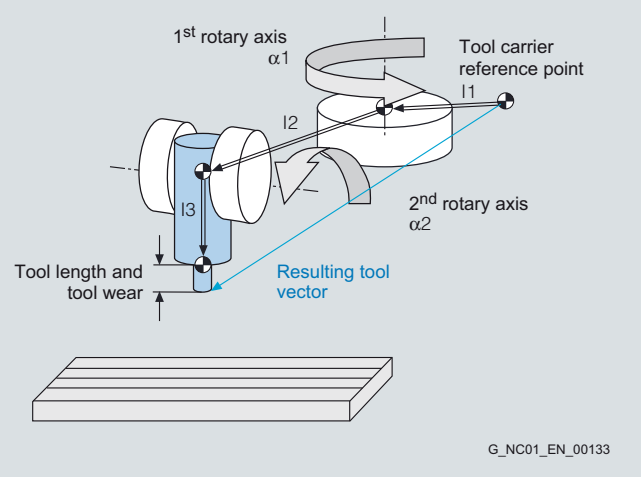
At a specific temperature, measure the actual-value offset over the positioning range of the axis to obtain the error curve for this temperature value. Error curves for different temperatures can be defined for each axis.

In order to ensure proper compensation of thermal expansion in changing temperatures, the temperature compensation value, reference position, and linear angle of lead 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 control in order to prevent machine overload and avoid triggering watchdog monitors unnecessarily.

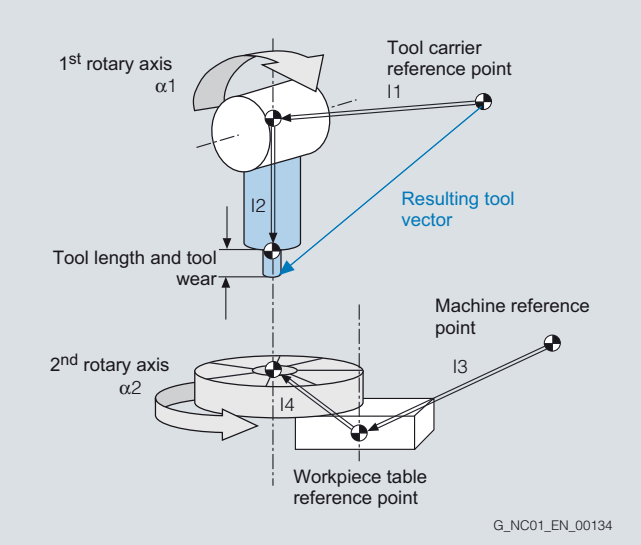
Thread cutting

→ Spindle functions

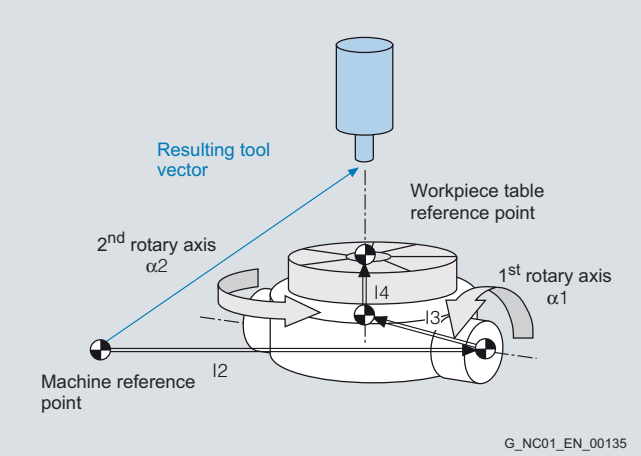
Tool carrier with orientation capability



Kinematics type T



Kinematics type M



Kinematics type P

For machine tools, which have tool carriers with settable tool orientation, the user of a SINUMERIK control can freely configure these kinematics without using 5-axis transformation. The "tool carrier with orientation capability" function enables 2½D/3D machining with fixed spatial orientation of the tool/workpiece table.

Vectors l1 to l4 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 $\alpha 1$ and $\alpha 2$ 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)

Tool change via T number

In chain, rotary-plate and box magazines, a tool change normally takes place in two stages: a T command locates the tool in the magazine, and an M command inserts it in the spindle. In 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 identification systems

→ Access MyTool ID

The Siemens Tool Management tool load and unload dialog boxes offer a link to an automatic tool identification system. This allows you to replace manual input of the tool data with automatic reading and writing of the tool code carrier.

