

Control structure and configuration

Mode group

Option

Order code: C01 ... C0x

Article No.: 6FC5800-0BC00-0YB0

Function

A mode group combines CNC channels with axes and spindles to form a machining unit. A mode group contains channels that must always operate in the same CNC operating mode at the same time during the machining sequence. Within a mode group, every axis can be programmed in every channel. A mode group can be regarded as an independent, multi-channel CNC.

The number can be increased to include up to 10 mode groups using the option.

Note

Basic version: 1 mode group

Benefits

Implementation of machines with multiple independent machining units, e.g. one unit is loaded while the other unit is machining a workpiece in automatic mode.

Control structure and configuration

Machining channel, additional

Option

Order codes: C11 ... C1x

Article No.: 6FC5800-0BC10-0YB0

Function

A machining channel must be regarded as a separate CNC with decoding, block preparation and interpolation. With the options C11 to C19, the number can be increased to up to 10 machining channels.

The channel structure makes it possible to process the part programs of the individual channels 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.

Note

Basic version: 1 machining channel

Benefits

Idle times can be shortened via a channel structure using parallel motion sequences, such as moving a loading gantry during machining.

→ *Mode group*

CNC user memory

Function

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.

Benefits

- Battery-buffered CNC user memory

Control structure and configuration

CNC user memory, additional

Option

Order codes: D01 ... Dxx

Article No.: 6FC5800-0BD00-0YB0

Function

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

Benefits

- Predominantly for CAD/CAM-generated part programs
- Buffered CNC user memory

→ *HMI user memory, expanded on SD card of the NCU/PPU (option P12)*

Control structure and configuration

CNC user memory, expanded

Option

Order code: P77

Article No.: 6FC5800-0BP77-0YB0

Function

The function "CNC user memory expanded" increases the CNC user memory to 100 MB. With the additional HMI user memory on the memory card of the NCU it is possible to increase the size of the CNC user memory up to 6 GB.

Benefits

- Large CNC user memory
- Direct execution from CNC user memory without Excall

→ *HMI user memory, expanded on SD card of the NCU/PPU (option P12)*

Control structure and configuration

HMI user memory, expanded on SD card of the NCU/PPU

Option

Order code: P12

Article No.: 6FC5800-0BP12-0YB0

Function

The memory for part programs and expansions to the operator interface can be extended using the expanded HMI user memory on the SD card of the NCU/PPU in systems without a SINUMERIK PCU or a SIMATIC IPC for SINUMERIK.

Note

With CNC software version 6.11 or higher the memory is extended by 2 GB (on a 4 GB SD card).

Benefits

- Larger HMI user memory without SINUMERIK PCU or SIMATIC IPC

→ *CNC user memory, additional (options D01 ... Dxx)*

Axis/spindle	Option
	<p>Order codes: A01 ... Axx Article No.: 6FC5800-0BA00-0YB0</p> <p><u>Function</u></p> <p>An additional interpolating axis/spindle can extend the number of axes/spindles in the basic configuration.</p> <ul style="list-style-type: none"> • POS/SPOS/M3, M4, M5 (from CNC block) • POSA/SPOSA (from CNC block, modally) • FC18/POS/SPOS/M3, M4, M5 (PLC axes) • PLC-VDI interface (M3, M4, M5 directly) • OSCILL (asynchronous oscillation) • OSCILL (synchronous oscillation) • do POS/SPOS/M3, M4, M5 (synchronized actions) • Couplings (TRAIL, LEAD, EG, CP, ...) • Path/geometry/additional path axes/GEOAX() • Spindles for thread cutting, tapping and thread cutting with compensating chuck • Setpoint output and actual values are available • Commissioning with SINUMERIK Operate <p><u>Benefits</u></p> <ul style="list-style-type: none"> • Expansion of number of axes/number of spindles compared with the basic version <p>→ <i>Positioning axis/auxiliary spindle</i> → <i>Simulation axis/spindle</i> → <i>Virtual axis</i> → <i>Internal/external drives</i></p>

Control structure and configuration

Positioning axis/auxiliary spindle	Option Order codes: B01 ... Bxx Article No.: 6FC5800-0BB00-0YB0																								
	<p><u>Function</u></p> <p>Positioning axes can execute movements in parallel to the actual machining process. They can be programmed with an axis-specific feedrate in the part program.</p> <p>Axis movement beyond block boundaries is also possible. Positioning axes can also be controlled by the PLC. This means that axis movements can be started independently of the part program without requiring an additional machining channel.</p> <p>Auxiliary spindles are speed-controlled spindle drives without an actual-position sensor, e.g. for tool drives.</p> <table border="0"> <tr> <td>POS/SPOS/M3, M4, M5 (from CNC block)</td><td>✓</td></tr> <tr> <td>POSA/SPOSA (from CNC block, modally)</td><td>✓</td></tr> <tr> <td>FC18/POS/SPOS/M3, M4, M5 (PLC axes)</td><td>✓</td></tr> <tr> <td>PLC-VDI interface (M3, M4, M5 directly)</td><td>✓</td></tr> <tr> <td>OSCILL (asynchronous oscillation)</td><td>✓</td></tr> <tr> <td>OSCILL (synchronous oscillation)</td><td>✓</td></tr> <tr> <td>do POS/SPOS/M3, M4, M5 (synchronized actions)</td><td>✓</td></tr> <tr> <td>Couplings (TRAIL, gantry and master/slave (positioning axis as slave axis), LEAD, EG, CP, ...)</td><td>✓</td></tr> <tr> <td>Setpoint and actual value available</td><td>✓</td></tr> <tr> <td>Commissioning and diagnostics with SINUMERIK Operate</td><td>✓</td></tr> <tr> <td>Path axes/geometry axes/additional path axis/GEOAX()</td><td>–</td></tr> <tr> <td>Tapping and thread cutting with compensating chuck</td><td>–</td></tr> </table> <p><u>Note</u></p> <p>An additional positioning axis/auxiliary spindle cannot interpolate with other axes.</p>	POS/SPOS/M3, M4, M5 (from CNC block)	✓	POSA/SPOSA (from CNC block, modally)	✓	FC18/POS/SPOS/M3, M4, M5 (PLC axes)	✓	PLC-VDI interface (M3, M4, M5 directly)	✓	OSCILL (asynchronous oscillation)	✓	OSCILL (synchronous oscillation)	✓	do POS/SPOS/M3, M4, M5 (synchronized actions)	✓	Couplings (TRAIL, gantry and master/slave (positioning axis as slave axis), LEAD, EG, CP, ...)	✓	Setpoint and actual value available	✓	Commissioning and diagnostics with SINUMERIK Operate	✓	Path axes/geometry axes/additional path axis/GEOAX()	–	Tapping and thread cutting with compensating chuck	–
POS/SPOS/M3, M4, M5 (from CNC block)	✓																								
POSA/SPOSA (from CNC block, modally)	✓																								
FC18/POS/SPOS/M3, M4, M5 (PLC axes)	✓																								
PLC-VDI interface (M3, M4, M5 directly)	✓																								
OSCILL (asynchronous oscillation)	✓																								
OSCILL (synchronous oscillation)	✓																								
do POS/SPOS/M3, M4, M5 (synchronized actions)	✓																								
Couplings (TRAIL, gantry and master/slave (positioning axis as slave axis), LEAD, EG, CP, ...)	✓																								
Setpoint and actual value available	✓																								
Commissioning and diagnostics with SINUMERIK Operate	✓																								
Path axes/geometry axes/additional path axis/GEOAX()	–																								
Tapping and thread cutting with compensating chuck	–																								

Control structure and configuration

Positioning axis/auxiliary spindle

Option

Order codes: B01 ... Bxx

Article No.: 6FC5800-0BB00-0YB0

Benefits

Positioning axes can reduce non-productive machining times considerably. They can be used to advantage when controlling workpiece and tool feeders or tool magazines.

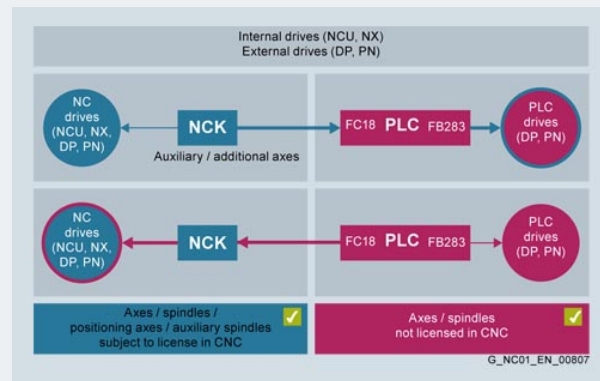
→ *Axis/spindle*

→ *Simulation axis/spindle*

→ *Virtual axis*

→ *PLC axes*

PLC axes



Function

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

Benefits

- Axes and spindles may optionally be controlled by the PLC or the NC.

→ *Axis/spindle*

→ *Simulation axis/spindle*

→ *Positioning axis/auxiliary spindle*

→ *Virtual axis*

→ *Internal/external drives*

Control structure and configuration

Multi-axis package

Option

Order code: M10

Article No.: 6FC5800-0BM10-0YB0

Function

The multi-axis package axes/spindles and channels includes all axes/spindles/positioning axes/auxiliary spindles and channels.

Benefits

- Cost-effective package for multi-axis machines

Internal/external drives

Function

Internal drives are connected to a SINUMERIK NCU or SINAMICS NX via DRIVE-CliQ. External drives are connected via PROFINET or PROFIBUS.

All internal and external drives are supported by SINUMERIK Operate:

- Parameter lists
- Diagnostics
 - Alarms with time synchronization
 - Traffic light screens
 - Service drives
- Series commissioning

Note

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

Benefits

- Freedom in the realization of machines

→ [Axis/spindle](#)

DCDCCONV – DC-DC converter for Motor Modules

Option

Article No.: 6SL3077-0AA02-0AB0

Function

In SINAMICS, the Technology Extension DCDCCONV (DC-DC CONVerter) is an application-based software extension for SERVO or VECTOR drive objects. Using Technology Extension DCDCCONV, you can use a Motor Module - which normally controls a three-phase motor - as DC converter. Additional components (e.g. energy storage reactor with smoothing capacitor) must be connected downstream of the Motor Module.

Benefits

- The closed-loop control can be parameterized as pure closed-loop DC current control - or as closed-loop cascade control, comprising closed-loop DC current control and closed-loop voltage control.
- The manipulated variables of the individual controllers can be limited so that they can be used for a wide range of applications.
- Evaluation of current and voltage measured values
- Output of the actual manipulated variables for diagnostics
- The parameterization of the closed-loop control, including the associated control restrictions, can also be modified when the controllers are enabled.

Requirements:

- SINAMICS firmware from V4.8 HF4
- STARTER from V4.5

DYNEGMA – Dynamic Energy Management

Option

Article No.: 6SL3077-0AA03-0AB0

Function

The Technology Extension DYNEGMA (Dynamic Energy Management) is a peak load management and overload protection function for machine tool infeeds.

The task of DYNEGMA is to protect the infeed unit during power peaks; these can occur when several drives simultaneously accelerate or brake. DYNEGMA only intervenes when actually required in order to guarantee not only safe and reliable operation, but also maximum dynamic performance.

To protect the infeed unit against overload, and in turn resulting tool, material machine damage, when required DYNEGMA also limits the torque of individual drives. This extends the acceleration times of these drives. As soon as there is no longer a risk of the infeed unit being overloaded, DYNEGMA again withdraws the torque limits.

Benefits

- Prevents overload situations, without restricting the dynamic performance in normal operation
- Available for all Active Line Modules (ALM) and Smart Line Modules (SLM) in combination with any SERVO drive objects
- Simple and intuitive configuration
- Prevents overload conditions, even when an EMERGENCY STOP is forced
- When the power fails, the emergency response can be performed via the V_{dc} controller

Requirements:

- SINAMICS firmware from V4.7 (corresponds to CNC software V4.7 SINUMERIK 840D sl) without using PROFIdrive telegrams 146, 147 or 149
- SINAMICS firmware from V5.2 (corresponds to CNC software V4.91 SINUMERIK 840D sl and V6.11 SINUMERIK ONE) using PROFIdrive telegrams 146, 147 or 149
- STARTER from V4.5

POLYGON – Signal generator as value table

Option

Article No.: 6SL3077-0AA00-7AB0

Function

For SINAMICS, the OA application "polygonal line" (POLYGON) is an expansion for the SERVO, VECTOR, and HLA drive objects. The polygonal line application makes it possible to create a master-value-dependent output signal based on a parameterized characteristic in the current controller sampling time (or an integer multiple of the current controller sampling time). The characteristic is defined by up to 10000 equidistantly distributed interpolation points between which linear interpolation is performed. The y-values of the interpolation points can be taken from an Excel sheet using a script.

Benefits

- Generation of special signal shapes together with the OA application SETPGEN, which generates the setpoint
- Compensation for dead times and transfer function of speed setpoint oscillation
- Encoder actual value as the master value for operating on a user-defined torque profile
- Drive-integrated cam

Requirements:

- SINAMICS firmware from V4.5
- STARTER from V4.2

SERVCOUPL – Coupling of SERVO Motor Modules

Option

Article No.: 6SL3077-0AA00-8AB0

Function

The Technology Extension SERVCOUPL (SERVO COUPLING) is a SINAMICS extension for the SERVO drive object.

In the SERVO control mode, SERVCOUPL allows motors to be operated which, as a result of their size and/or type of construction, cannot be operated from an individual SINAMICS power unit. These motors are split up into several drive objects, each of which can be operated on a Motor Module, and can be coupled with one another. As a consequence, the individual drive objects can be seen as partial motors with their own winding and terminals.

Irrespective of the number of coupled drive objects, only one encoder is required for the operation of the motor because Technology Extension SERVCOUPL implements a digital encoder splitter in the Control Unit. In this way, the actual encoder values together with the setpoint specifications are passed on to all encoderless drive objects.

Benefits

- In comparison to closed-loop control via analog encoder splitter:
simpler to use, higher degree of ruggedness, fewer hardware components
- Support of incremental and absolute encoders
- Safety Basic Functions possible on all drive objects and Safety Extended Functions on the master drive object
- Automatic fault handling between master and slave drive objects
- The angular offset between master and slave drive objects can be set, for example, to compensate for a mechanical offset.

Requirements:

- SINAMICS firmware from V4.7 (corresponds to CNC software V4.7 SINUMERIK 840D sl)
- STARTER 4.3 SP1 or higher
- Create MyConfig 4.6 or higher

Traversing range

Function

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 [mm, degrees]
Linear axes X, Y, Z,	± 399999.999	± 999999.999
Rotary axes A, B, C, ...	± 999999.999	± 999999.999
Interpolation parameters I, J, K	± 399999.999	± 999999.999

Benefits

- Traversing range can be adapted to the machine

Rotary axis, turning endlessly

Function

Depending on the application, the operating range of a rotary axis can be selected via software limit switches:

- Operating range, e.g. between 0° and 60°
- Corresponding number of rotations, e.g. limited to 1000°
- Endlessly rotating in both rotational directions

This function can also be used with absolute encoders.

Benefits

- Endlessly rotating rotary axes

Velocity

Function

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

The resulting velocity from the programmed path lengths in the CNC block and IPO cycle is always limited to the maximum velocity or, in the case of short path lengths, reduced to the velocity that can be travelled during one IPO cycle.

The minimum velocity must not go below 10^{-3} units/IPO cycle. The minimum and maximum axis velocities are dependent on the selected computational resolution. The maximum velocity of the axis is generally limited by the mechanics, the limit frequency of the encoder, or actual-value acquisition.

The speed value range (max. 300 m/s) is not limited by the CNC.

Benefits

- Speed can be adapted to the workpiece, tool, and machining situation

Feedrate override

Function

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 along the contour to be kept constant, the feedrate calculation is referred to the operating point or tool end point.

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

Benefits

- Influence programmed speed by operator or PLC

Jerk adaptation

Option

Order code: S22

Article No.: 6FC5800-0BS22-0YB0

Function

Using two switchable position setpoint filter circuits, the axial jerk value can be adapted to the specific machining type, and can be changed during machining if necessary.

Filters that provide very high damping and very high axial jerk values can be used outside of machining operations so that machining times can be significantly reduced.

The filter changeover can be initiated using a cycle (for example CUST_832).

The jerk adaptation function is a subfunction of Run MyCC /AJET.

Benefits

- Increase in the jerk value outside of machining operations in conjunction with a decrease in machining time
- Suitable for reducing the cycle time in processes with high downtimes

Acceleration with jerk limitation

Function

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-limited acceleration profile.

When you select acceleration with jerk limitation, the speed characteristic over the path is generated as a bell-shaped curve.

Benefits

- Soft acceleration pattern with simultaneously reduced wear on the mechanical parts

Programmable acceleration

Function

With the programmable acceleration function, it is possible to modify the axis acceleration in the program.

The path or positioning axis is then accelerated at the programmed value. The acceleration value set in the machine data can be exceeded by up to 100 %.

This limitation is active in the Automatic mode and in all interpolation modes.

Benefits

- Restriction of mechanical vibrations in critical program sections
- More precise workpiece surface due to intelligent motion control

Follow-up mode

Function

In follow-up mode, an axis/a spindle can be moved independently, while the actual value continues to be detected. The traversing paths are updated in the display.

Standstill, clamping and positioning monitoring functions are not effective in the follow-up mode.

Once follow-up mode is canceled, reference-point approach of the axis does not have to be repeated.

Benefits

- Recording the actual value of an independently moved axis/spindle

Measuring systems 1 and 2, selectable

Function

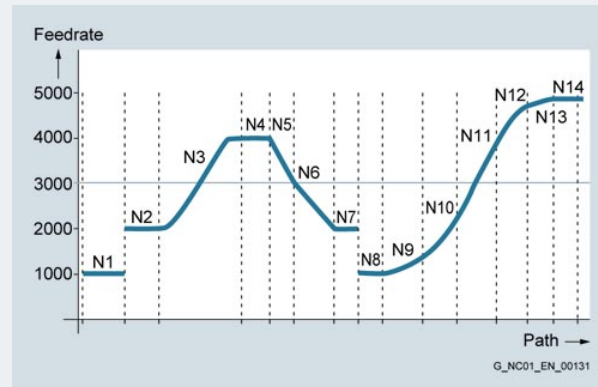
For special applications, 2 encoders can be assigned to one axis, such as for a direct measuring system for the machining process with high demands on accuracy, and an indirect measuring system for high-speed positioning tasks.

The switchover between measuring systems 1 and 2 is performed via the PLC.

Benefits

- Quicker positioning
- More precise production

Feedrate interpolation



Programming example: Feedrate interpolation

N1 Constant feedrate profile F1000: FNORM

N2 Abrupt setpoint velocity change F2000: FNORM

N3 Feedrate profile via polynomial: $F = FPO(4000, 6000, -4000)$

N4 Polynomial feedrate 4000 as modal value

N5 Linear feedrate profiles F3000: FLIN

N6 Linear feedrate 2000 as modal value

N7 Linear feedrate, as modal value

N8 Constant feedrate profile with abrupt acceleration change F1000: FNORM

N9 All subsequent F values are linked by splines F1400: FCUB

N13 Switch off spline profile

N14 FNORM

Function

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.

You can program the following feedrate profiles:

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

Benefits

- Programming of continually smooth speed characteristics depending on the curvature of the workpiece to be processed
- Manufacturing of uniform workpiece surfaces by way of jerk-free acceleration changes

→ *Polynomial interpolation*

Separate feedrate for roundings and chamfers

Function

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.

Benefits

- More precise contour traversal

Travel to fixed stop

Function

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.