During unloading, the data block for the tool is saved; during loading, it is read via the code carrier and entered in the tool management. In the interim, the tool data can be re-edited as during tool selection from the tool catalog (offset data, etc.).

Using an editable description file containing precisely defined tool and cutting data, the code carrier data are converted during loading into dialog data, which can be read by the tool management. During unloading, the dialog data are converted back into code carrier data with the aid, once again, of the description file.

SINUMERIK 840D sl Type 1B

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 throughout all magazines and turrets for active tools and spare tools
- Ascertaining of 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 when next D number is encountered
- Quantity, tool life and wear monitoring with prewarning limit monitoring function
- Support for the use of 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.

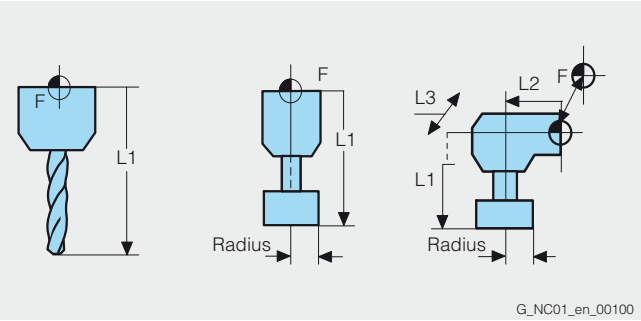
Tool management also takes tool length compensations for adapters that are permanently mounted at certain magazine locations and fitted with different tools into account.

Tool management with more than 3 magazines

Option 6FC5800-0AM88-0YB0 / M88

In the basic version of the SINUMERIK 840D sl, 3 magazines (magazine, spindle, tool buffer) are included.
With this option, the number can be increased to a maximum of 30 real magazines.

Tool offsets



Tool offsets

By programming a T function (5-figure integer number or identifier) in the block, you can select the tool. Every tool can be assigned up to 12 cutting edges (D addresses). The number of tools to be managed in the control is set at the configuration stage. A tool offset block comprises 25 parameters, e.g.:

- Tool type
- Up to three tool length offsets
- 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 need not 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 produced, the tool paths, depending on the relevant tool geometry, are controlled so that the programmed contour can be produced with every tool used.

You enter the tool data separately in the control's tool table, and in the program you call only the required tool with its offset data. During program execution, the control fetches the required offset data from the tool files and corrects the tool path for various tools automatically.

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

You can define as many as 32,000 D values per control. D numbers can be freely assigned, checked, renamed, ascertained with the associated T number, invalidated, and activated on a site-dependent basis during programming.

Tool offsets, grinding-specific

→ Grinding wheel surface speed

Grinding-specific tool offsets are available (minimum wheel radius, maximum speed, maximum surface speed, etc.) for grinding technology. When a cutting edge is created for grinding tools (tool types 400 to 499), these are stored automatically for the tool in question.

Tool types are:
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 to 499: Dresser

With the TMON command, you can activate geometry and speed monitoring for grinding tools (type 400 to 499) in the CNC part program. Monitoring remains active until deactivated in the part program with TMOF. The current wheel radius and the current wheel width are monitored. The speed setpoint monitoring 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.

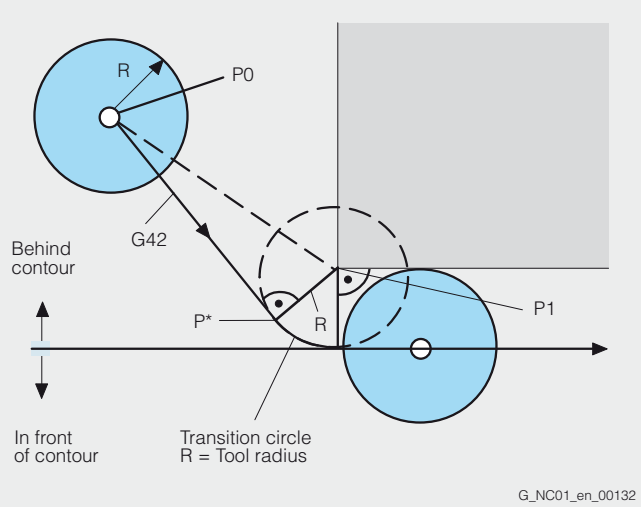
Tool orientation interpolation

→ Transformation, generic

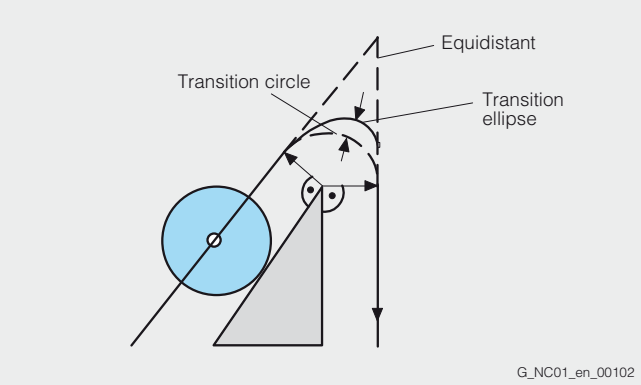
Interpolations of tool orientations supplement generic transformation: the tool orientation can be programmed in a plane as large circle interpolation (ORIVECT program command), on the outside surface of a taper in the clockwise or counterclockwise direction (ORICONCW/ORICONCCW), or even with free definition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

SINUMERIK 840D sl Type 1B

Tool radius compensation



KONT for behind the contour



Bypassing the outside corners with transition circle/transition ellipse

When tool radius compensation is enabled, the control automatically computes 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. You can match the approach and retract paths to the required contour profile or rough-part 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, in NORM the normal position at the starting point P1 is approached.

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

- With transition radii (circle or ellipse)
- Intersection of equidistant paths

For soft approach to/retraction from the contour, i.e., tangential approach and retraction irrespective of the position of the starting point, various strategies are available: Approach and retract from left or right, on a straight line, on a quadrant or semicircle, in space or in the plane.

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

The offset process of tool radius compensation may be interrupted only by a certain number of successive blocks or M commands containing no motion commands or positional data in the compensating plane.

This number of successive blocks (or M commands) can be set using machine data (standard 3, max. 5).

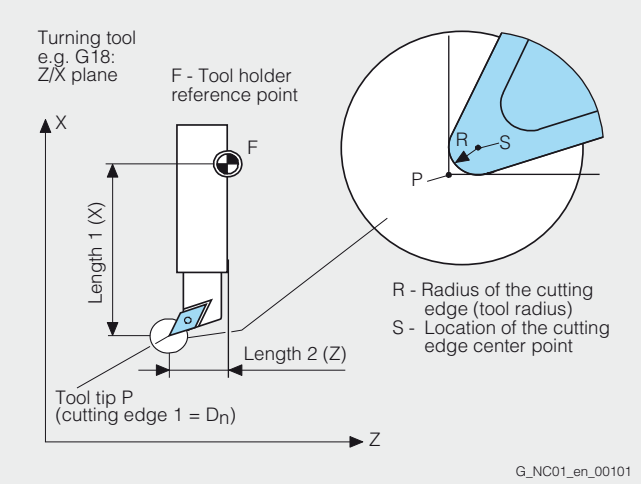
Tool radius compensation, 3D

Option 6FC5800-0AM48-0YB0 / M48

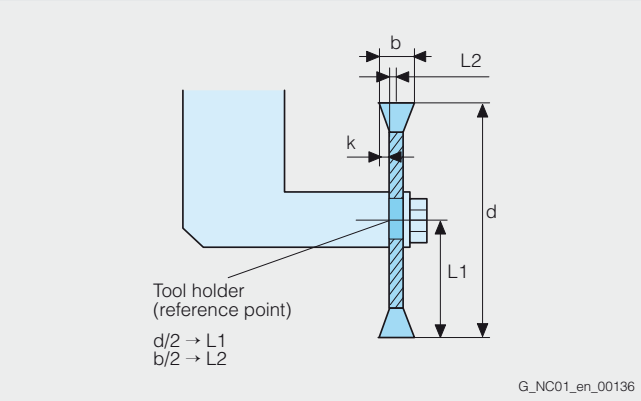
Inclined surfaces can be machined with 3D tool radius compensation or tool offset in space. This function supports circumferential milling and face milling with a defined path. The inclined tool clamping position on the machine can be entered and compensated. The control computes the resulting positions and movement automatically. 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 turned not only in the X, Y and Z planes, but also by the lead or hitch angle and the side angle.

Tool types



Geometry of turning tool



Geometry of slotting saw

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 control 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.