Benefits

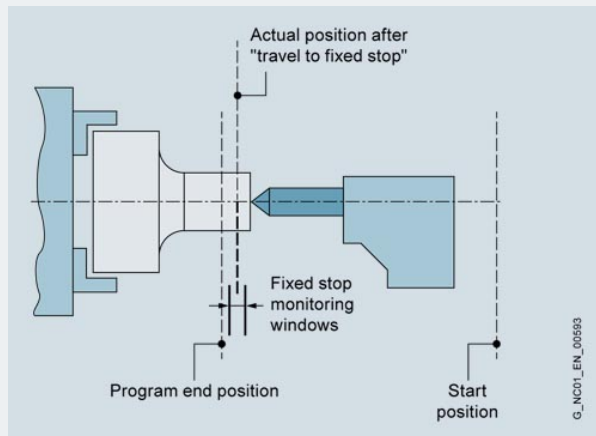
- Clamping of workpieces

Travel to fixed stop with Force Control

Option

Order code: M01

Article No.: 6FC5800-0BM01-0YB0



Function

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.

Benefits

- Clamping of workpieces with limited torque/limited force

Setpoint exchange

Option

Order code: M05

Article No.: 6FC5800-0BM05-0YB0

Function

Setpoint exchange is used on milling machines with special milling heads on which, for example, the spindle motor is used both for driving the tool and for orientation of the milling head. In this case, the spindle and the milling head axes are defined as independent axes in the CNC, but are powered by the same motor one after the other.

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

Benefits

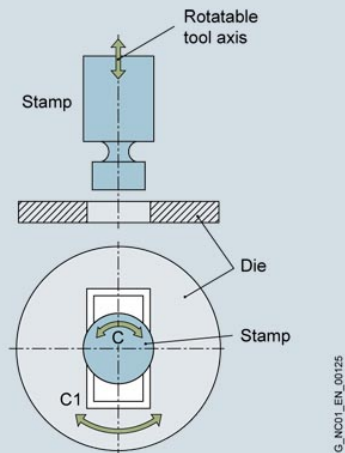
- Savings on costs by using a single drive for multiple tasks on the machine

Tangential control

Option

Order code: M06

Article No.: 6FC5800-0BM06-0YB0



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

Function

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

On punching and nibbling machines with a rotatable punching tool and associated die, the following functions may be used to ensure that the tool can be used universally:

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

Benefits

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

Position switching signals/cam controller

Option

Order code: M07

Article No.: 6FC5800-0BM07-0YB0

Function

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

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

Benefits

- Applications include e.g. the activation of protection areas
- Position-dependent initiation of movements, e.g. hydraulic oscillating axes during grinding

→ *High-speed CNC inputs/outputs*

Advanced Position Control ECO (APC ECO)

Option

Order code: M12

Article No.: 6FC5800-0BM12-0YB0

Function

Advanced Position Control ECO is a rugged favorably-priced vibration dampening process. Advanced Position Control ECO uses the status signals measured at the motor exclusively for closed-loop control. Signals from direct measuring systems or from external sensors are not used for the suppression of vibration.

To allow the function to be used successfully, the vibration to be dampened must have adequate, i.e. measurable, feedback on the motor of the axis.

Benefits

- Increase in productivity, as well as stability and quality during machining
- Machines which do not feature a direct measuring system, and in particular drives with linear or torque motors
- Machine axes where mechanical natural frequencies can be clearly identified in the motor signals

Advanced Position Control APC

Option

Order code: M13

Article No.: 6FC5800-0BM13-0YB0

Function

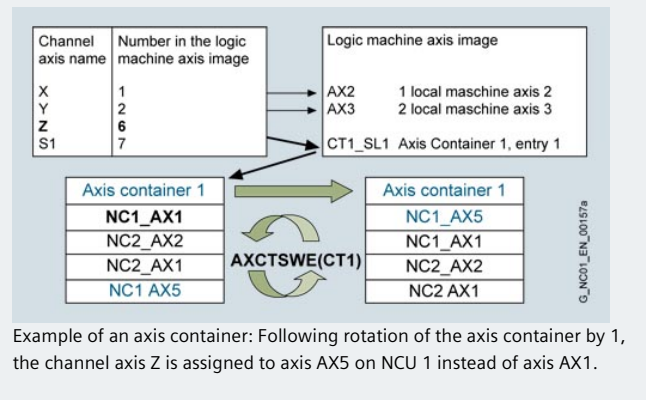
Mechanical natural frequencies of the machine axes can be actively damped with Advanced Position Control APC via a closed control loop. Dampening the oscillations is achieved by moving the motor to oppose the oscillation.

The status variable necessary for damping can be derived from the direct or/and indirect position measuring system, as well as from an external sensor, e.g. acceleration sensor.

Benefits

- Actively suppresses natural oscillations of the controlled machine axis which are evoked via the setpoint entry or via disturbance variables from the process.
- With identical mechanics, it is possible to set a higher k_v factor and generally also a higher dynamic response
- A direct consequence in this regard is an increase in productivity, as well as stability and quality during machining. This allows, for example, higher machining feedrates or increased chip thickness for heavy-duty cutting. Higher jerk and deceleration rates of the machine axes can be implemented for the finishing of surfaces in mold making.

Axis container



Function

On rotary indexing machines/multi-spindle machines, the axes holding the workpiece move from one machining unit to the next. Since the machining units are handled through different NCU channels, the axes holding the workpiece must be dynamically reassigned to the corresponding NCU channel if there is a change in station/position. Only one workpiece clamping axis/spindle is active on the local machining unit at a time. The axis container combines the possible connections to all clamping axes/spindles, of which only one is active at a time for the machining unit. The available axes that are defined in the axis container can be changed by shifting the entries in the axis container. Shifting can be triggered by the part program or synchronized actions: Keyword AXCTSWE(CT1).

Benefits

- Implementation of rotary indexing machines/multi-spindle machines for which the axes holding the workpiece move from one machining unit to the next machining unit.

Spindle functions

Function

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 spindle modes:
- Spindle speed with spindle override
- Five 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 SPOS¹⁾
- Spindle monitoring with the functions¹⁾:
 - Axis/spindle stationary ($n < n_{\min}$)
 - Spindle in setpoint range
 - Spindle speed, max.
 - Programmable lower (G25) and upper (G26) spindle speed limitation
 - Speed of the gear stage minimal/maximum
 - Maximum 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 surface finish and thus better surface quality.
- Spindle control via PLC for oscillation (so that it is easier to engage a new gear stage) and positioning
- Changeover to axis mode:

For machining with a position-controlled spindle (e.g. face machining of turned parts), the main spindle drive can be switched to the axis mode using a program command. A common encoder can be used for both axis and spindle modes. The zero mark of the spindle is also the reference mark of the C axis, so there is no longer any need to home the C axis (synchronize C axis on the fly).

Spindle functions

- Thread cutting with constant pitch¹⁾:
The following thread types can be produced with G33: cylindrical, taper and face thread, single-start or multiple-start, as left-hand or right-hand thread. In addition, multiple-block threads can be produced by concatenating threading blocks.
- Thread cutting with variable lead¹⁾:
Threads can also be programmed with linearly progressive (G34) or linearly degressive (G35) lead.
- Programmable thread run-in and run-out path:
When thread cutting, DITS/DITE (displacement thread start/end) can be used to program the path ramp for the acceleration or deceleration process as a distance. This makes it possible, for example, to adjust the acceleration on the thread shoulder when the tool run-in or run-out is too short and initiate smoothing at the next CNC start.
- Tapping with compensating chuck/rigid tapping:
When tapping with compensating chuck (G63), the compensating chuck equalizes differences between spindle movement and drilling axis. A prerequisite for rigid tapping (G331/G332) is a position-controlled spindle with position measuring system. The traversing range of the drilling axis is therefore not restricted. By using the method where the spindle, as a rotary axis, and the drilling axis interpolate, threads can be cut to a precise final drilling depth, e.g. for blind hole threads.

Benefits

- Spindle monitoring facilitates the diagnosis of faults
- Uniform surface finish and thus better surface quality as a result of constant cutting speed at the tool cutting edge
- Fast switchover between axis/spindle mode without referencing
- Machining of cylindrical, taper and face threads, single-start or multiple-start, as left-hand or right-hand thread. In addition, multiple-block threads can be produced by concatenating threading blocks and rigid tapping is possible.

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

Scalable analog setpoint Run MyCC /SANS

Option

Order code: N48

Article No.: 6FC5800-0BN48-0YB0

Function

Run MyCC /SANS implements isochronous switchover of analysis of an analog setpoint in the position controller via a high-speed digital output of the NCU.

Benefits

- Operation of spindles from Fischer

Requirement

Loadable compile cycle

Restricted functionality for export versions:
Not possible.

Linear interpolation

Function

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

Benefits

- The specified path is traced precisely

Restricted functionality for export versions:

The number of simultaneously interpolating axes is restricted to 4.

Circle via center point and end point

Function

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

The required circle is described by:

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

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

Benefits

- Simple programming of the circle, as dimensioned in the drawing

Circle via intermediate point and end point

Function

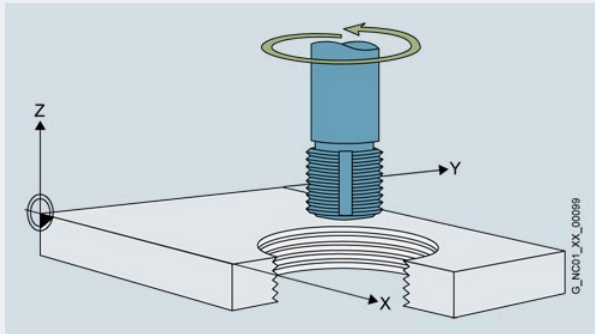
If a circle is to be programmed, which does not lie in a paraxial plane but obliquely in space, an intermediate point can be used to program it instead of the circle center.

Three points are required to program the circle: the starting point, the intermediate point and the end point.

Benefits

- Circular-path programming obliquely in space

Helical interpolation 2D+6



Helical interpolation: Thread milling with form cutter

Function

Helical interpolation composes the helix from 2 motions:

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

The programmed feedrate F either refers only to the circular movement or to the total path velocity of the 3 CNC axes involved. In addition to the 2 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.

Benefits

- Ideal for machining of internal or external threads with form cutters
- Ideal for milling of lubricating grooves

Universal interpolator NURBS (non-uniform rational B splines)

Function

Internal motion control and path interpolation are performed on the basis of non uniform rational B splines NURBS. This provides a uniform method for all internal interpolations that can also be used for future complex interpolation tasks.

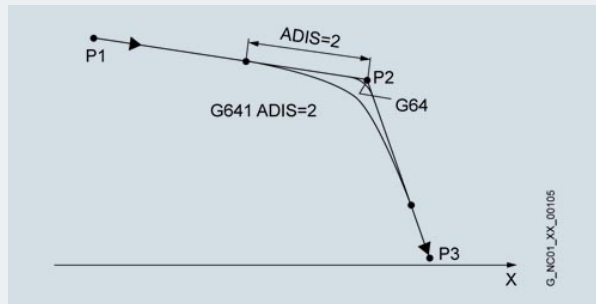
The following input formats are available irrespective of the internal structure:

- Linear interpolation
- Circular interpolation
- Helical interpolation
- Involute interpolation
- Splines (A, B, C)
- Polynomial

Benefits

- Structured program layout

Continuous-path mode with programmable rounding clearance



Continuous-path mode with programmable rounding clearance

Function

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.

If continuous-path mode G64 is selected, reduction in velocity takes place and contour corners are rounded at non-tangential transitions. With G641 ADIS=... a soft contour transition without a jump in acceleration can be programmed.

Benefits

- No undercuts are made on the workpiece because the tool does not stop at block boundaries

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

Option

Order code: M24

Article No.: 6FC5800-0BM24-0YB0

Function

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

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

Benefits

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

Requirements:

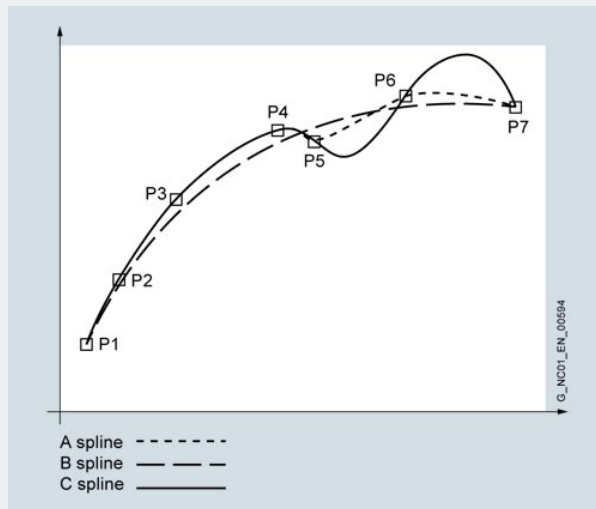
- Loadable compile cycle
- Cross-mode actions (option M43)

Spline interpolation (A, B and C splines)

Option

Order code: S16

Article No.: 6FC5800-0BS16-0YB0



Function

Using spline interpolation it is possible to obtain a very smooth curve from just a few defined interpolation points along a set contour. The interpolation points are connected by polynomials. The compressor converts linear movements, e.g., from CAD, at block transitions to splines of constant speed COMPON or splines of constant acceleration COMPCURV.

This yields soft transitions that reduce wear on the mechanical parts of the machine tool. However, if the interpolation points are placed close together, quite sharp edges can also be programmed. Spline interpolation also considerably reduces the number of program blocks required.

Extremely smooth workpiece surfaces are often very important in mold and tool making, both optically and technologically, e.g. for rubber seals. Tool radius compensation is possible in spline interpolation, as it is in linear or circular interpolation.

Every polynomial can represent a spline.

Only the algorithm determines the type of spline.

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

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

Benefits

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

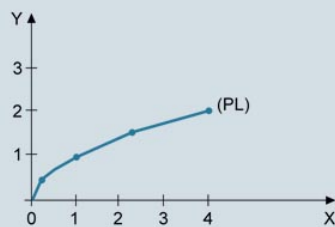
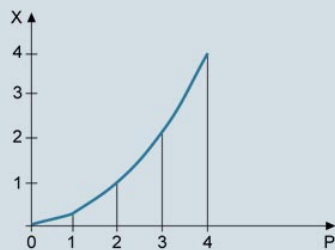
Polynomial interpolation

Option

Order code: M18

Article No.: 6FC5800-0BM18-0YB0

Example:
N9 X0 Y0 G90 F100
N10 POLY PO[Y] = 2 PO[X] = (4.025) PL = 4



G_MCO1_EN_00118

Polynomial interpolation

Function

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

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

Coefficient a_0 is the end point of the previous block, a_1 is calculated as the end point of the current block, a_2 , a_3 , a_4 and a_5 must be calculated externally and then programmed. With polynomial interpolation, it is possible to generate many different curve characteristics, such as linear, parabolic and exponential functions.

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

Benefits

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).

Requirement

For the efficient utilization of this polynomial interpolation, corresponding CAD/CAM systems are required.

Involute interpolation

Option

Order code: M21

Article No.: 6FC5800-0BM21-0YB0

Function

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 shorter machining time. Undesirable facets, which could result from coarse polygon functions, are thus avoided.

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

Benefits

Ideal for programming of complex spiral motions or contours.

Multi-axis interpolation > 4 interpolating axes

Option

Order code: M15

Article No.: 6FC5800-0BM15-0YB0

Function

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.

Benefits

- Implementation of multi-axis machines

Restricted functionality for export versions:
Not possible.

Crank interpolation Run MyCC /CRIP

Option

Order code: N04

Article No.: 6FC5800-0BN04-0YB0

Function

The Run MyCC /CRIP crank interpolation function supports simple programming and machining of pin bearing seats on a crankshaft. The function can be utilized in more than one channel, which means that a single workpiece can be machined simultaneously with several grinding units.

The function calculates the compensating movement of the grinding wheel in relation to the rotating workpiece surface. As with normal cylindrical grinding, the pin bearing journal is programmed as a radial distance X between the workpiece and the grinding wheel.

Benefits

Improved and user-friendly grinding of crankshaft bearing seats (cylindrical grinding) using existing technological cycles wherever possible.

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Not possible.

Advanced Surface motion control

Option

Order code: S07

Article No.: 6FC5800-0BS07-0YB0

Function

The Advanced Surface function is used to optimize the motion control.

Benefits

- More accurate contours and even more perfect surfaces can be achieved even at higher machining speeds.
- With optimized speed control, Advanced Surface delivers better workpiece surfaces with a higher workpiece yield.

Top Surface motion control

Option

Order code: S17

Article No.: 6FC5800-0BS17-0YB0

Function

The Top Surface option with its innovated COMPSURF compressor guarantees even more enhanced workpiece surfaces. Top Surface is considerably more proficient for demanding CAD/CAM programs. The workpiece surfaces are top quality, especially for non-homogeneous point distribution often visible with oblique line-by-line finishing programs and with bi-directional milling.

The Top Surface option provides additional benefits for the programmer and the machine operator: Preset default values for the contour and orientation tolerance, applicable for practically every part program, ensures independence from the calculation tolerance used in CAD/CAM.

Benefits

- Minimization of risks and high customer satisfaction
- Extended service life due to reduction of wear
- Appreciation for maximum surface quality

Requirement:

- Advanced Surface

Top Speed motion control

Option

Short designation: S13

Article No.: 6FC5800-0BS13-0YB0

Function

Top Speed uses FIR filters for velocity control.

A combination of Top Speed and Top Surface for maximum performance during 3-axis and 5-axis simultaneous machining is obligatory.

Benefits

- Increase of the axial jerk value during 3-axis and 5-axis simultaneous machining while maintaining high contour precision
- The immediate result is higher traversing speeds, leading to reduced machining times

→ *Top Surface motion control*

Computational motion control, Run MyCC /EMC

Option

Article No.: 6FC5800-0BN47-0YB0

Product ID: MCS2120

Function

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

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

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

Benefits

- Productivity is significantly increased once again due to higher dynamic responses
- No instabilities with so-called natural frequencies of the machine
- Increased stiffness of motion control ensures higher quality on the workpiece
- Subsequent enhancements of productivity and precision can be implemented with ease
- The Bearing or mechanical guides are subjected to vibrations – Run MyCC /EMC eliminates these vibrations.

Requirements:

- Loadable compile cycle

Note

Includes the nodding compensation Run MyCC /NOCO

Restricted functionality for export versions:

Not possible.

→ *Nodding compensation Run MyCC /NOCO*

Setting the jerk filter time Run MyCC /AJET

Option

Article No.: 6FC5800-0BN82-0YB0

Product ID: MCS2110

Function

Run MyCC /AJET permits the intelligent adaptation of the machine dynamic response. It offers a solution for high productivity while maintaining maximum precision.

Benefits

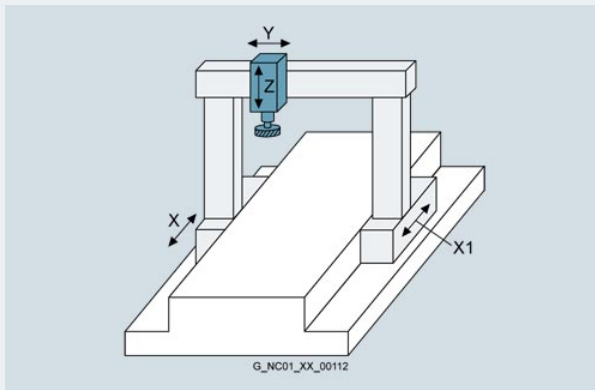
- Higher productivity through intelligent dynamic switchover
- Increased precision due to a more stable control loop
- Easy-to-use – dynamic response adaptations are automated via the part program

Pair of synchronized axes (gantry axes)

Option

Order code: M02

Article No.: 6FC5800-0BM02-0YB0



Gantry axes (pair of synchronized axes X/X1)

Function

The gantry axes function can be used to traverse the axes of up to 3 pairs of mechanically-coupled axes simultaneously without mechanical offset. The actual values are continuously compared and even the smallest deviations corrected. During both operation and programming, the axes defined in a gantry grouping are treated like a single machine axis. A gantry group consists of a guide axis and up to 2 synchronized axes. Two guide axes can be coupled using curve table interpolation.