The following tool types can be parameterized:
Group 1xy: milling cutters (from spherical head cutter to bevel cutter)
Group 2xy: drills (from twist drill to reamer)
Group 4xy: grinding tools (from surface grinding wheel to dresser)
Group 5xy: turning tools (from roughing tool to threading tool)
Group 700: slotting saw
Group 900: special tools

The saving of all tool offsets is supported by input screens!

For wood technology, the slotting saw tool is available as a tool type.

SINUMERIK 840D sl Type 1B

Tooltips

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

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. It can then be used much more flexibly and universally.

It is then possible to also control machine kinematics by the CNC where the orientation of the rotary axes is not exactly parallel to the linear axes. Generic 5-axis transformation has been extended to 3-axis and/or 4-axis transformation, i.e., it can also be used for machines with only one rotary axis (rotatable tool or workpiece).

Transformation: Double GENeric, DGEN

Option 6FC5800-0AN34-0YB0 / N34

With this transformation it is possible to couple two 5-axis kinematics on one machine. One kinematic is the "master" and is programmed like a standard 5-axis machine, while the second kinematic is the "slave" and is moved simultaneously to the programmed "master". The part program may contain movements for the "slave", but these are subject to a number of restrictions. This option is applied, for example, on 5-axis milling machines when it is necessary to compensate the machining forces on the opposite side.

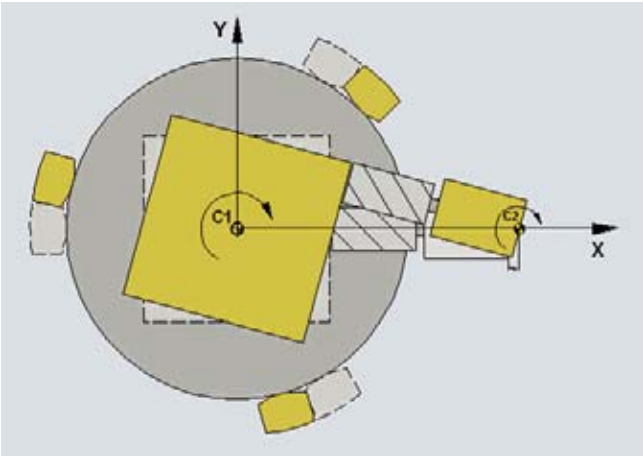
Transformation: Double slide, SKID

Option 6FC5800-0AM80-0YB0 / M80
(on request)

Transformation: DOUBLETRANSMIT, 2TRA

Option 6FC5800-0AM25-0YB0 / M25

The function supports machining on the end face or cylinder surface with a driven tool that is always held vertically with respect to the contour by means of an appropriate swivel movement and balancing movement in X.



Transformation, doubletransmit

Transformation: Dynamic swivel tripod, DSTT

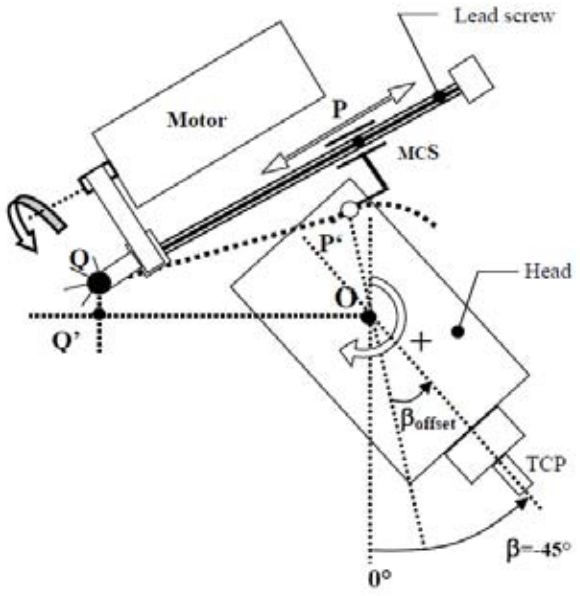
Option 6FC5800-0AM84-0YB0 / M84

This kinematic transformation 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.

Transformation: Eccentric axis, ECCA

Option 6FC5800-0AN44-0YB0 / N44

The ECCA compile cycle supplies adjustments to the position controller that are needed for the high-precision dynamic traversal of a rotary axis which is driven eccentrically by a linear motion (thread rod) in the range < 180 degrees.