Benefits

High-precision synchronous operation of two or multiple collinear, mechanically coupled axes. Only the guide axis is programmed.

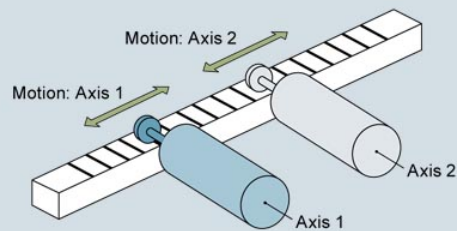
Master/slave for drives

Option

Order code: M03

Article No.: 6FC5800-0BM03-0YB0

Coupling: "OPEN"



Function

The "master/slave for drives" function is required when 2 electrical drives are mechanically linked to an axis. With this link, a torque controller ensures that both drives produce the exact same amount of torque, so that the motors do not work in opposition.

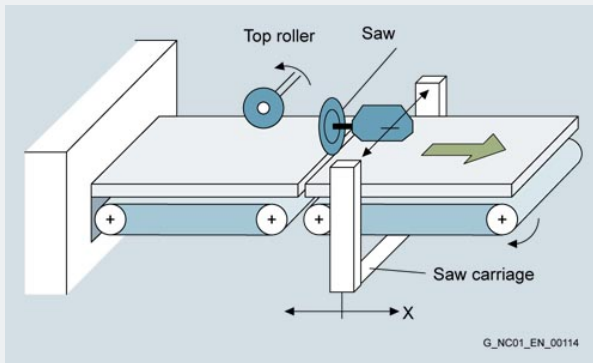
In order to attain tensioning between the master and slave drives, a tension torque specifiable via machine data can be applied to the torque controller.

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

Benefits

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

Master-value coupling and curve table interpolation



Example for cyclic machines: Flying saw

Function

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

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

Cyclic machines are distinguished by constantly repeated cyclic operations with high throughput and high productivity in machining, transport, packaging and parts handling (for example, packaging machines, presses, woodworking machines, printing machines).

In addition, the electronic functionality permits fast, axis-specific optimization, high-speed phase and path compensation, fast responses to faulty or missing parts, and fast synchronization and resynchronization, as well as decoupling from the leading axis and execution of autonomous movements. Axis cycles and synchronization calculations are carried out in the IPO cycle. Measuring from synchronized actions is used for detecting edges on continuous workpieces and for measuring print marks on continuous film, for example.

Master-value coupling and curve table interpolation

Benefits

- Technology functions such as synchronous operation, electronic transfer and positioning for cyclic machines
- Mechanical systems (line shaft, gearing, cam disks, couplings and cam disks) are replaced by an electronic solution (master value coupling, curve tables, synchronized actions, and electronic cams).

Restricted functionality for export versions:

The number of simultaneously traversing axes is restricted to 4.

→ *Generic coupling*

→ *Measuring stage 2 (option M32)*

→ *Synchronous spindles, multi-edge turning*

Virtual axis

Function

A virtual axis is one that is interpolated in the follow-up mode.

Benefits

Virtual axes are used in leading value couplings. Examples include:

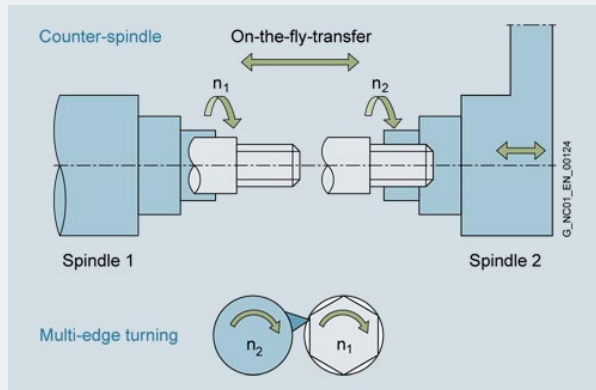
- Electronic transfer CP
- Master-value coupling with curve table
- Flying saw

→ *Axis/spindle*

→ *Positioning axis/auxiliary spindle*

→ *Simulation axis/spindle*

Synchronous spindles, multi-edge turning



Function

Precise angular synchronization between one leading and one or more following spindles enables on-the-fly workpiece transfer, particularly for turning machines, from spindle 1 to spindle 2, for example for the purpose of finishing.

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 D

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

Multi-edge turning:

$$n_2 = T \times n_1$$

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

Benefits

- Avoidance of idle times due to re-clamping

→ *Generic coupling*

Axes, coupled motion

Function

When a defined leading axis moves, the coupled-motion axes (following axes) assigned to it travel the traverse paths derived from the leading axis, taking into account a coupling factor (setpoint coupling). Together, the leading axis and the following axes form a coupled-axis grouping.

Definition and activation of a coupled-axis grouping take place simultaneously with the modal instruction TRAILON. A coupled-axis grouping can consist of any desired combinations of linear and rotary axes. A coupled-motion axis can be assigned up to 2 leading axes (in different coupled-axis groupings). A simulated axis can also be defined as the leading axis. In this case the real axis is traversed, taking into account the coupling factor.

Benefits

An application for coupled-motion axes is the use of 2 coupled-axis groupings to machine the 2 sides of a workpiece.

Axial coupling in the machine coordinate system MCS

Function

The subfunction "Axis collision protection" is implemented by Protect MyMachine /CC Axis Monitor.

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

→ *Collision protection Protect MyMachine /CC Axis Monitor*

→ *Generic coupling*

Electronic gear

Function

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).

Benefits

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 for export versions:

The number of simultaneously traversing axes is restricted to 4.

→ *Generic coupling*

Generic couplings

Function

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

The options for generic coupling 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 for export versions:

The number of simultaneously interpolating axes is restricted to 4

→ *Generic coupling CP-Standard*

→ *Generic coupling CP-Static*

→ *Generic coupling CP-Basic*

→ *Generic coupling CP-Comfort*

→ *Generic coupling CP-Expert*

Generic coupling CP-Standard

Function

Up to 4 × simple coupled motion with one leading axis and no use in synchronized actions.

Benefits

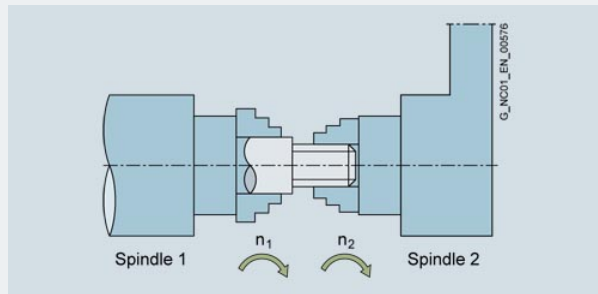
- Coupled motion of axes

Generic coupling CP-Static

Option

Order code: M75

Article No.: 6FC5800-0BM75-0YB0



Function

One simple synchronous spindle with coupling ratio 1:1.

Note

- No multi-edge machining

Benefits

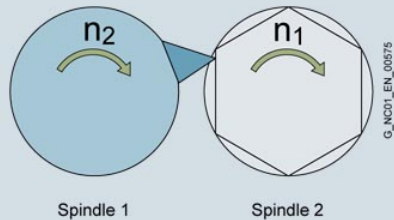
- Coupled motion of axes

Generic coupling CP-Basic

Option

Order code: M72

Article No.: 6FC5800-0BM72-0YB0



Function

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

Benefits

- Coupled motion of axes
- Multi-edge machining and synchronous spindle

Restricted functionality for export versions:

The number of simultaneously interpolating axes is restricted to 4.

Generic coupling CP-Comfort

Option

Order code: M73

Article No.: 6FC5800-0BM73-0YB0

Function

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

Additionally:

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

Benefits

- Coupled motion of axes
- Multi-edge machining and synchronous spindle
- Electronic gear without curve tables

Restricted functionality for export versions:

The number of simultaneously interpolating axes is restricted to 4.

→ *Electronic gear*

Generic coupling CP-Expert

Option

Order code: M74

Article No.: 6FC5800-0BM74-0YB0

Function

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

Additionally:

- 8 electronic gear for 3 leading axes with curve table interpolation and with cascading
- 5 electronic gear for 5 leading axes with curve table interpolation and with cascading

Benefits

- Coupled motion of axes
- Multi-edge machining and synchronous spindle
- Electronic gear with and without curve tables

Restricted functionality for export versions:

The number of simultaneously interpolating axes is restricted to 4.

→ *Electronic gear*

Variable-based axis movement Run MyCC /AMOV

Option

Order code: N62

Article No.: 6FC5800-0BN62-0YB0

Note

The specific function Run MyCC /AMOV is not intended for general use.

Restricted functionality for export versions:
Not possible.

Cartesian Point-to-Point travel PTP

Function

For handling and robot-related tasks, 2 types of movement are required: either in the Cartesian coordinate system Continuous Path CP, or as a Point-to-Point PTP motion.

With PTP, the shortest way to reach the end point is with activated TRAORI transformation. PTP generates a linear interpolation in the axis space of the machine axis.

By smoothing from PTP to CP movement, it is possible to switch from fast infeed to a mounting or positioning movement with optimum timing. PTP travel does not result in an axis overload when traveling through a singularity, such as the changing of an arm position during handling.

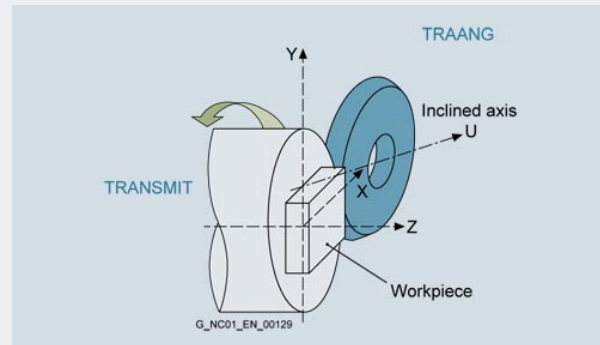
PTP travel is also possible in JOG control mode and does not require Cartesian positions (e.g., from CAD systems) to be converted into machine axis values.

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.

Benefits

- Support for handling and robot tasks, as well as for cylindrical grinding machines with inclined axis

Concatenated transformations



Grinding a TRANSMIT contour with inclined axis

Function

Two transformations can be concatenated with the TRACON command:

TRAANG (inclined axis), as the base transformation, can be linked with TRAORI (5-axis transformation), TRANSMIT (front end machining of turned parts) and TRACYL (cylinder surface transformation).

Benefits

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

Generic transformation

Function

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

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

Benefits

- Orientation of the tool in space can be arbitrarily defined

Restricted functionality for export versions:

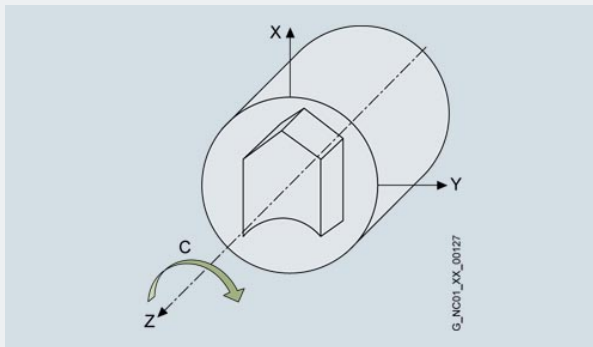
Not possible.

TRANSMIT/cylinder surface transformation

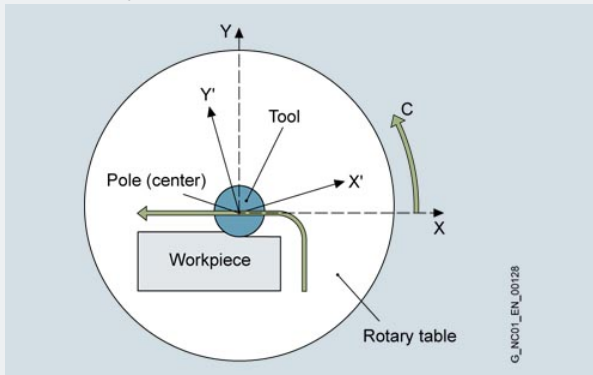
Option

Order code: M27

Article No.: 6FC5800-0BM27-0YB0



Face machining with TRANSMIT



Tool center point path through the pole

Function

TRANSMIT is used for milling outside contours on turned parts, e.g. square head – linear axis with rotary axis. As a result, the programming process becomes much more simple, and complete machining increases machine efficiency: Turning and milling can be performed on one machine without rechucking.

3D interpolation with 2 linear axes and one rotary axis is possible. The two linear axes are mutually perpendicular and the rotary axis lies at right angles to one of the linear axes. TRANSMIT can be called up in different channels simultaneously. The function can be selected and deselected with a preparatory function (straight line, helix, polynomial and activating tool radius compensation) in the part program or MDI.

With TRANSMIT, the area of the transformation pole is reached when the tool center can be positioned at least to the turning center of the rotary axis entering the transformation.

TRANSMIT through the pole is implemented in different ways:

- When traveling through the pole, the rotary axis is turned automatically through 180° when the turning center is reached and the remaining block is then executed.
- When traversing close by the pole, the CNC automatically reduces the feedrate and the path acceleration.
- If the path contains a corner in the pole, the position jump in the rotary axis is compensated by the control through automatic block insertion.

Benefits

The cylinder surface transformation is used for turning machines and milling machines and allows machining of the peripheral surface, for example on turned parts.

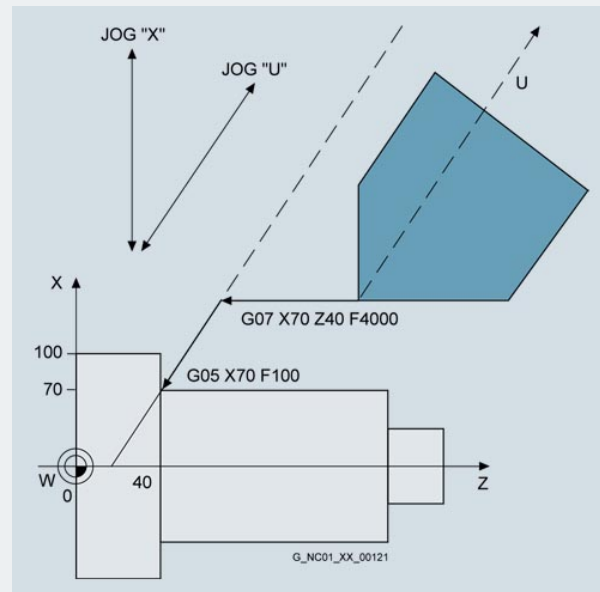
The cylinder surface transformation or TRACYL cylinder surface transformation can be used to manufacture grooves of any shape on the surface of cylindrical bodies with or without groove side offset. The shape of the grooves is programmed in reference to the unwound plane cylinder surface.

Inclined axis

Option

Order code: M28

Article No.: 6FC5800-0BM28-0YB0



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

Function

The inclined axis function is used for fixed-angle interpolation using an inclined infeed axis. This is used primarily in conjunction with cylindrical grinding machines. The axes are programmed and displayed in the Cartesian coordinate system. Tool offsets and work offsets are also entered in the Cartesian system and transformed to the real machine axes. For oblique plunge-cutting with G05, it is necessary to program the start position with G07.

In the JOG CNC operating mode, the grinding wheel can be traversed either in the Cartesian coordinate system or in the direction of inclined axis U (selection via the channel DB).

Benefits

- Inclined axis for cylindrical grinding

Geometrically redundant axes Run MyCC /GRED

Option

Order code: N73

Article No.: 6FC5800-0BN73-0YB0

Note

The specific function Run MyCC /GRED is not intended for general use.

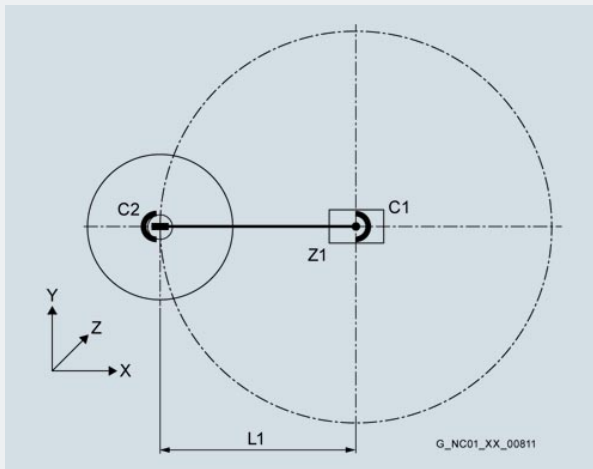
Restricted functionality for export versions:
Not possible.

Transformation: Rotating workpiece and tool, Run MyCC /2RPT

Option

Order code: N43

Article No.: 6FC5800-0BN43-0YB0



Function

Run MyCC /2RPT transformation rotating workpiece and tool is a special 3-axis transformation. The path and path velocity are programmed in the same way as for 3-axis systems. Run MyCC /2RPT calculates the resultant motion of the 3 machine axes. The generated machining programs are therefore not machine-specific.

With these kinematics, the tool is swiveled around a rotary axis (C1) that rotates precisely about the Z direction of the basic coordinate system. The workpiece is turned by the second rotary axis (C2) which travels in parallel to the first axis.

At the zero position of the first rotary axis, the toolholder is positioned exactly above the center of rotation of the second rotary axis. The tool unit also has a serially arranged Z axis (Z1).

The basic coordinate system is at the center of rotation of the second rotary axis. Length L1 describes the distance between the two rotary axes. The X coordinate of the basic coordinate system points towards the first rotary axis.

Benefits

- Implementation of machines with specific kinematics

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

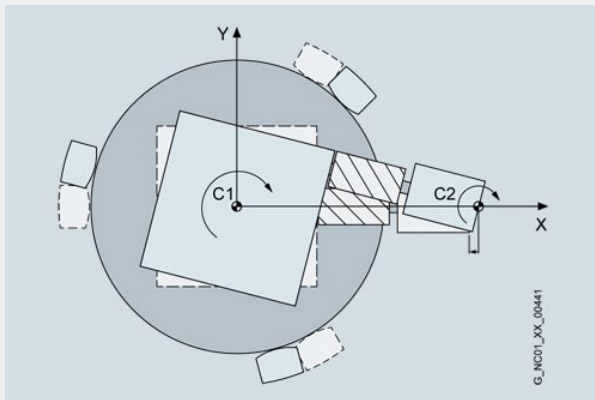
Not possible.

Transformation: DOUBLETRANSMIT Run MyCC /2TRA

Option

Order code: M25

Article No.: 6FC5800-0BM25-0YB0



Function

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

Benefits

- Implementation of machines with specific kinematics

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Not possible.

Transformation: Double Generic, Run MyCC /DGEN

Option

Order code: N34

Article No.: 6FC5800-0BN34-0YB0

Function

With Run MyCC /DGEN double generic transformation it is possible to couple two 5-axis kinematics for a machining task. The axes of both 5-axis machines must be programmed in one channel. Programming coupled 5-axis kinematics is subject to certain restrictions that do not apply to normal 5-axis programming. The application is used in 5-axis milling operations on thin structures (e.g. outer shells of aircraft) that require simultaneous movement of 5-axis kinematics as a counter-support immediately beneath the tool center point.

Benefits

- Coupling of two 5-axis kinematics for one machining task

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Not possible.

Transformation: Dynamic swivel TRIPOD, Run MyCC /DSTT

Option

Order code: M84

Article No.: 6FC5800-0BM84-0YB0

Function

The kinematic transformation Dynamic Swivel TRIPOD Run MyCC /DSTT is a 5-axis or 6-axis transformation function 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.

Benefits

- Implementation of machines with specific kinematics

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

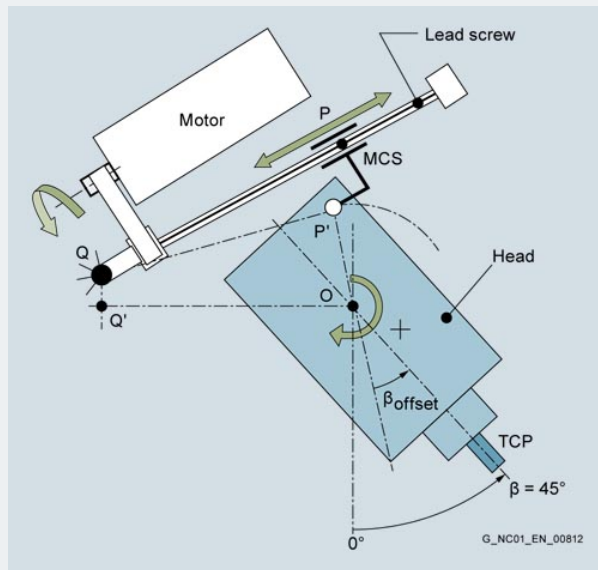
Not possible.

Transformation: Eccentric axis, Run MyCC /ECCA

Option

Order code: N44

Article No.: 6FC5800-0BN44-0YB0



Function

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

Benefits

- Implementation of machines with specific kinematics

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

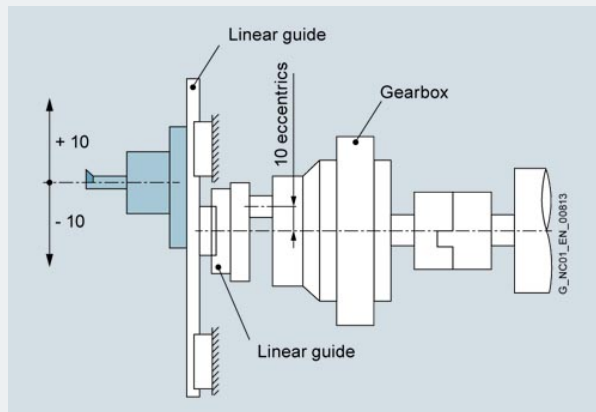
Not possible.

Transformation: Eccentric, Run MyCC /ECCE

Option

Order code: N41

Article No.: 6FC5800-0BN41-0YB0



Function

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

Benefits

- Implementation of machines with specific kinematics

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Not possible.

Transformation: HEXAPOD, Run MyCC /HEX

Option

Order code: M71

Article No.: 6FC5800-0BM71-0YB0

Function

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

Versions:

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

It therefore allows a tool (e.g. milling cutter, laser beam) to be oriented to the workpiece in the machining space. The path and path velocity are programmed in the same way as for 3-axis systems. The tool orientation is programmed additionally in the traversing blocks.

Run MyCC /HEX calculates the resulting motion of all 6 axes. The generated machining programs are therefore not machine-specific. CNC functions such as tool radius and tool length compensation, and the selection of coordinate systems, are available without any restriction.

Benefits

- Implementation of parallel kinematics machines PKM

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Not possible.

Transformation: PARACOP 3 axes, Run MyCC /PACO

Option

Order code: M44

Article No.: 6FC5800-0BM44-0YB0

Function

PARACOP kinematic transformations and pantograph kinematics are used on parallel-kinematics machines PKM. Parallel kinematics means that the drive forces engage on the spindle head (Stewart platform) simultaneously (in parallel). PARACOP machines are tripod systems on which the Stewart platform is moved by 3 actuators. Design measures are used to ensure that the Stewart platform cannot move in an undefined manner on these tripod types. On PARACOP machines, 2 parallel rods run on a slide for each actuator. These machines are suitable for 3-axis machining. When using kinematic transformations, workpieces can be programmed in Cartesian coordinates as usual. The SINUMERIK CNC calculates the required motion of the machine axes online. Therefore, the programmer can create part programs in the same way as on a conventional machine, and does not have to take the special kinematics of the machine into account.

Benefits

- Implementation of parallel kinematics machines PKM

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Not possible.

→ *Transformation pantograph kinematics Run MyCC /SCIS (option M51)*

Transformation: Swivel axis, Run MyCC /PIVA

Option

Order code: N52

Article No.: 6FC5800-0BN52-0YB0

Function

The Run MyCC /PIVA transformation swivel axis allows the user to program in a Cartesian coordinate system with X, Y and Z axes, while at the machine, axes X1 and Z1 plus a rotary axis AY are moved.

The rotary axis moves a rotatable arm with driven tools around the X axis in the Y-Z plane. The tools are always positioned in parallel to the X axis.

Benefits

- Implementation of machines with specific kinematics

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

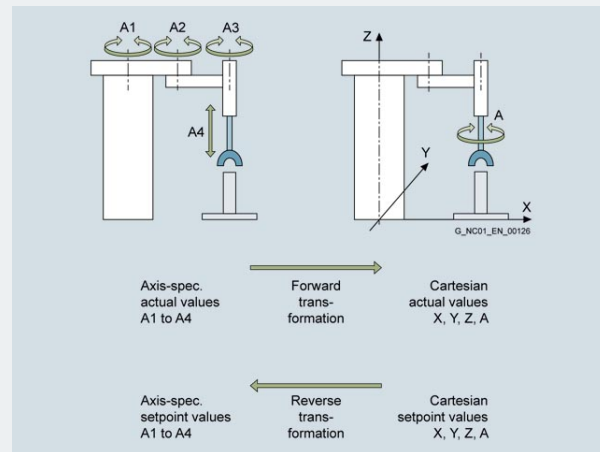
Not possible.

Transformation: Handling, Run MyCC /RCTRA

Option

Order code: M31

Article No.: 6FC5800-0BM31-0YB0



Transformation package, handling

Function

The Run MyCC /RCTRA handling transformation contains the so-called standard transformation block, by means of which typical 2-axis to 4-axis handling devices such as gantries or Scara kinematics can be implemented.

This coordinate transformation converts programmed Cartesian setpoints (e.g. X, Y, Z, A) into axis positions (e.g. A1 to A4) for the handling device.

Note

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

Benefits

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

Requirement:

Loadable compile cycle

Restricted functionality for export versions:

Not possible.

→ *Transformation robotic extended Run MyCC /ROBX (option N54)*

Transformation: Robotic extended, Run MyCC /ROBX

Option

Order code: N54

Article No.: 6FC5800-0BN54-0YB0

Function

The Run MyCC /ROBX transformation supports robot kinematics with between 4 and 6 axes.

A 6-axis robot can orientate the tool in any desired relationship to the workpiece in every point of the machining space. To improve accessibility to machining points on large workpieces, up to 3 additional linear axes, which move the robot or the workpiece, can be included in the transformation. The workpiece can be turned in space by an additional 3 rotary axes that are also included in the transformation.

Tool motion and orientation are still programmed with reference to the workpiece.

Benefits

- Implementation of robot kinematics with the SINUMERIK CNC

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Not possible.

Transformation for articulated arm robots Run MyRobot /ROBX AR

Option

Order code: R05

Article No.: 6FC5800-0BR05-0YB0

Function

Run MyRobot /ROBX AR allows high-precision path control for 6-axis robots. This function is marketed under the name Run MyRobot /Direct control.

Benefits

- High-precision path control with selected robot types
- Easy and familiar operation via SINUMERIK Operate
- Complete CNC programming and process integration
- Simple and fast commissioning due to auto-install script

Requirements:

- Loadable compile cycle
- Multi-axis interpolation > 4 interpolating axes
- 3 additional axes/spindles
- SINUMERIK Operate /NCU
- Access protection for cycles Lock MyCycles
- Automated commissioning Create MyConfig

Restricted functionality for export versions:

Not possible.

→ Multi-axis interpolation > 4 interpolating axes (option M15)

→ 3 additional axes/spindles (option A03)

→ SINUMERIK Operate /NCU (option S00)

→ Access protection for cycles Lock MyCycles (option P54)

→ Automated commissioning Create MyConfig

Robot Compensation Accuracy Run MyCC /ROCO Accuracy

Option

Article No.: 6FC5800-0BN83-0YB0

Function

Run MyCC /ROCO Accuracy is a mechatronic extension function for 6-axis robots for operation with SINUMERIK ONE.

Run MyCC /ROCO Accuracy consists of 3 independent subfunctions:

- Torque feedforward control for the reduction of the position and velocity-dependent following error
- 3D nodding compensation for the compensation of acceleration processes in the respective robot axis and across axes (over-coupling)
- Compensation of external forces, e.g. gravitational or process forces

Benefits

- Increasing the positioning and machining path accuracy of the robot

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Not possible.

→ *Nodding compensation Run MyCC /NOCO*

Robot Compensation Productivity Run MyCC /ROCO Productivity

Option

Article No.: 6FC5800-0BN84-0YB0

Function

Run MyCC /ROCO Productivity is a mechatronic extension function for 6-axis robots for operation with SINUMERIK ONE.

Run MyCC /ROCO Productivity ensures the dynamic adaptation of acceleration and jerk during preprocessing by utilizing the maximum axis torque.

Benefits

- Increasing the axis dynamics for handling tasks performed in the entire working area of the robot
- Shortening the machining time

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Not possible.

Transformation: Rotating eccentric, Run MyCC /ROTE

Option

Order code: N37

Article No.: 6FC5800-0BN37-0YB0

Function

Run MyCC /ROTE transformation rotating eccentric is a 4-axis transformation. The path and path velocity are programmed in the same way as for 3-axis systems. Run MyCC /ROTE calculates the resultant motion of the 4 machine axes. The generated machining programs are therefore not machine-specific.

The Y coordinate is converted to a corresponding position of axis 4. Compensating motion of axes 1 and 2 are then calculated as a function of axis 3.

Benefits

- Implementation of machines with specific kinematics

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Not possible.

Transformation: Pantograph kinematics, Run MyCC /SCIS

Option

Order code: M51

Article No.: 6FC5800-0BM51-0YB0

Function

The Run MyCC /SCIS transformation pantograph kinematics is a type of 2/4-axis transformation with parallel kinematics. The transformation can work with variable-length rods, or with movable base points of fixed-length rods.

The SINUMERIK CNC calculates the required motion of the machine axes online. Therefore, the programmer can create part programs in the same way as on a conventional machine, and does not have to take the special kinematics of the machine into account. Workpieces can be programmed in the usual way in Cartesian coordinates.

Benefits

- Implementation of machines with pantograph kinematics

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

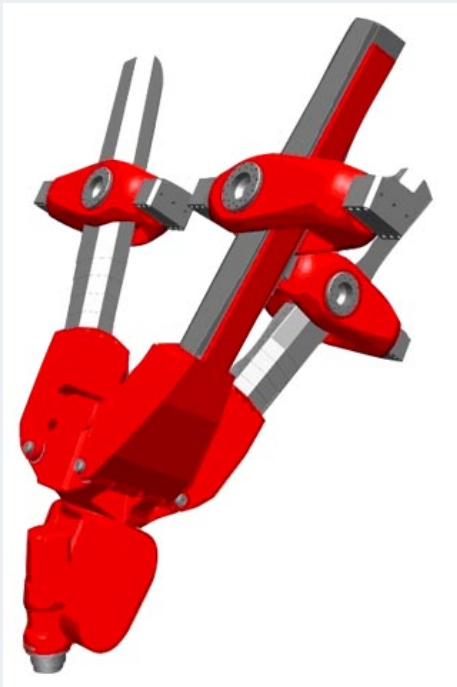
Not possible.

Transformation: TRIPOD hybrid kinematics, Run MyCC /THYK

Option

Order code: N36

Article No.: 6FC5800-0BN36-0YB0



Parallel kinematic type Exechon

Function

The kinematic transformation TRIPOD hybrid kinematics THYK is a 5-axis transformation for 3 parallel linear axes and 2 rotary axes. The tool movements are programmed complete with their orientation in the same manner as the Cartesian 5-axis machine. The real-time transformation calculates the necessary paths and speeds for the real machine axes.

Benefits

- Implementation of machines with TRIPOD hybrid kinematics

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

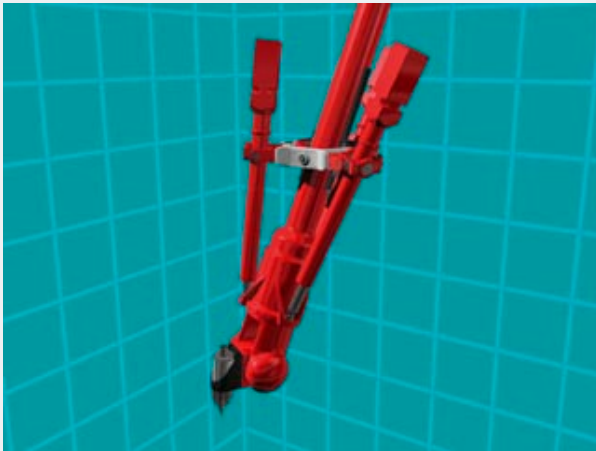
Not possible.

Transformation: TRICEPT 5-axis transformation, Run MyCC /TRIC

Option

Order code: M46

Article No.: 6FC5800-0BM46-0YB0



TRICEPT animation

Function

Run MyCC /TRIC is a kinematic transformation for a TRICEPT (tripod with center tube). TRICEPT kinematic transformations are used on parallel-kinematic machines PKM. Parallel kinematics means that the drive forces engage on the spindle head (Stewart platform) simultaneously (virtually in parallel).

TRICEPT machines are TRIPOD systems on which the Stewart platform is moved by 3 actuators. Design measures are used to ensure that the Stewart platform cannot move in an undefined manner. On TRICEPT, an additional passive telescope (center tube) is used.

On TRICEPT, 2 additional rotary axes are required to define the tool orientation in space. These axes can be arranged like the fork head of a 5-axis machine, for example, thus allowing the machine to carry out 5-axis machining.

The SINUMERIK CNC calculates the required motion of the machine axes online. Therefore, the programmer can create part programs in the same way as on a conventional machine, and does not have to take the special kinematics of the machine into account.

Benefits

- Implementation of machines with TRICEPT (tripod with center tube) kinematics

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

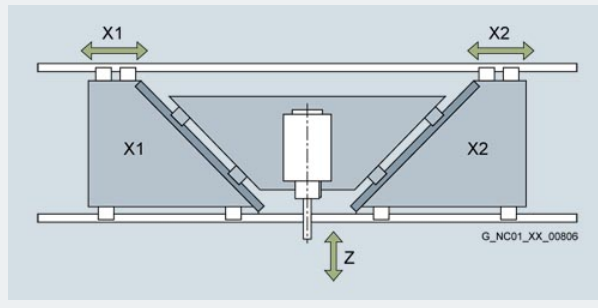
Not possible.

Transformation: Double slide, Run MyCC /SKID

Option

Order code: M80

Article No.: 6FC5800-0BM80-0YB0



Function

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

Benefits

- Implementation of machines with specific kinematics

Restricted functionality for export versions:

Not possible.

Transformation: GRINDING, Run MyCC /GRIN

Option

Order code: N40

Article No.: 6FC5800-0BN40-0YB0

Function

The specific function Run MyCC /GRIN is not intended for general use.

Restricted functionality for export versions:

Not possible.

Transformation: W-Z Tracking, Run MyCC /WZTR

Option

Order code: N22

Article No.: 6FC5800-0BN22-0YB0

Note

The specific function Run MyCC /WZTR is not intended for general use.

Restricted functionality for export versions:

Not possible.

Transformation additional function 6th axis Run MyCC /TR-6A

Option

Order code: M66

Article No.: 6FC5800-0BM66-0YB0

Note

The specific function Run MyCC /TR-6A is not intended for general use.

Restricted functionality for export versions:

Not possible.

Transformation: Redundant axes at workpiece Run MyCC /RDCC

Option

Order code: N26

Article No.: 6FC5800-0BN26-0YB0

Function

The Run MyCC /RDCC kinematic transformation redundant axes at workpiece is a multi-axis transformation. This function is used particularly for the purpose of assembling large aircraft components which need to be positioned and oriented in a working area. The component is aligned by programming 3 Cartesian coordinates and 3 orientation angles.

The function can move up to 31 axes in such a way that the contact points on the component, to each of which up to 3 axes are assigned, do not change their reciprocal positions relative to the moving part.

In combination with the additional function Run MyCC /AMOV, axes of another NCU coupled by means of NCU-Link can also be included in the interpolation with the result that the motion of more than 31 axes in total can be synchronized.

Benefits

- Assembly of large aircraft components which need to be positioned and oriented in a working area

Restricted functionality for export versions:

Not possible.

→ *Variable-based axis movement Run MyCC /AMOV (option N62)*

Differential tower transformation, Run MyCC /DTOW

Option

Order code: N68

Article No.: 6FC5800-0BN68-0YB0

Note

The specific function Run MyCC /DTOW is not intended for general use.

Restricted functionality for export versions:

Not possible.

Measuring functions/measuring cycles

Function

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

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

- Measuring in JOG
- Two 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

Benefits

- Ensuring the machining precision
- Shortening of idle times
- Elimination of error sources
- Automation of additional production processes

Measuring stage 1

Function

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

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

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

Benefits

- Influencing the process during machining and ensuring machining precision

Measuring stage 2

Option

Order code: M32

Article No.: 6FC5800-0BM32-0YB0

Function

While the measuring function in motion blocks in the part program is limited to one block, you can activate measuring functions from synchronized actions at any time, independent of the part program. The measuring events can be assigned to the axes in the CNC block. In the case of simultaneous measuring, up to 4 trigger events can be evaluated per position control cycle. Measured values are read as a function of the 3 parameters: Probe, axis and measuring edge.

In the case of continuous (cyclic) measurement, the measurement results are written to a FIFO variable. Endless measurement can be achieved by reading out the FIFO values cyclically. Measurement results can be logged in the form of a file. The measuring cycles (option, order code P28) contain a standard log, which can be freely configured by the user.

Benefits

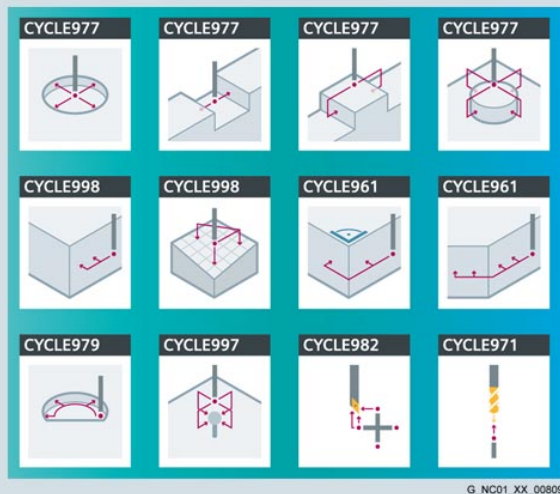
- Influencing the process during machining
- Ensuring machining precision
- Logging of the measurement results

Measuring cycles for drilling/milling and turning

Option

Order code: P28

Article No.: 6FC5800-0BP28-0YB0



Measuring cycles for drilling/milling and turning

Function

Measuring cycles are general subroutines designed to solve specific measurement tasks. They can be adapted to specific problems via parameter settings.