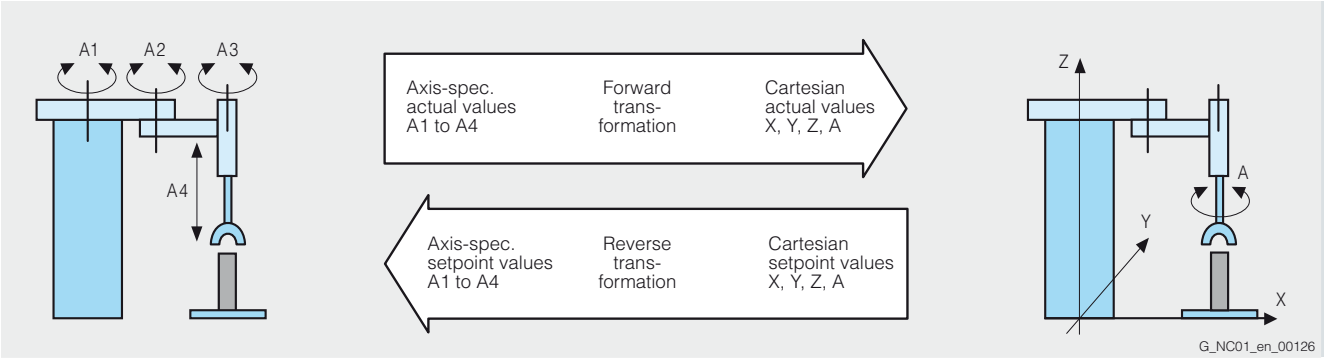
Eccentric axis kinematics, ECCA

Transformation: Eccentric, ECCE

Option 6FC5800-0AN41-0YB0 / N41
(on request)

Transformation: Handling, RCTRA

Option 6FC5800-0AM31-0YB0 / M31



Transformation: Handling

The Handling transformation package contains the so-called standard transformation block, with whose help typical 2-axis to 5-axis handling devices such as gantries or SCARAs can be operated. This coordinate transformation converts the axis-specific actual values for the axes (e.g., A1 to A4) into Cartesian values (e.g., X, Y, Z, A) and the programmed Cartesian setpoints back into axis-specific values for the handling devices.

Thanks to this coordinate transformation, the movements of the handling device become simpler and more user-friendly. The handling device can be set up, that is, manually traversed not only in the axis-specific coordinate system, but also in the handling device's own Cartesian coordinate system, using, for example, the jog keys on the handheld programming unit. Adaptation of the respective kinematics is carried out via machine data.

A 6-axis transformation for defined applications is also available (please consult your local Siemens sales office).

Restricted functionality of export control versions: Not possible.

SINUMERIK 840D sl Type 1B

Transformation: Pantograph kinematics, SCIS

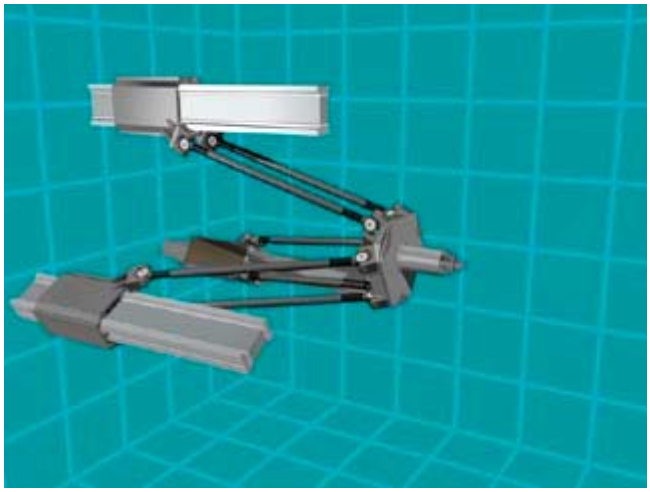
Option 6FC5800-0AM51-0YB0 / M51

"Pantograph" kinematic transformation is a type of 2-/4-axis transformation with parallel kinematics. It can work with fixed-length rods or variable-length rods.

When using kinematic transformations, workpieces can be programmed in Cartesian coordinates as usual. The SINUMERIK control calculates the required movements 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.

Transformation: PARACOP 3 AXES, PACO

Option 6FC5800-0AM44-0YB0 / M44



PARACOP animation

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 (Stuart platform) simultaneously (virtually in parallel).