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

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

The result of the workpiece measurement can either include an automatic work offset 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. SINUMERIK measuring cycles offer a standard log, which can be freely configured by the user.

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

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

Measuring cycles for drilling/milling and turning

Option

Order code: P28

Article No.: 6FC5800-0BP28-0YB0

- Milling measurement:
 - Corner/right-angled corner/any corner
 - Calibrate probe: Length/radius in ring/radius at edge/calibrate on sphere
 - Hole/rectangular pocket
 - Hole/inner circle segment
 - Edge distance – slot/web
 - Edge distance – set edge
 - Edge distance – align edge
 - Circular/rectangular spigot
 - Spigot/outer circle segment
 - 3-D measurement – aligning the plane
 - 3D measurement – 1 sphere/3 spheres
 - Measure tools
 - Measure tools – calibrate tool probe
- Turning measurement:
 - Calibrate probe – length/radius at surface/radius in slot
 - Inner/outer diameter
 - Measure tools – turning tools/milling tools/drills
 - Measure tools – calibrate tool probe

Note

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

Benefits

- Almost all measuring tasks that have to be realized in a turning/milling machine can be handled
- Reliable quality of the manufactured parts by automatic measurement directly in the machine
- Fast programming even for complex measuring tasks thanks to input screens with graphic support
- Superior workpiece accuracy and optimum transparency of the quality data

Measure kinematics

Option

Order code: P18

Article No.: 6FC5800-0BP18-0YB0

Function

The Measure kinematics function supports the calibration of kinematic structures of 5-axis machines. The SINUMERIK ONE is able to determine the parameters of kinematic transformations of the digitally or manually alignable rotary axes quickly and automatically.

Benefits

- First commissioning possible without dimensioned layout of the machine
- Regular check of the production process wherever high precision is required

Drive current measurement Run MyCC /KPXT

Option

Order code: M82

Article No.: 6FC5800-0BM82-0YB0

Note

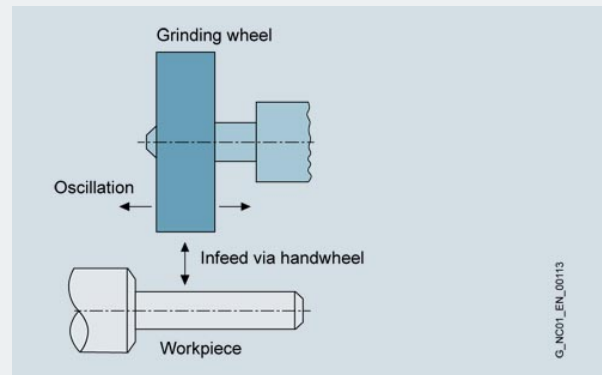
The specific function Run MyCC /KPXT is not intended for general use.

Restricted functionality for export versions:

Not possible.

→ *Auto Servo Tuning AST*

Handwheel override



Handwheel override in AUTO CNC operating mode

Function

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

Benefits

- Implementation of grinding cycles with manual intervention conditions, e.g. locating the initial grinding point

Contour handwheel

Option

Order code: M08

Article No.: 6FC5800-0BM08-0YB0

Function

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

A feedrate specified via the part 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 up to the start of the block – continuation is prevented

Benefits

- Used with conventional turning and grinding machines for setup/scratching
- More user-friendly operation of the machine in setup mode

→ *Feedrate interpolation (feed characteristic)*

Punching/nibbling

Option

Order code: M33

Article No.: 6FC5800-0BM33-0YB0

Function

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

- Language commands
The punching/nibbling functions are activated and deactivated using simple, clear high-level language elements: 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 CNC's position-control cycle. This allows maximum speed to be attained with utmost precision.
- Automatic path segmentation, selectable:
 - Stroke path SPP
With SPP, the travel path is subdivided into programmable segments of identical size (modal effect).
 - Stroke rate SPN
SPN subdivides the travel path into a programmable number of path sections (non-modal effect).

Benefits

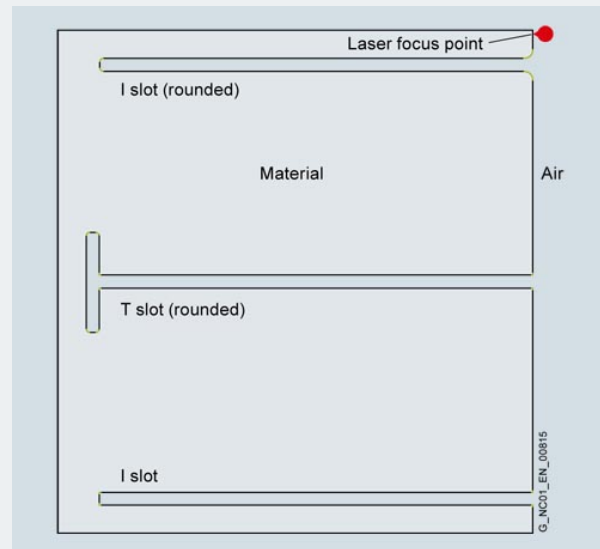
- Implementation of machines for punching and nibbling

Slot shape recognition

Option

Order code: S18

Article No.: 6FC5800-0BS18-0YB0



Function

With slot shape recognition and active tool radius compensation, the tool paths are monitored through look-ahead contour calculation. The CNC allows potential contour violations on the workpiece to be detected in good time and permits the control to actively avoid them by omitting such contours.

In certain technological situations, however, it is suitable to permit the machining operation even in case of contour violation. The slot shape recognition option therefore tries to detect specific contours or contour definitions with the aim of preventing the removal of related CNC blocks through tool radius compensation in case of an impending contour violation.

The slot shape recognition function detects two contour definitions which are relevant for 2D laser machining - the "I slot" and the "T slot". If, for these contour definitions, the laser focus point is wider than the slot width, the slot is machined with activated slot shape recognition, and the programmed contour is violated deliberately.

Benefits

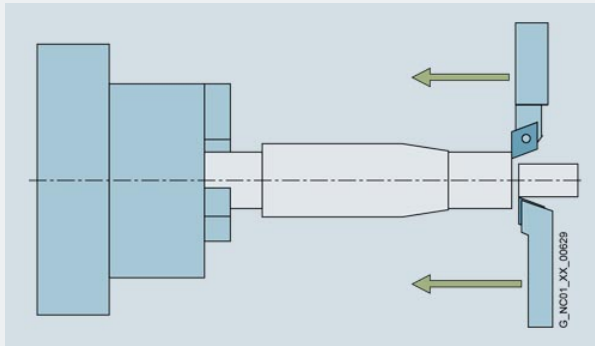
- Timely detection of contour violations on the workpiece and active prevention by the CNC
- Intentional violation of the contour in certain technological situations

Balance cutting: 4-axis roughing

Option

Order code: S05

Article No.: 6FC5800-0BS05-0YB0



Function

Balance cutting (4-axis roughing) allows a workpiece contour to be machined with 2 tools arranged opposite to one another simultaneously. Doubling the cutting volume per unit of time can be achieved either by doubling the feedrate or by doubling the effective infeed. To do this, different strategies of tool path control are required.

Benefits

- Increased productivity and precision:
The number of parts is increased and unstable workpieces are supported
- Complete system integration:
The work step is configured by setting fewer parameters. Synchronization of the tools is implemented automatically in the background.

Interpolation turning

Option

Order code: P57

Article No.: 6FC5800-0BP57-0YB0

Function

Interpolation turning enables turning operations on machining centers without a rotary table, as well as off-center turning on machining centers with rotary tables or lathes with a B axis.

During interpolation turning the CNC guides the turning tool in a position-controlled way around a fixed workpiece.

To do this, the main spindle of the machining center is changed to axis mode. The feed axes interpolate in a spiral/circular arrangement in the x-y/x-y-z plane, while at the same time the main spindle with the turning tool tracks the feed axes.

Benefits

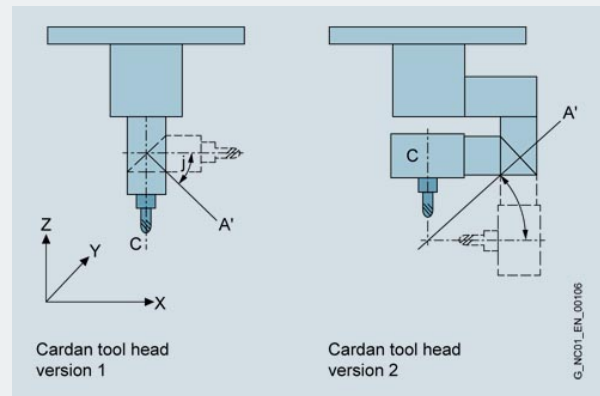
- Off-center turning on machining centers (milling machines) and lathes with a B axis with reduced machining time because machining operations are completed in one clamping operation.
- Reduced investment costs for the end customer

Machining package 5 axes

Option

Order code: M30

Article No.: 6FC5800-0BM30-0YB0



Function

With the machining package 5 axes, 5-axis machining tasks can be implemented.

- 5-axis transformation with tool orientation TRAORI
In 5-axis machining, geometry axes X/Y/Z are supplemented by additional axes such as rotary axes for swiveling the tool. The machining task can be completely defined in Cartesian spatial coordinates with Cartesian position and orientation. The path vector is converted in the control into the machine axes, including position and orientation, via 5-axis transformation.
- 5-axis tool length compensation for 5-axis machining
When machining with the 4th/5th axis, the lengths of the selected tool are automatically included and compensated in the axis movement.
- Oriented tool retraction
If machining is interrupted e.g. because of tool breakage, a program command can be used to carry out defined, oriented tool retraction.
- Tool-oriented RTCP
With the Remote Tool Center Point RTCP function, the tool swivel axes can be positioned in the manual mode, as long as there is compliance with the tool center point marked by the tool tip. The RTCP function simplifies the inclusion of program interpolation points in manual mode with orientation of the tool.
- Cardan milling head/nutating head
Requirements: Machining package 5 axes with 5-axis transformation.
Using a cardan milling head in conjunction with the Nutating head function, it is possible to machine outside contours of spatially shaped parts at high feedrates. To do this, the CNC performs a 5-axis transformation. The 3 translatory main axes X/Y/Z determine the tool operating point; 2 rotary axes, one of which is an inclined axis (angle can be set in the machine data), permit virtually any orientation in the working area. Cardan milling heads, versions 1 and 2 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

- Includes multi-axis interpolation > 4 interpolating axes.
- The machining package 5 axes does not include any additional axes/spindles which might be required.

Machining package 5 axes

Option

Order code: M30

Article No.: 6FC5800-0BM30-0YB0

Benefits

- Executing 5-axis machining tasks easily and conveniently, e.g. milling of free-form surfaces

Restricted functionality for export versions:

Not possible.

→ *Multi-axis interpolation > 4 interpolating axes (option M15)*

→ *Axis/spindle*

Machining package 5 axes additional function 7th axis

Option

Order code: S01

Article No.: 6FC5800-0BS01-0YB0

Function

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 working area, with 5th/6th axis tool kinematics active at the same time.

Benefits

- Implementation of fiber placement machines in the aerospace industry

Restricted functionality for export versions:

Not possible.

Milling technology package SINUMERIK MDynamics 3 axes

Option

Order code: S32

Article No.: 6FC5800-0BS32-0YB0

Function

The Milling technology package 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 (A, B and C splines)
- TRANSMIT/cylinder surface transformation
- Measuring cycles for drilling/milling and turning
- User memory additionally on the User SD Card of the NCU

Benefits

- Surfaces with increased perfection due to innovated motion control and an optimized CNC data compressor
- Quick adaptation to the workpiece, tool, and program handling
- Optimum machining by way of flexible programGUIDE and ShopMill programming, and consequently the shortest of programming times

Milling technology package SINUMERIK MDynamics 5 axes

Option

Order code: S33

Article No.: 6FC5800-0BS33-0YB0

Function

The Milling technology package SINUMERIK MDynamics 5 axes contains the options:

- Machining package 5 axes
- Multi-axis interpolation > 4 interpolating axes
- ShopTurn/ShopMill
- Residual material detection and machining for contour pockets and stock removal
- 3D simulation 1 (finished part)
- Simultaneous recording
- Advanced Surface
- Spline interpolation (A, B and C splines)
- TRANSMIT/cylinder surface transformation
- Measuring cycles for drilling/milling and turning
- User memory additionally on the User SD Card of the NCU
- 3D tool radius compensation
- Measure kinematics

Benefits

- Surfaces with increased perfection due to innovated motion control and an optimized CNC data compressor
- Quick adaptation to the workpiece, tool, and program handling
- Optimum machining by way of flexible programGUIDE and ShopMill programming, and consequently the shortest of programming times

Restricted functionality for export versions:

Not possible.

Cartesian travel to fixed stop Run MyCC /FXSC

Option

Order code: N38

Article No.: 6FC5800-0BN38-0YB0

Function

Cartesian travel to fixed stop Run MyCC /FXSC is used as a substitute for the standard travel to fixed stop function for machine kinematics with OEM transformation. With this function, e.g. tailstocks or sleeves can be traversed to a fixed stop in order to clamp workpieces.

The clamping torque and a fixed stop monitoring window can be programmed in the part program and set in the machine data. The function regulates the force to an adjustable value in the approach direction. In this case, it can be utilized simultaneously in several channels.

Benefits

- Technological function for spinning

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Not possible.

Multiple feedrates in one block

Function

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.

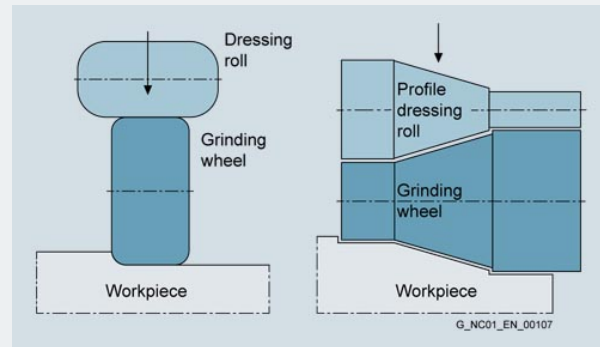
The input signals are combined in an input byte with a permanently assigned function. The retraction is initiated by an amount defined in advance within an IPO cycle.

Retraction movement or dwell time (e.g., sparking-out time during grinding) lead to deletion of the distance-to-go.

Benefits

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.

Continuous dressing (parallel dressing)



Function

With the continuous dressing function, the form of the grinding wheel can be dressed in parallel with the machining process. The grinding wheel compensation resulting from dressing the wheel takes immediate effect as tool length compensation. When the tool radius compensation is programmed to machine the contour and the tool radius changes because of the dressing of the grinding wheel, the CNC computes the dressing amount online as a true tool radius compensation. Grinding is characterized by the machining process (grinding), as well as sharpening of the grinding wheel (dressing).

Benefits

- Simultaneous grinding and dressing possible

Grinding Advanced

Option

Order code: S35

Article No.: 6FC5800-0BS35-0YB0

Function

The SINUMERIK Grinding Advanced option provides advanced functions for the grinding technology, e.g. the axis-parallel dressing and pre-profiling cycle.

Benefits

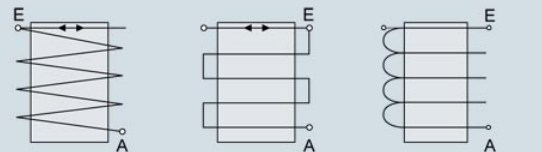
- Axis-parallel dressing
- Pre-profiling

Oscillation functions

Option

Order code: M34

Article No.: 6FC5800-0BM34-0YB0



Oscillating
motion with
continuous infeed

Oscillation stroke
with infeed in the
reversal points

Oscillation stroke 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

Function

With this function, an axis oscillates at the programmed feedrate between 2 reversal points.

Asynchronous oscillation across block boundaries

Several oscillating axes may be active. During the oscillation movement, other axes can interpolate at will. The oscillating 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 oscillating axis at the reversal point:

- A change of direction is initiated
- Without reaching the exact stop limit (soft reversal)
- After reaching the programmed position or
- after reaching the programmed position and expiration of the dwell time
- The following manipulations are possible:
 - Cancellation of the oscillation movement and infeed by deletion of distance-to-go
 - Modification of the reversal points via the part program, PLC, handwheel, or direction keys
 - Manipulation of the oscillating axis feedrate via part program, PLC, or override
 - Control of the oscillation movement via the PLC

The spindle can also perform an oscillation movement.

Benefits

- Utilization for grinding machines

Handling package	Option Order code: S31 Article No.: 6FC5800-0BS31-0YB0
	<p><u>Function</u></p> <p>For handling systems, the handling package offers a cost-effective solution:</p> <ul style="list-style-type: none"> • 3 additional axes • 3 additional machining channels • Transformation handling, Run MyCC /RCTRA • Synchronized actions stage 2 <p><u>Benefits</u></p> <ul style="list-style-type: none"> • Cost-effective option package for handling <p><u>Note</u></p> <p>Tool offsets and spindles are not possible.</p> <p><u>Requirement:</u></p> <ul style="list-style-type: none"> • Loadable compile cycle <p>Restricted functionality for export versions: Not possible.</p> <p>→ <i>Transformation handling Run MyCC /RCTRA (option M31)</i> → <i>Synchronized actions stage 2 (option M36)</i></p>

Technological functions with compressor Run MyCC /COTE

Option

Order code: N50

Article No.: 6FC5800-0BN50-0YB0

Note

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

Restricted functionality for export versions:
Not possible.

Electronic transfer CP

Option

Order code: M76

Article No.: 6FC5800-0BM76-0YB0

Function

In presses with multi-stage tools as well as in large-part transfer presses, a modern transfer system handles part transport. Positioning drives are controlled in step with the press's main motion.

The electronic transfer CP includes the options:

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

The combination of these options satisfies all requirements for highly dynamic and positionally accurate transfer controls.

Benefits

Control of the motion sequences in transfer systems, such as gripper rails or suction systems, etc., depending on a master value which corresponds to the current ram position of the press.

Restricted functionality for export versions:

The number of simultaneously traversing axes is restricted to 4.

→ *Position switching signals/cam controller (option M07)*

→ *Polynomial interpolation (option M18)*

→ *Generic coupling CP-Comfort (option M73)*

→ *Cross-mode actions ASUB and synchronized actions (option M43)*

→ *Synchronized actions stage 2 (option M36)*

→ *Pair of synchronized axes (gantry axes) (option M02)*

→ *I/O interfacing via PROFIBUS DP*

High-speed CNC inputs/outputs

Function

The high-speed CNC inputs/outputs function supports the input or output of 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:

- Multiple feedrate values per block (calipers function)
The function allows modification of the feedrate through external signals. 6 digital inputs can be combined with 6 different feedrate values in a CNC block. There is no feed interruption in this case. An additional input can be used to terminate the infeed (starting a dwell time), and another input can be used to start immediate retraction. Depending on the input, the retraction of the infeed axis (or axes) is initiated by a previously specified absolute value in the IPO cycle. The remaining distance-to-go is deleted.
- Multiple auxiliary functions in the block
Several auxiliary functions can be programmed in one CNC block. These functions are transferred to the PLC depending on a comparison operation or an external signal.
- Axis-specific deletion of the distance-to-go
The high-speed inputs affect a conditional stop and deletion of the distance-to-go for the path or positioning axes.
- Program branches
The high-speed inputs make program branches within a user program possible.
- Fast CNC start
Machining can be enabled conditionally in the part 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 function Position switching signals/cam controller.
- Programmable outputs
- Analog value output
- Safety-related signals such as safety door interlock

High-speed CNC inputs/outputsBenefits

- Rapid response to special events

→ *Position switching signals/cam controller (option M07)*

Synchronized actions

Function

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

Synchronized actions are executed in the interpolation cycle. Multiple actions can be processed in the same IPO cycle.

Synchronized actions without validity identifier are active non-modally only in the automatic mode. In the automatic mode, synchronized actions with validity identifier ID are modal in the subsequently programmed blocks.

Statically effective synchronized actions with the identifier IDS remain active in all CNC operating modes: cross-mode actions.

Possible actions in synchronized actions are e.g.:

- Output of auxiliary functions to PLC
- Writing and reading of main run variables
- Positioning of axes/spindles
- Activation of synchronous procedures such as: read-in disable, delete distance-to-go, end preprocessing stop
- Activation of technology cycles
- Online calculation of function values
- Online tool offsets
- Activation/deactivation of couplings/coupled motion
- Take measurements
- Enabling/disabling of synchronized actions

Synchronized actions

Benefits

- Response to machine states in all modes
- Optimization of the tool change
- Endlessly rotating rotary axes
- Fast, axis-specific deletion of the distance-to-go in response to input signals
- Manipulation of the read-in disable for the CNC block using external signals
- Monitoring of system variables such as velocity, power and torque
- Control of process variables such as velocity, speed and distance
- Machine manufacturer-defined reactions to certain states which can no longer be influenced by the user

Restricted functionality for export versions:

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

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

→ *Cross-mode actions ASUB and synchronized actions*

Motion-synchronous actions

Synchronized actions stage 2

Option

Order code: M36

Article No.: 6FC5800-0BM36-0YB0

Function

More than 24 synchronous actions can be active in the CNC block. As many as 255 parallel actions can be programmed in each channel. Technology cycles can be combined into programs using Synchronized actions stage 2, making it possible, for example, to start axis programs in the same IPO cycle by scanning digital inputs.

Benefits

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

Restricted functionality for export versions:

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

Positioning axes/spindles via synchronized actions

Function

Axes/spindles can be positioned depending on conditions (e.g. the actual values of other axes, high-speed inputs) with a special feedrate or speed to a specific setpoint via synchronized actions. Synchronized actions are executed in the interpolation cycle, are carried out in parallel with the actual workpiece machining procedure, and are not limited to CNC block boundaries.

These so-called command axes and command spindles can be started in the IPO cycle directly from the main program. The path to be traversed is either predefined or 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.

Benefits

Positioning of axes/spindles parallel to actual workpiece machining without being restricted to CNC block boundaries.

Analog value control in the interpolation cycle

Function

This function makes it possible to directly specify values from the part program via analog outputs.

The value specified by the NCK can be changed by the PLC before it is output to the hardware of a SIMATIC DP ET 200 analog module.

The hardware outputs are written in the interpolation cycle.

Benefits

- Implementation of machine-specific functions by the machine manufacturer

Requirement:

- Analog input

Motion-synchronous actions

Path velocity-dependent analog output (laser power control)

Option

Order code: M37

Article No.: 6FC5800-0BM37-0YB0

Function

Using the path-velocity-dependent analog output, the actual path velocity can be output in the interpolation cycle via a SIMATIC DP ET 200 analog module.

The function is programmed via synchronized actions.

Benefits

- Laser power control

High-speed laser switching signal, Run MyCC /HSLC

Option

Order code: M38

Article No.: 6FC5800-0BM38-0YB0

Function

For high-speed laser machining, the laser is switched on and off automatically and with a very high positional accuracy relating to the path. Under the prerequisite that all movements for which the laser must be switched off are made in rapid traverse mode G0, it is possible to logically combine the switching signal for the laser with the rising or falling edge of G0. The laser switching signal can also be coupled to an adjustable G1 feedrate threshold value, if required. To achieve the fastest possible responses, the switching on and off of the digital laser signal is controlled by a hardware timer with microsecond precision depending on the actual axis position. No programming measures are required for switching the laser itself on and off, as these procedures are directly linked to the programmed G functions.

It is only necessary to program a release with CC_FASTON (DIFF1, DIFF2) for the path section in which the laser is switched on or off. Together with this release, the two offset values, which can offset the switching on and off of the laser by a specific path differential in relation to the position setpoint are entered.

Benefits

- High-speed laser machining, e.g. of aperture plates

Requirement:

- Loadable compile cycle

Clearance control 1D in the IPO cycle

Function

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

Benefits

- Implementation of complex mathematical algorithms
- Know-how for these special algorithms remains protected

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

Option

Order code: M40

Article No.: 6FC5800-0BM40-0YB0

Function

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

Benefits

- Laser cutting of flat or spatially shaped metal sheets and of tubes
- Closed-loop force control with a force transducer, e.g. friction welding

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Clearance control can be applied to only one axis because 5-axis transformation is not available.

The number of interpolating axes is restricted to 4 without Run MyCC /CLC and to 3 with Run MyCC /CLC.

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

Option

Order code: M65

Article No.: 6FC5800-0BM65-0YB0

Function

For supplementing the Run MyCC /CLC function, Run MyCC /CLC-FD can be used to specify the spatial direction of the controlled motion with 3 simulated axes.

Benefits

It is possible to program the control direction independently of the tool orientation (laser beam orientation) specified by the 5-axis transformation.

Requirements:

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

Restricted functionality for export versions:

Clearance control can be applied to only one axis because 5-axis transformation is not available.

The number of interpolating axes is restricted to 4 without Run MyCC /CLC and to 3 with Run MyCC /CLC.

Motion-synchronous actions

Clearance control Run MyCC /CLCX

Option

Order code: N78

Article No.: 6FC5800-0BN78-0YB0

Function

The Run MyCC /CLCX clearance control maintains the technologically required clearance for special machining procedures, e.g. for laser cutting or water jet cutting.

Benefits

- More precise manufacturing

Requirement:

- Loadable compile cycle

Adaptive Control (evaluation of internal drive variables)

Option

Order code: M41

Article No.: 6FC5800-0BM41-0YB0

Product ID: MCS5100

Function

The evaluation of internal drive variables can be used to control a second process variable depending on a measured process variable – Adaptive Control. Evaluation of internal drive variables is a prerequisite for implementing adaptive control (AC).

The Adaptive Control can be parameterized within the part program as:

- Additive control
The programmed value (F word) is corrected by adding
- Multiplicative control
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

Benefits

- A second process variable can be controlled depending on the measured spindle current, e.g. path-specific or axis-specific feedrate. This permits, for example, the cutting volume to be kept constant when grinding, or faster covering of the grinding gap when scratching (first touch).
- Protecting machines and tools against overload
- Shorter machining times
- A superior surface quality of the workpieces can be achieved

Requirement:

- Loadable compile cycle

Asynchronous subprograms ASUB

Function

An asynchronous subprogram is a CNC program which can be started in response to an external event (e.g. a digital input) or from the PLC.

Inputs are allocated to subprograms and activated by programming SETINT.

Multiple asynchronous subprograms must be assigned different priorities PRIO so that they can be processed in a certain order.

Benefits

- Immediate cancellation of the CNC block currently being executed when the external event occurs
- The CNC program can be continued later at the point of interruption
- ASUBs can be disabled and enabled in the CNC program: DISABLE/ENABLE

Requirement:

- High-speed CNC inputs/outputs

→ *Interrupt routines with fast retraction from the contour (option M42)*

Motion-synchronous actions

Interrupt routines with fast retraction from the contour

Option

Order code: M42

Article No.: 6FC5800-0BM42-0YB0

Function

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 at the 8th program level and lead as high as the 12th program level.

An interrupt such as the switching of a high-speed CNC input can trigger a movement via the special subprogram, which allows fast retraction of the tool from the workpiece contour currently being machined.

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

Benefits

- Automated finished dimension detection, e.g. cylindrical grinding with external probe

Motion-synchronous actions

Cross-mode actions ASUB and synchronized actions

Option

Order code: M43

Article No.: 6FC5800-0BM43-0YB0

Function

Asynchronous subprograms ASUB make it possible to respond immediately to high-priority events not only during program execution, but in all CNC operating modes and program states. In the case of such an interrupt, it is possible to start an asynchronous subprogram in JOG.

Statically effective IDS synchronized actions, which are active in all controller operating modes, are also enabled.

Benefits

- For safer production, the grinding wheel can be maneuvered to a safe position, for example, to avoid collision

→ *Interrupt routines with fast retraction from the contour (option M42)*

Programming language

Function

The CNC programming language is based on DIN 66025. The new functions of the CNC high-level language also contain macro definitions, the combination of individual instructions.

Benefits

- Programming of part programs with and without high-level language elements

Working plane

Function

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

- Plane for the tool radius compensation
- Infeed direction for the tool length compensation depending on the type of tool
- Plane for the circle interpolation

When calling the tool path correction G41/G42, the working plane must be defined so that the CNC can correct the tool length and radius.

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

Benefits

- Define working plane for the desired contour

→ *Tool radius compensation*

Main program call from main program and subprogram

Function

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

Eleven subprogram levels (including 3 levels for interrupt routines) are possible in a main program.

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

Benefits

- Freedom during programming of part programs
- Programming of repeated processes in autonomous subprograms

Job list

Function

This can be used to create a job list (loading list) for every workpiece to be machined, for extended workpiece selection. 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 variables GUD, work offsets UFR, tool/magazine data TOA/TMA, setting data SEA, protection areas PRO, and sag/angularity from the hard disk of the PCU into the CNC work memory.
- Preparations for 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 work memory to the hard disk

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

Benefits

Simple selection of all programs and data required for machining of the workpiece.

Number of subprogram passes

Function

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

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

Benefits

- Structured program layout

Subprogram levels and interrupt routines

Function

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

When working with Siemens cycles, 3 levels are required. If such a cycle is to be called from a subprogram, the call can be made up to the 9th level.

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

Benefits

- Structured program layout

Skip blocks

Function

Skip blocks are identified by placing a "/" character in front of the block number. The instructions in the skip blocks are not executed and the program resumes with the next block that is not skipped.

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

Benefits

- CNC blocks that are not to be executed in every program run, e.g. execution of a trial program run, can simply be skipped

Polar coordinates

Function

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

Benefits

- Simplified programming

Metric/inch dimensions

Function

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

The following geometrical data can be entered directly and converted by the CNC into the other measuring system (examples):

- Position data X, Y, Z, etc.
- Interpolation parameters I, J, K and circle radius CR
- Thread pitch
- Programmable work offset (TRANS)
- Polar radius RP

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

Inverse-time feedrate

Function

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 on CNCs. 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.

Benefits

- Simplifies programming
- Improves general productivity

CNC high-level language

Function

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

System variables

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

User variables

If a program is to be used flexibly, variables and parameters are used instead of constant values. SINUMERIK CNCs give you the option of executing all CNC functions and addresses as variables. The names of the variables can be freely defined by the user. Read and write access protection can also be assigned using attributes. This means that part programs can be written in a clear and neutral fashion and then adapted to the machine as required, for example, free selection of axis and spindle address designations.

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

Local user variables LUD are available to the user for parameterizing part programs. These data can be redefined in every part 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.

CNC high-level language

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
- Power of 2 (squared), square root
- Absolute value
- Integer component, round to integer
- Exponential function, natural logarithm
- Offset, rotation, mirroring
- Scale modification

Comparison operations and logic combinations

Comparison operations with variables can be used to formulate jump conditions.

The comparison functions that can be used are:

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

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

Macro techniques

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

CNC high-level language

Control structures

The CNC normally processes the CNC blocks in the order in which they are programmed. Control structures allow the programmer to define additional alternatives and program loops as well as program jumps. The commands make structured programming possible, and make the programs much easier to read:

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

Benefits

- Generation of flexible part programs which can be variably adapted to the corresponding machine where required

Preprocessing memory, dynamic FIFO

Function

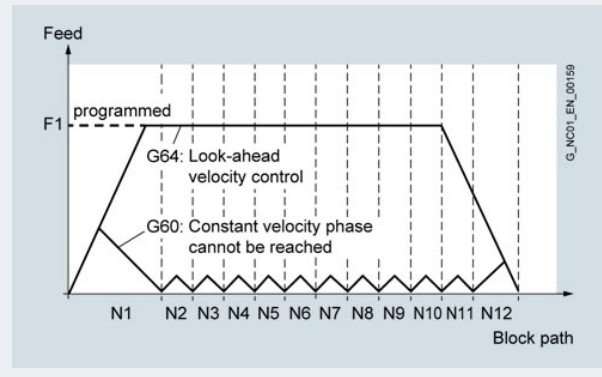
The traversing blocks are prepared prior to execution and stored in a preprocessing memory (FIFO = first in first out) of specifiable size. In contour sections that are machined at high velocities with short path lengths, blocks can be executed from this preprocessing memory at very high speed.

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.

Benefits

- Increased productivity due to fast execution

Look Ahead



Function

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

Benefits

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

→ *Continuous-path mode with programmable rounding clearance*

Frame concept

Function

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

For SINUMERIK CNCs, the frame in the CNC program transfers from one Cartesian coordinate system to another, and represents the spatial description of the workpiece coordinate system.

The following are possible:

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

The following instructions are used to program these options:

- TRANS programmable work offset
- ROT rotation in space or in a plane
- ROTs rotation referred to the solid angle projected into the planes
- SCALE scaling (scale factor)
- MIRROR mirroring
- TOFRAME frame according to tool orientation
- TOROT rotary component of programmed frame
- PAROT frame for workpiece rotation (table rotation)
- MEAFRAME frame calculation from 3 measuring points in space for measuring cycles

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

Additive frame instructions:

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

Frame concept

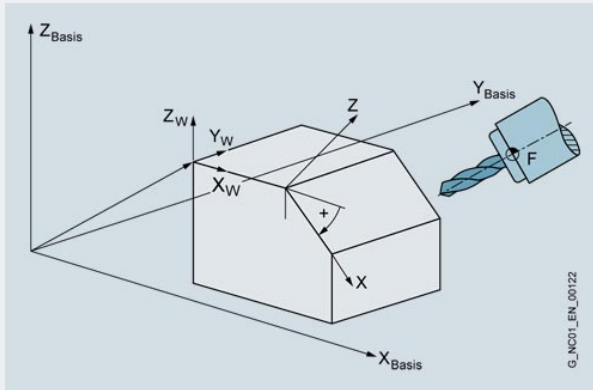
Benefits

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

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

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

Inclined-surface machining with frames



Function

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.

Benefits

- Easy machining of inclined surfaces

→ *Frame concept*

Axis/spindle interchange

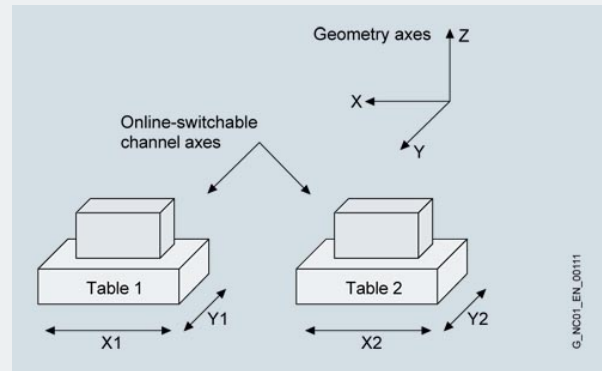
Function

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

Benefits

- Implementation of special machine tools

Geometry axes, switchable online in the part program



Function

In the CNC, geometry axes form axis groups per channel for the interpolation of path motions in space. Channel axes are assigned to geometry axes via machine data.

Benefits

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

Program preprocessing

Function

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

Benefits

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-synchronized actions or cutting cycles, execution times can be reduced by up to 1/3.

Online ISO dialect interpreter

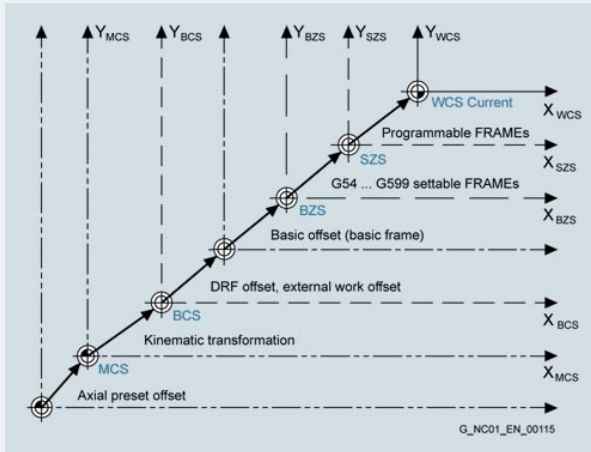
Function

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

Benefits

- Part programs from other manufacturers can be used

Work offsets



Function

Coordinate transformation during which the coordinate origin is moved.

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 formed by all the available physical machine axes.
- Basic coordinate system BCS
The basic coordinate system consists of 3 Cartesian axes (geometry axes), as well as other non-geometry axes (special axes).
- BCS and MCS are always in conformance when the BCS can be mapped to the MCS without kinematic transformation (e.g. TRANSMIT/face transformation, 5-axis transformation and max. 3 machine axes).
- Basic zero system BZS
DRF offsets, external work offsets and basic frames map the BCS on the BZS.
- Settable zero system SZS
An activated settable work offset G54 to G599 transfers the BZS to the SZS.
- Workpiece coordinate system WCS
The programmable frame determines the WCS representing the basis for programming.

Work offsets are therefore used to transform the machine zero point into the workpiece zero point in order to simplify programming.

The following work offsets are possible:

- Settable work offsets:
It is possible to enter up to 100 work offsets (G54 to G57, G505 to G599), offset coordinates, angles and scaling factors in order to call zero points program-wide for various fixtures or clamping operations, for example. The work offsets can be suppressed block-by-block.
- Programmable work offsets:
Work offsets can be programmed with TRANS (substitution function, basis G54 to G599) or ATRANS (additive function). This allows you, for example, to work with different work offsets for repetitive machining operations at different positions on the workpiece. G58/G59 make previously programmed work offsets axially replaceable.
- External work offsets:
Axis-related linear work offsets can also be activated via the PLC user software (function blocks) with assignment of system variable \$AA_ETRANS [axis].

Work offsetsBenefits

- Easier programming of workpieces

→ *Frame concept*

Scratching, determining work offset

Function

Manual traversing of the tool in the direction of the workpiece until the tool only just touches the workpiece.

A work offset can also be determined through scratching, taking into consideration an (active) tool and, where applicable, the base offset,

By moving the axis to the workpiece and entering the desired setpoint position (e.g. 0); the CNC then calculates the work offset.

Benefits

- Manual synchronization of workpiece zero point with the machine zero point

Preset

Function

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

Benefits

- Redefining the zero point in the machine coordinate system of the CNC

→ *Set actual value*

Set actual value

Function

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.

Benefits

- Defined setting of workpiece coordinates without resetting the reference point signal

Variables and arithmetic parameters

Function

Variables allow reaction to signals, e.g. measured values. If variables are used as a setpoint value, the same program can be used for different geometries.

The CNC uses 3 sorts of variables:

- User-defined variables
Variables defined by the user with name and type, e.g. arithmetic parameters
- Arithmetic parameters
Special, predefined arithmetic variables whose address is R plus a number. The predefined arithmetic variables are of the REAL type
- System variables
Variables provided by the CNC that can be processed in the program (write, read). System variables enable access to work offsets, tool offsets, actual values, measured axis values, CNC conditions, etc.