PARACOP machines are TRIPOD systems on which the Stuart platform is moved by three actuators. Design measures are used to ensure that the Stuart platform cannot move in an undefined manner on these TRIPODEN types. On PARACOP machines, two 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 control calculates the required movements 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.

→ Transformation: Pantograph kinematics, SCIS

Transformation: Redundant axes at workpiece, RDCC

Option 6FC5800-0AN26-0YB0 / N26

The kinematic transformation "RDCC" (Redundant Distributed Cartesian Coordinates) is a multi-axis transformation with serial-parallel kinematics. This function is used particularly for the purpose of assembling large aircraft components which need to be positioned and orientated in the machining space. The individual positioning units are programmed with 3 axes in each case (X, Y and Z) in order to align the component. The real-time transformation "RDCC" performs the calculation of the resulting motion of all axes involved in the transformation. The path positions and path velocity are programmed by the same method employed for 3- to 5-axis programming.

The "AMOV" option is available for applications which require more than 20 axes to be involved in the transformation.

→ Variable-based axis movement, AMOV

Transformation: ROBotic eXtended, ROBX

Option 6FC5800-0AN54-0YB0 / N54

This transformation (ROBX) supports machine kinematics where between 4 and a maximum of 6 robot axes will be included in the transformation. This corresponds to up to 6 spatial degrees of freedom, i.e. 3 degrees for translation and 3 degrees for orientation. On a 6-axis machine, the tool can be oriented and turned in any desired relation to the workpiece in every point of the machining space. Moreover, a further 3 linear axes which move the robot relative to the workpiece can be included in the transformation.

The workpiece can be turned in space by a further 3 rotary axes which are included in the transformation. The tool movements including orientation are programmed in the Cartesian coordinate system.

Transformation: Rotating eccentric, ROTE

Option 6FC5800-0AN37-0YB0 / N37
(on request)

Transformation: Rotating workpiece and tool, 2RPT

Option 6FC5800-0AN43-0YB0 / N43
(on request)

Transformation: SCARA, 2/3 axes

Option 6FC5800-0AM68-0YB0 / M68
(on request)

Transformation: Swivel axis, PIVA

Option 6FC5800-0AN52-0YB0 / N52
(on request)

Transformation: Swivel by 2 linear axes, SW2A

Option 6FC5800-0AN45-0YB0 / N45
(on request)

Transformation: Tripod hybrid kinematics, THYK

Option 6FC5800-0AN36-0YB0 / N36

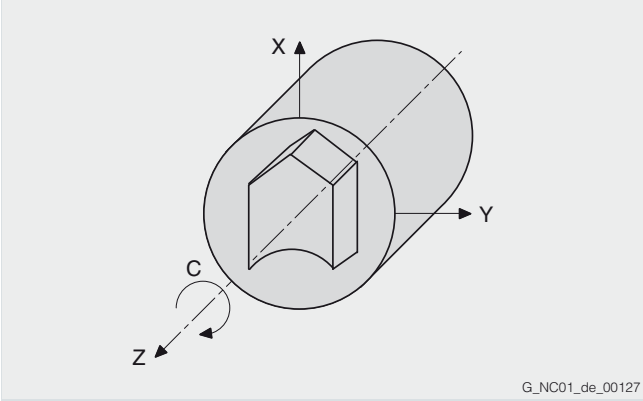
This kinematic transformation (THYK) is a 5-axis transformation for 3 parallel linear axes and two rotary axes (see figure "Exechon parallel kinematics"). The tool movements are programmed complete with their orientation in the same manner as the Cartesian 5-axis machine. The real-time transformation calculates the necessary paths and speeds for the real machine axes.

Parallel kinematic type Exechon

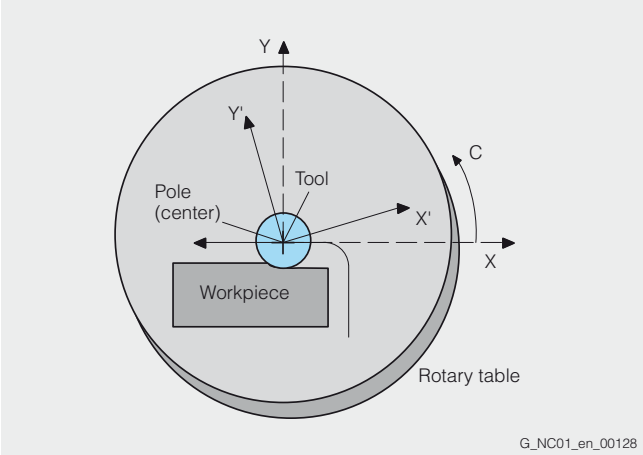
SINUMERIK 840D sl Type 1B

TRANSMIT/cylinder surface transformation

Option 6FC5800-0AM27-0YB0 / M27



Face machining with TRANSMIT



Tool-center-point path through the pole

TRANSMIT is used for milling outside contours on turned parts, e.g., square parts (linear axis with rotary axis).