Variable type	Meaning	Value range
INT	Integers with sign	$\pm(2^{31} - 1)$
REAL	Real numbers (fractions with decimal point, LONG REAL according to IEEE)	$\pm(10^{-300} \dots 10^{+300})$
BOOL	Boolean values: TRUE (1) and FALSE (0)	1.0
CHAR	ASCII character specified by the code	0 ... 255
STRING	Character string, number of characters in [...], maximum of 200 characters	Sequence of values with 0 ... 255
AXIS	Axis names (axis addresses) only	All axis identifiers in the channel
FRAME	Geometrical parameters for offset, rotation, scaling, and mirroring	

Benefits

- Using variables instead of constant values allows the program to be developed flexibly.

Auxiliary function output

Function

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

The following functions can be transferred to the PLC:

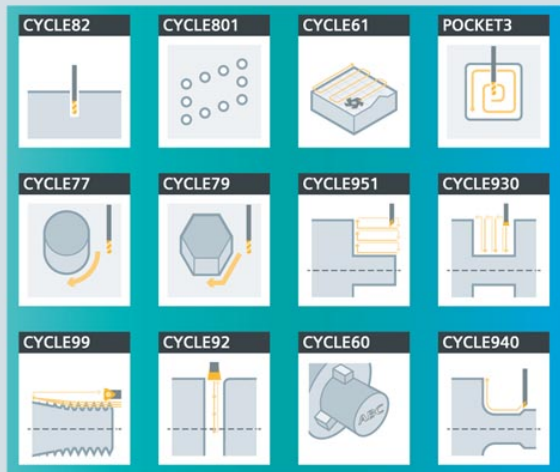
- Tool selection T
- Tool offset D/DL
- Feedrate F/FA
- Spindle speed S
- H functions
- M functions

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

Benefits

- Timely coordination between part program and PLC

Technology cycles



G_NC01_XX_00808

Technology cycles

Function

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

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

- Drilling technology
Drilling/centering, drilling/counterboring, deep-hole drilling, tapping with compensating chuck and rigid tapping, 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

Benefits

- Quick and simple programming of regularly recurring machining tasks, e.g. tapping of a thread or milling of a pocket

Cycle support

Function

The technology cycles for drilling, milling and turning and the measuring cycles are supported by cycle screens. The same input screens are also available to you for programming contours.

Users can also define their own softkeys, entry fields and screens using SINUMERIK Operate.

Benefits

- Optimum operation of technological cycles to suit the respective technology

→ *Use HMI applications Run MyHMI /3GL*

→ *Use HMI applications Run MyScreens*

programGUIDE

Function

The programGUIDE, with animated elements and cursor text, provides perfect support for integrating cycles into part programs.

Users can define their own softkeys, input fields and screens using Run MyScreens.

Benefits

- Help for fast programming of part programs
- Dynamic programming graphics
- Animated elements

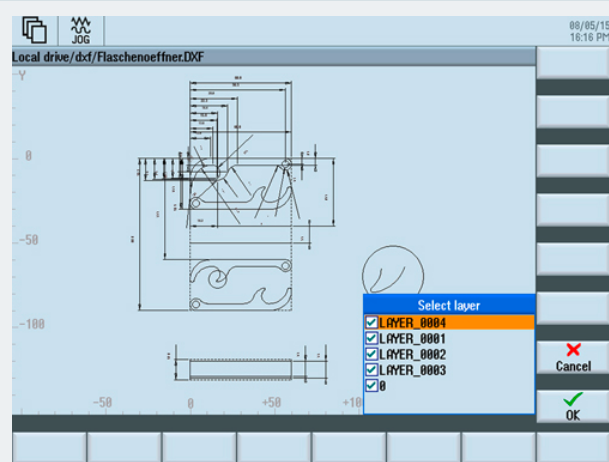
→ *Use HMI applications Run MyScreens (option P64)*

DXF Reader

Option

Article No.: 6FC5800-0BP56-0YB0

Product ID: MCS4130



Function

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

Benefits

- Fast transfer of contours and drilling patterns from the drawing to the part program
- Avoidance of faults and inaccuracies
- Higher machining quality on the workpiece

Manual machineFunction

We offer the Manual Machine function for beginners switching over from conventional machines, but also for experienced CNC machine operators who often only use individual machining steps.

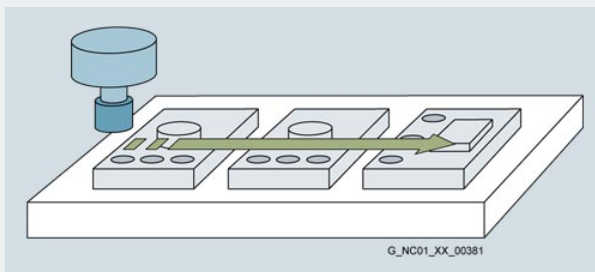
The MANUAL basic screen is displayed immediately after booting the machine offering the direct machining options without having to create a part program.

Benefits

A CNC machine can be operated in the same way as a conventional machine. Creation of a part program is not necessary.

→ *ShopTurn/ShopMill (option P17)*

Multiple clamping of identical/different workpieces



Multiple clamping of different workpieces

Function

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

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

Benefits

- Time-saving production of identical and different workpieces
- Idle times are reduced due to fewer tool changes because a tool performs as many machining operations as possible in all clampings before the next tool change is initiated.

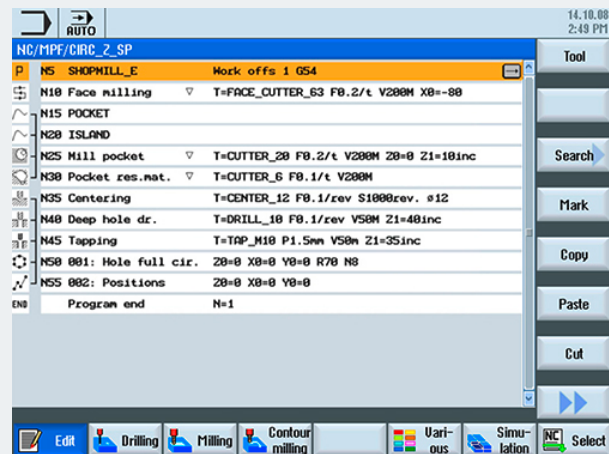
→ *ShopTurn/ShopMill (option P17)*

ShopTurn/ShopMill

Option

Order code: P17

Article No.: 6FC5800-0BP17-0YB0



Function

ShopTurn/ShopMill includes the functions:

- Machining step programming
- Multiple clamping of identical/different workpieces
- Manual machine

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 easy to read. Associated sequences are automatically interlinked and can be assigned any position patterns.

Benefits

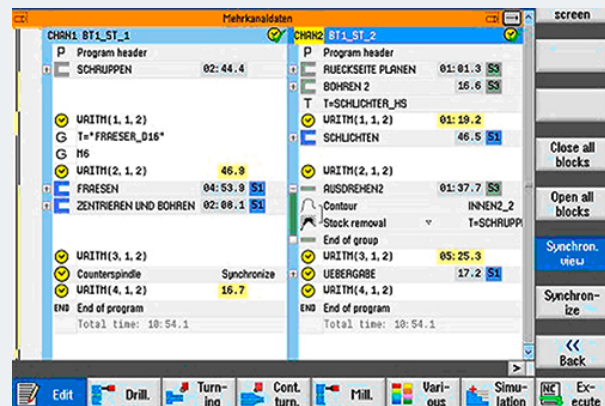
- Unique ease of programming
- The shortest programming times, even for highly demanding machining tasks

programSYNC

Option

Order code: P05

Article No.: 6FC5800-0BP05-0YB0



programSYNC: Dual editor

Function

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.

Benefits

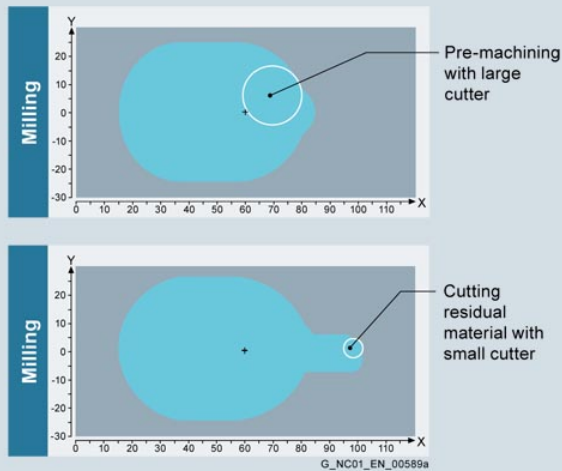
- Creation of time-optimized programs

Residual material detection and machining for contour pockets and stock removal

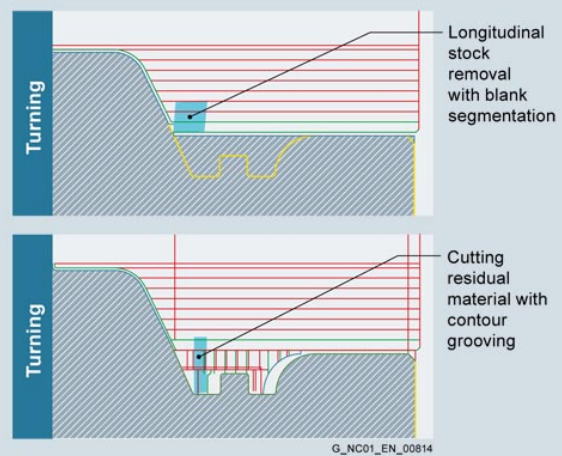
Option

Order code: P13

Article No.: 6FC5800-0BP13-0YB0



Residual material detection - milling



Residual material detection - turning

Function

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

Contour turning offers:

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

Contour milling offers:

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

Benefits

- Shorter machining times through the use of a large tool for the substantial part of the stock removal and a smaller tool for the remaining residual material
- Avoidance of non-cutting movements while achieving extremely simple programming
- Available for turning and milling

Access protection for cycles Lock MyCycles

Option

Order code: P54

Article No.: 6FC5800-0BP54-0YB0

Product ID: MCS1100

Function

With Lock MyCycles (cycle protection), cycles can be encrypted and then saved in the CNC where they are protected. Execution in the CNC is possible without restrictions, but it is not possible to view the cycle.

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

Benefits

- Protection for technological solutions
- Internal know-how can be protected

Lock MyPLC

Function

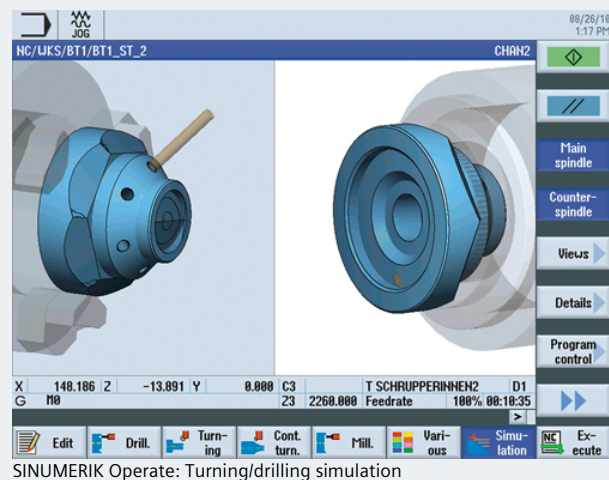
The PLC machine program can be comprehensively protected by the know-how protection functionality provided by SIMATIC STEP 7 software version 5.5 SP3.

This software protects technological knowledge against unauthorized access and modification. It prevents the reverse engineering of machines, thus safeguarding the investments made.

Benefits

- Internal know-how can be protected

Simulation SINUMERIK Operate



Function

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

The simulation data can be matched statically as required with the NCK environment (initialization data, macros, user data, tool data, machining cycles) or also dynamically when tool data or machining cycles are changed.

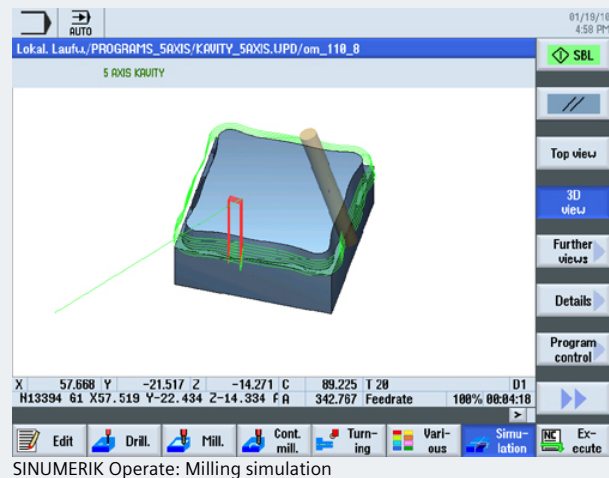
Machining simulations, with emphasis on the drilling, milling and turning technologies, can be performed in the workpiece coordinate system for certain machine kinematics on the user interface of the CNC.

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

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

The simulation can be extended to 3D representation by means of the 3D simulation 1 (finished part) function.

Simulation SINUMERIK Operate



Note

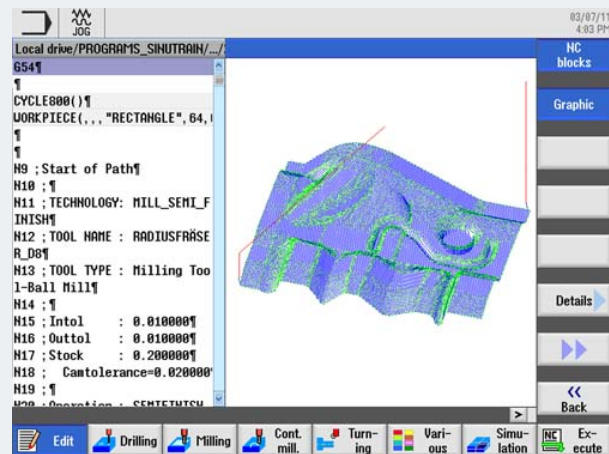
- Very fast simulation result, but consequently low dynamic resolution
- Fine recording is too slow for complex displays/fine recording fails to run for very complex parts
- Quickview for mold-making part programs
- Simulation of special kinematics not available when compile cycles are used
- Blank CAD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

Benefits

- Program control without machining

→ 3D simulation 1 (finished part) (option P25)

Quickview for mold-making programs

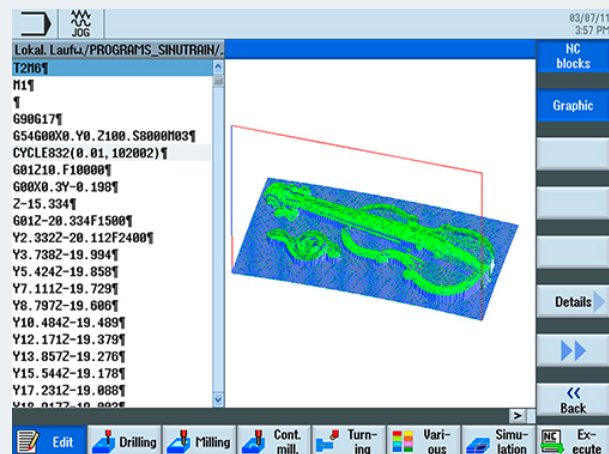


Function

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

Benefits

- Additional reliability when using mold-making programs



3D simulation 1 (finished part)

Option

Order code: P25

Article No.: 6FC5800-0BP25-0YB0

Function

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

- Particularly realistic simulation through representation of the tool
- Also available in automatic mode
- Reliability:
 - 3-plane view and solid model of the finished part, with zoom to details and free rotation of the viewing angle
- Support:
 - Simulation speed controllable by override; single block operation and start / stop available at any time
- Check:
 - Automatic calculation of the machining time

Parallel simulation (background simulation) is possible in conjunction with the NCU 720 and NCU 730, i.e. simulating a part program while another part program is being simultaneously machined.

Benefits

- 3D program control without machining
- Available for turning and milling

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

Option

Order code: P22

Article No.: 6FC5800-0BP22-0YB0

Function

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

Note

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

Benefits

- Machining can also be monitored in the complex machine room
- Check of the programming result (dry run feedrate)

→ *Simulation SINUMERIK Operate*

Simulation axis/spindle

Function

The speed control loop of an axis can be simulated for test purposes. The axis traverses with a following error, similar to a real axis. Setpoint output and actual values are not available.

For simulated axes, the program is executed exactly the same as for real axes, i.e. no functional restrictions apply to simulated axes.

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

Benefits

- Program test without having to move the axes or spindles

→ *Axis/spindle*

→ *Positioning axis/auxiliary spindle*

→ *Virtual axis*

→ *PLC axes*

CNC operating modes

Function

Three control modes can be selected in the Machine operating area:

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

Benefits

- Simple and clear operation of the CNC

Reference point approach

Function

When using a machine axis in the program-controlled mode, it is important to ensure that the actual values supplied by the measuring system agree with the machine coordinate values. Reference point approach (limit switches) is performed separately for each axis at a defined velocity either using the direction keys, in a sequence that can be defined in the machine data, or automatically via program command G74.

If length measuring systems with distance-coded reference marks are used, reference point approach is shorter, as it is necessary to approach only the nearest reference mark. Reference point approach of an axis with absolute encoders is carried out automatically when the control is switched on (without axis motion), if the corresponding axis is recognized as being calibrated.

Benefits

- Synchronization of the machine axes with the measuring system

Repos

Function

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

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

Benefits

Repositioning to the contour:

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

Execution from external storage EES

Option

Order code: P75

Article No.: 6FC5800-0BP75-0YB0

Product ID: MCS5110

Function

With Execution from External Storage (EES), it is possible to execute part programs from external memories without EXTCALL.

- Part program is executed directly from the external memory
- No restrictions regarding jump commands for a subprogram call (forward, backward jumps or loops)
- Subprogram call by program name, part program can be transferred to another machine 1:1
- The available memory on the machine is basically expanded to the size of the external memory
- Error correction possible without CNC reset
- Program access and change of several machines on a central part program

Benefits

- Productive by editing the part programs without CNC reset and prevention of errors in the syntax during the subprogram call.
- User-friendly due to an easy subprogram call and unlimited program jumps
- The size of the part program memory is virtually unlimited. The size of the memory available on the machine can be expanded economically with external media.
- A machine with several NCUs can use a common part program memory.

DRF offset (differential resolver function)

Function

The DRF offset generates an additional incremental work offset in the AUTO control mode via the electronic handwheel.