As a result, programs become much more simple and complete machining increases machine efficiency. Turning and milling can be performed on one machine without rechucking.

3D interpolation with two linear axes and one rotary axis is possible. The two linear axes are mutually perpendicular and the rotary axis lies at right angles to one of the linear axes.

TRANSMIT can be called up in different channels simultaneously. The function can be selected and deselected with a preparatory function (straight line, helix, polynomial and activating tool radius compensation) in the part program or 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:

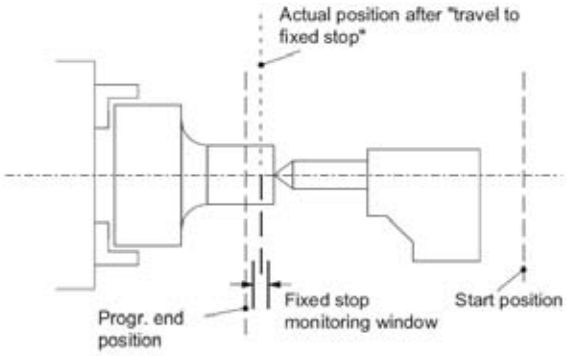
- When traveling through the pole, the rotary axis is turned automatically by 180° when the turning center is reached and the remaining block is then executed.
- When traveling close by the pole, the control automatically reduces the feedrate and the path acceleration.
- If the path contains a corner 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 slots is programmed in reference to the plane cylinder surface processed.

Travel to fixed stop with Force Control

Option 6FC5800-0AM01-0YB0 / M01



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.

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.

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, etc.	± 399999.999	± 999999.999
Rotary axes A, B, C, etc.	± 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 compensation interface, UCI

Option 6FC5800-0AN75-0YB0 / N75

SINUMERIK Integrate RunMy CCI /UCI provides an interface for customized compensation algorithms. This interface is set up universally and can be expanded by Siemens if required.

Universal interpolator NURBS

Internal motion control and path interpolation are performed using NURBS (non-uniform rational B splines). This provides a uniform method for all internal interpolations that can also be used for future complex interpolation tasks.

The following input formats are available irrespective of the internal structure:

Linear, circular, helical, involute interpolation, splines (A, B, C) and polynomials.

SINUMERIK 840D sl Type 1B

User interface

The user interface has a clear layout with 8 horizontal and 8 vertical softkeys. The targeted use of Windows-type technology permits simple and user-friendly operation of the machine.

The 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. On changing the operating area, the last active menu is always stored.

User machine data

The NCK makes machine data available for configuring the PLC user program. These user machine data are stored in the NCK-PLC interface during control 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.

Variable-based axis movement, AMOV

Option 6FC5800-0AN62-0YB0 / N62

Many couplings are already covered by the functional scope of SINUMERIK. However, there are applications where the target position is the result of a current calculation in the part program. This result is generally made available in a computation variable. Here, the target position could be newly entered in each IPO cycle via a (static) synchronized action, such as "DO POS[X]=$R1". However, the POS interpolator calculates the velocity characteristic to the specified endpoint so that the axis would come to a standstill precisely there. At this point in time the POS interpolator has no information about the next calculation and cannot know whether the velocity will remain constant or not. This behavior is undesirable in cases where calculations are to be used to traverse several axes simultaneously along the path.

The compile cycle "AMOV" described here solves this problem and assumes that every next block will contain a position and a velocity specification; the user is responsible for ensuring that the variable contents are such that they result in smooth transitional motions.

The compile cycle is also employed for special applications involving more than 20 interpolating axes in different channels.

Variables and arithmetic parameters

Using variables in place of constant values permits the development of flexible programs. 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.

Sorts of variables

The control 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 variable whose address is R plus a number. The predefined arithmetic variables are of the REAL type.
System variables	Variables provided by the control that can be processed in the program (write, read). System variables enable access to work offsets, tool offsets, actual values, measured axis values, control conditions, etc.

Types of variables

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)	$\pm(10^{-300} \ldots 10^{+300})$
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	Any 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 interpolation cycle (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).

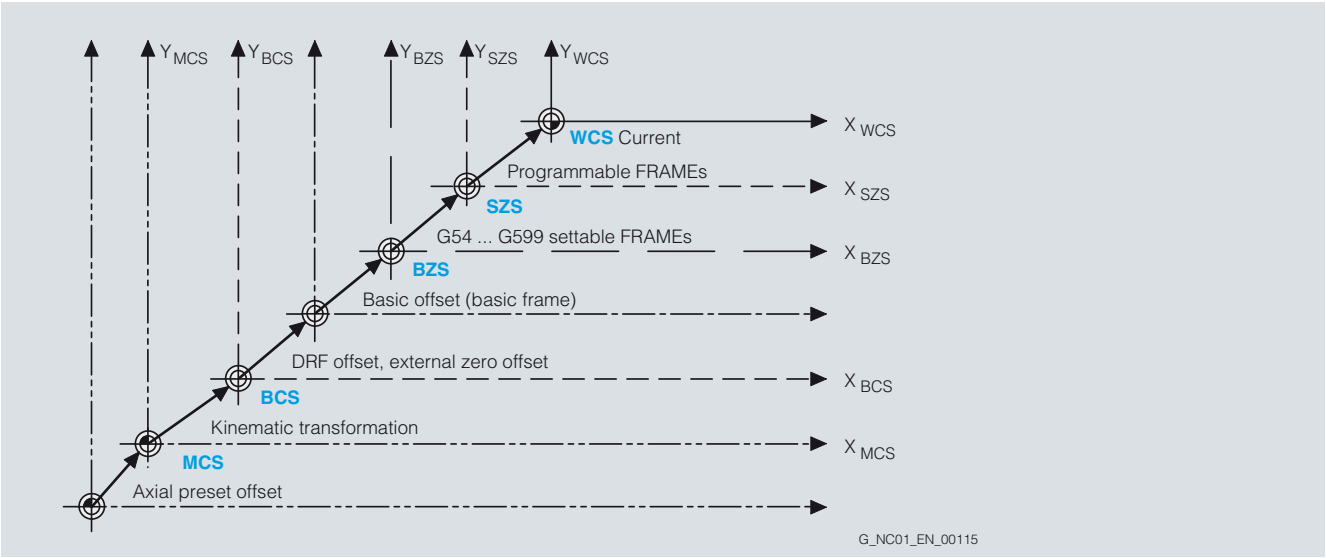
Vibration extinction VIBX

Option 6FC5800-0AN11-0YB0 / N11

The function is implemented as a loadable compile cycle and supports the 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 can significantly reduce disturbing oscillations that follow positioning actions.

Work offsets

→ Frame concept



Coordinate system

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 three Cartesian axes (geometry axes), as well as other non-geometric axes (special axes).
- The BCS and the 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. three 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.

Thus, you use work offsets to transform your machine zero point into the workpiece zero point in order to simplify programming. You can choose from among various work offsets:

- Settable work offsets:
  You can enter offset coordinates, angles and scaling factors in up to 100 possible work offsets (G54 to G57, G505 to G599), in order to call zero points from any program 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:
  You can also activate axis-related linear work offsets via the PLC user software (function blocks) with assignment of system variable $AA_ETRANS [axis].

Working area limitation

→ Work offsets

Working area limitations describe the area in which machining is permitted.

These limitations refer to the basic coordinate system. Checks are made to see whether the tool tip has penetrated the protected working area (also taking into account the tool radius). One value pair (plus/minus) per axis may be used to describe the protected working area.

The upper and lower working area limits, which can be set and activated via setting data, may be modified using the G25/G26 commands. Working area limitations restrict the traversing range of the axes in addition to the limit switches. Protection areas in which tool movements are suspended and which protect equipment such as tool turrets, measuring stations, etc., against damage, are thus set up in the machine's working area.

Working plane

→ Tool radius compensation

When specifying the working plane in which the desired contour is to be machined, the following functions are defined at the same time:

- 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 control can correct the tool length and radius.

In the basic setting, the working plane G17 (X/Y) is preset for drilling/milling, and G18 (Z/X) for turning.