Benefits

- Compensate for tool wear within a programmed block

→ *Handwheel override*

Block search

Function

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

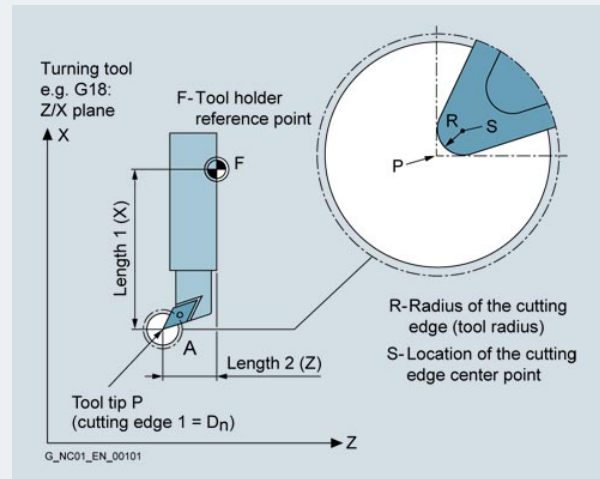
Search variants:

- With calculation at the contour
During the block search, the same calculations are executed as during 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 the block end point
This function allows approach to a target position (e.g. tool change position). All calculations are also executed here as during normal program operation. The end point of the target block or the next programmed position is approached based on the interpolation valid in the target block.
- Without calculation
This method enables a high-speed search in the main program. No calculations are carried out during the search. The internal control values remain the same as before the block search.
- External block search without calculation
In the menus "Search position" and "Search pointer", the softkey "External without calc." can be used to start an accelerated block search for programs which are executed by an external device (local hard disk or network drive).
- Specify the search target by:
 - Directly positioning the cursor on the target block
 - Specifying a block number, a jump label, any character string, a program name, or a line number

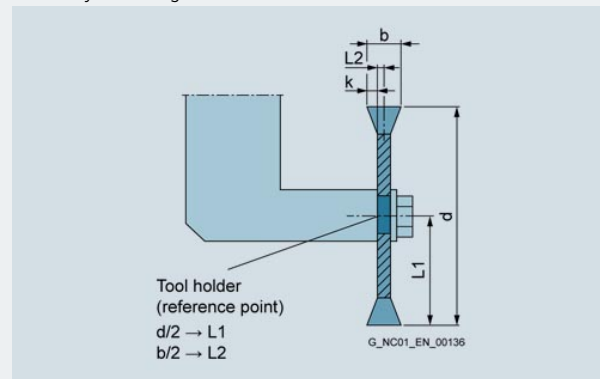
Benefits

- Testing of part programs or restart following a cancellation in machining

Tool types



Geometry of turning tool



Geometry of slotting saw

Function

The tool type determines the geometry specifications required for the tool offset memory, and how they are to be used. Entries are made for the relevant tool type in tool parameter DP. The CNC combines these individual components to produce a result variable, e.g., total length, total radius. The relevant overall dimension becomes operative 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. Input screen forms can be used to save all of the tool offsets.

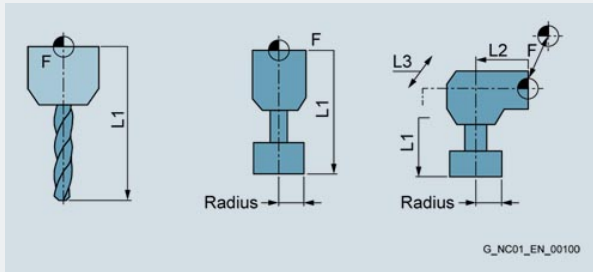
Tool types, parameterizable

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

Benefits

The tool types define how the geometry specifications are computed.

Tool offsets



Function

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

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

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

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

When a workpiece is machined, the tool paths, depending on the relevant tool geometry, are controlled so that the programmed contour can be produced with every tool used. The tool data are entered separately in the tool table of the CNC, and only the required tool with its offset data is called in the program.

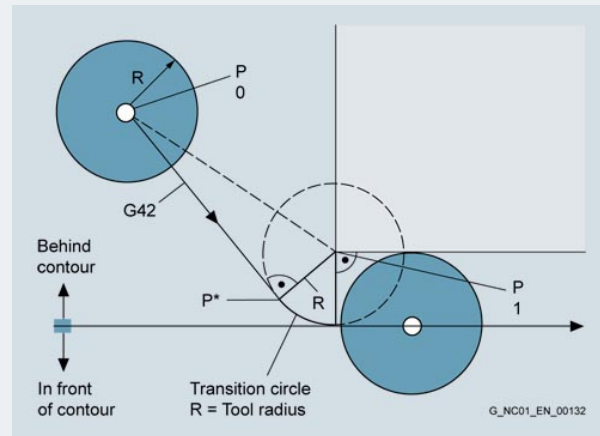
During program execution, the CNC retrieves the required offset data from the tool files and automatically corrects the tool path for various tools.

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

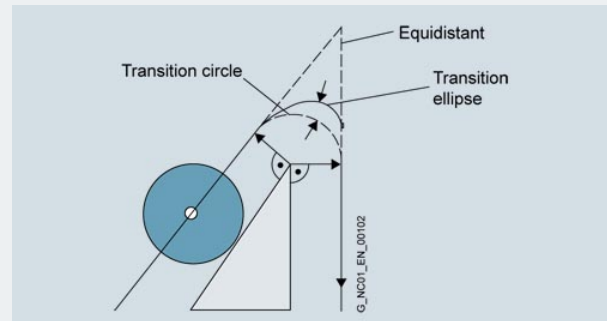
Benefits

- Easier programming, as the tools are only taken into consideration upon execution

Tool radius compensation



CONT for selection behind the contour



Traveling around the outside corners with transition circle/transition ellipse

Function

When tool radius compensation is active, the CNC automatically calculates the equidistant tool paths for different tools. To do so, it requires the tool number T, the tool offset number D (with cutting edge number), the machining direction G41/G42, and the relevant working plane G17 to G19. The path is corrected in the programmed level depending on the selected tool radius. The approach and retract paths can for example be adapted to the required contour profile or to blank forms: NORM
The tool travels in a straight line directly to the contour and is positioned perpendicular to the path tangent at the starting point: CONT

If the starting point is behind the contour, the corner point P1 of the contour is bypassed. If the starting point is in front of the contour, the normal position at the starting point P1 is approached in the same way as with NORM. In the part program it is also possible to select the strategy with which the outside corners of the contour are to be bypassed:

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

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

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

The CNC can also automatically insert a circle or a straight line in the block with the tool radius compensation when no intersection with the previous block is possible. Compensation mode with the "Tool radius correction" may only be interrupted by a certain number of successive blocks or M functions which do not contain motion commands or positional data in the compensation level. The number of successive blocks or M commands can be set using machine data (standard 3, max. 5).

Benefits

- Programming of contours without knowledge of the tool radius

→ 3D tool radius compensation

Intermediate blocks for tool radius compensation

Function

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

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

Benefits

Freedom to insert elements of the CNC high-level language during programming of part programs.

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Not possible.

→ *Tool radius compensation*

3D tool radius compensation

Option

Order code: M48

Article No.: 6FC5800-0BM48-0YB0

Function

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

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

Benefits

- Machining of inclined surfaces

Tool offsets, grinding-specific

Function

Grinding-specific tool offsets are available for grinding technology:

- Minimum wheel radius
- Maximum speed
- Maximum surface speed

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

Tool type

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

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

The speed setpoint is monitored cyclically against 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.

Benefits

- Grinding-specific tool offset data for the grinding technology

→ *Grinding wheel surface speed, constant*

Tool change via T number

Function

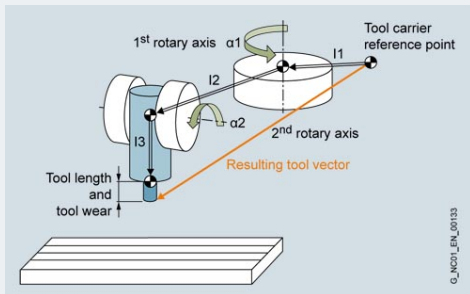
In chain, rotary-plate and box magazines, a tool change normally takes place in 2 stages: A T command locates the tool in the magazine, and an M command inserts it in the spindle.

In turret magazines on turning machines, the T command carries out the entire tool change, that is, locates and inserts the tool. The tool change mode can be set using machine data.

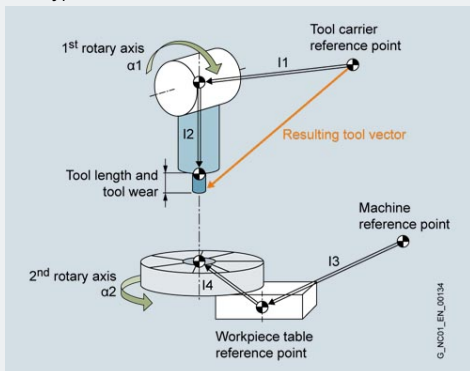
Benefits

- Adapting the tool change to the machine

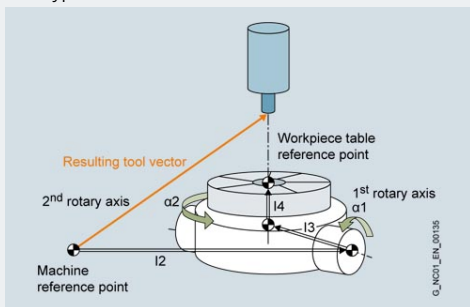
Tool carrier with orientation capability



Tool type T



Tool type M



Tool type P

Function

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

The tool carrier with orientation capability function enables 2½D/3D machining with fixed spatial orientation of the tool/workpiece table. Vectors I1 to I4 represent the geometrical dimensions of the machine. The rotary axes need not move in parallel to the Cartesian axes, but instead can be inclined at any angle (e.g., cardan milling head with 45° inclination). The angles α_1 and α_2 can be either specified or computed from the active frame and assigned to the tool carrier with orientation capability or to the workpiece table.

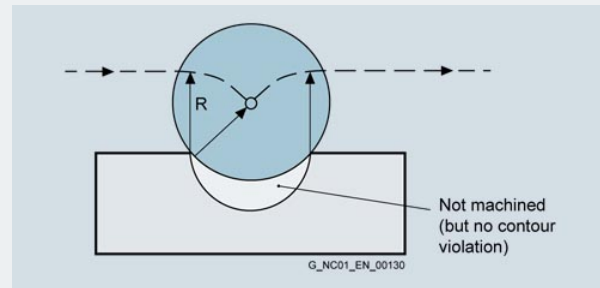
The following kinematics can be configured flexibly:

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

Benefits

Ensuring that any position of the tool/TCP can be reached in the machining space.

Look-ahead detection of contour violations



Behavior when tool radius > circle radius

Function

With collision detection CDON on and active tool radius compensation, the CNC monitors tool paths through look-ahead contour calculation. Potential collisions are promptly identified in this way and actively prevented by the CNC. The CNC detects the following critical machining situations, e.g., when the tool radius is too large, and effects compensation through tool path modification.

- Bottleneck detection
Because the tool radius is too large to produce a narrow inside contour, the bottleneck is bypassed and an alarm 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.

Benefits

- Automatic avoidance of potential collisions in good time

Grinding wheel surface speed, constant

Function

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

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

Benefits

- Constant grinding wheel circumferential speed

Tool orientation interpolation

Function

With active kinematic 5/6-axis transformation, the orientation behavior of the tool is programmed with execution of the part program for the Tool Center Point (TCP).

The orientation behavior (orientation interpolation) can be specified, for example, linearly (ORIAXES), in the plane of a large circle (ORIVECT/ ORIPLANE), on a taper surface area (ORICONCW/ORICONCCW) as a free vector interpolation of the tool (ORICURVE).

Benefits

- Optimized orientation of the tool for machining improves quality of the workpiece

Requirements:

- Milling technology package SINUMERIK MDynamics 5 axes, or
- Machining package 5 axes

Restricted functionality for export versions:

Not possible.

→ *Machining package 5 axes (option M30)*

→ *Milling technology package SINUMERIK MDynamics 5 axes (option S33)*

Orientation offset static/dynamic

Option

Order code: S14

Article No.: 6FC5800-0BS14-0YB0

Function

The static/dynamic orientation offset offers the possibility of superimposing an offset of tool orientation in real time for the programmed tool orientation and may be used independently of the applied 5-axis machine kinematics. The superimposition then takes effect for the programmed orientation in each block and may also be changed from block to block as needed.

The orientation offset can be activated in different ways:

- In the part program by describing the program variables for offset of the orientation.
- In synchronized actions by describing the system variables for offset of the orientation.
- Activation of a dynamic tool orientation offset in accordance with path velocity, without having to define the synchronized actions in this regard.

Note

The specific function Orientation offset static/dynamic is not intended for general use.

Benefits

- Known deviations in the tool orientation can be corrected
- A direct increase in manufacturing precision for 5-axis machining processes is possible, as long as the cause of the deviation can be established and described in adequate detail.
- Application area, e.g. water jet machining

Tool management

Function

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 turret magazines, chain magazines or box magazines.

The most important functions of tool management are:

- Tool selection across all magazines and turrets for active tools and spare tools
- Determining a suitable empty location depending on tool size and location type
- Tool-dependent location coding: fixed and variable
- Initiation of tool changes with T or M command
- Axis movements during a tool change with automatic synchronization at the next D number
- Quantity, tool life and wear monitoring with prewarning limit monitoring function
- Support when using multi-tools

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

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

Benefits

The tool management allows fast tool changes. The tool management prevents scrap by monitoring the tool operation time and machine downtimes taking replacement tools into consideration.

Tool management with more than 4 magazines

Option

Order code: M88

Article No.: 6FC5800-0BM88-0YB0

Function

4 magazines are included in the basic version of SINUMERIK ONE:

- Magazine
- Spindle
- Tool buffer

Benefits

- The number of magazines is increased

Monitoring for maximum tool speed/acceleration

Option

Order code: S08

Article No.: 6FC5800-0BS08-0YB0

Function

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

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

Benefits

- Tool protection

Identify tool demand**Option**

Order code: M77

Article No.: 6FC5800-0BM77-0YB0

Function

The tools needed for machining are recorded in a list during execution or during simulation of a part program. If the machining task is repeated it is possible to check if all the necessary tools are available. Based on the list it is possible to load and unload one or more tools simultaneously. In addition, new tools can be created based on the data recorded in the list.

Benefits

- Quick and simple test if all tools are loaded before starting the program
- Simplified loading and unloading of tools

Tool identification systems

Function

The tool load and unload dialog of the Siemens tool management system provides a link to an automatic tool identification system, e.g. SIMATIC Ident. This means that you can automatically read and write to the tool code carrier, instead of having to manually enter tool data.

When unloading, the data record for the tool is saved; when loading, it is read via the code carrier and entered in the tool management system. In the interim, the tool data can be re-edited as during tool selection from the tool catalog, e.g. offset data.

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. When unloading, the dialog data is converted back into code carrier data using the description file.

Benefits

Manual entry of the tool data can be substituted with automatic reading and writing of the tool code carrier. This eliminates the need for manual entries which are susceptible to errors.

→ [Tool Ident Connection](#)

Tool Ident Connection

Option

Article No.: 6FC5800-0BP52-0YB0

Product ID: MCS5130

Function

The Tool Ident Connection function allows tool identification systems such as SIMATIC Ident to be linked to SINUMERIK CNCs for operation in conjunction with the standard tool management system, and also supports the transfer of tool data.

Benefits

Manual entry of the tool data can be substituted with automatic reading and writing of the tool code carrier. This eliminates the need for manual entries which are susceptible to errors.

→ *Tool identification systems*

Data exchange between machining channels

Function

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.

Benefits

- Implementation of complex machining tasks on machines with multiple machining channels

→ *CNC high-level language*

I/O interfacing via PROFIBUS DP and PROFINET

Function

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

Benefits

- Application of I/O devices from many diverse manufacturers

→ *PROFINET*

PROFINET

Function

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.

Versions of PROFINET:

- PROFINET IO (input output) for controlling actuators, sensors and drives using one or several central controllers

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

PROFINET includes:

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

Benefits

- Openness offers flexibility for future innovations
- Greater freedom to implement tailored machine and plant concepts
- Unequaled efficiency allows optimum utilization of resources
- Unique performance ensures high precision and product quality
- Sustained increase in productivity

→ *PROFINET IO*

→ www.siemens.com/profinet

PROFINET IO

Function

PROFINET IO is used to control sensors and actuators using a central controller in production engineering.

A PROFINET IO system is assembled from the following devices:

- An IO controller is a controller, typically a PLC, CNC, robot control or motion controller, that controls the automation task. An IO controller is a master as compared to PROFIBUS.
- An IO device is a distributed field device which is linked via PROFINET IO. It is controlled by an IO controller. An IO device can consist of several modules and submodules. All data to be exchanged are assigned slots and subslots for the purpose of addressing. These are defined in the General Station Description (GSD) file. ET 200 distributed I/O or a SINAMICS drive are examples of PROFINET IO devices. When compared to PROFIBUS, an IO device is a slave.
- An IO Supervisor is typically a programming device, a PC or an HMI device for commissioning or diagnostics. It features an engineering tool, which can be used to parameterize and diagnose individual IO devices. When compared to PROFINET, this would be a class 2 master in terms of function.

PROFINET IO provides protocol definitions for the following functions:

- Cyclic transmission of IO data
- Acyclic transmission of alarms which require acknowledgement
- Acyclic transmission of data (parameters, detailed diagnostic information, commissioning data, I&M data)

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

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

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

Benefits

- Openness offers flexibility for future innovations
- Greater freedom to implement tailored machine and plant concepts

Communication and data management

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

Option

Order code: N61

Article No.: 6FC5800-0BN61-0YB0

Function

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

Benefits

- Connection of Renishaw's SPRINT measuring system

Requirement:

- Loadable compile cycle

Communication and data management

Master computer connection Access MyMachine /OPC UA

Option

Order code: P67

Article No.: 6FC5800-0BP67-0YB0

Function

The Access MyMachine /OPC UA function enables the communication channel for the OPC UA server on the SINUMERIK PCU, SIMATIC IPC, or the SINUMERIK NCU. The OPC UA Data Access method permits the reading and writing of CNC and PLC variables.

OPC Unified Architecture OPC UA is a standardized, industrial communication protocol for the access to control data, e.g. by control systems.

Benefits

- Commercial success is increased with support of I4.0 communication standards
- Recognition as a manufacturer of innovative machine tools

→ www.opcfoundation.org

Communication and data management

Access MyMachine /OPC UA for PLC

Option

Order code: P82

Article No.: 6FC5800-0BP82-0YB0

Function

The Access MyMachine /OPC UA for PLC option activates the OPC UA Server and the Client of the integrated S7-1500 PLC. This enables direct communication with the S7-1500 via the standardized OPC UA protocol.

The OPC UA Server of the PLC enables high-performance access to the data of the PLC and also offers the possibility to implement own methods in the PLC code, which can be called via an OPC UA Client.

The OPC UA Client enables the PLC to establish an active communication via OPC UA.

Benefits

- Access to PLC data via OPC UA
- Implementation of own OPC UA methods enabled
- Simple controller-controller communication between two SINUMERIK PLCs
- Access to consistent data with methods (OPC UA Server) or active writing via OPC UA Server

Requirement:

- Loadable compile cycle

User interface

Function

The operator interface is clearly arranged with a modern and transparent layout. The numerous functions ensure a high level of user friendliness. This includes the latest touch and gesture control, side screen, and working with animated elements.

Benefits

- The animated elements permit even easier and more user-friendly operation.
- All information is available to the user for every operating situation at a glance, and it is possible to scroll through the content both horizontally and vertically on the side screen, which can be hidden and shown as required.
- The specific use of Windows-type technology permits simple and user-friendly machine operation.

Animated elements

Function

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

Benefits

The operator is provided support to carry out his work easily and comfortably, thus allowing him to achieve his desired results more quickly.

Tooltips

Function

A tooltip is a small pop-up window in application programs. It provides a description for an element of the graphic user interface.

Benefits

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

SINUMERIK Operate Display Manager

Option

Order code: P81

Article No.: 6FC5800-0BP81-0YB0

Function

The SINUMERIK Operate Display Manager offers the following features:

- Predefined display layout through standardized Siemens design
- Integration of 3rd-party applications in the operator interface, with possible simultaneous representation
- Standardized functions, e.g. PDF viewer and keyboard
- Display Manager functionality completed in the interfaces in SINUMERIK Create MyHMI /3GL

Benefits

- Higher flexibility for the machine manufacturer when configuring operator interfaces
- Optimal utilization of the display area with high resolutions
- Fast solution for the representation of several applications on one display through predefined division into different screens in Siemens design
- Creation of individual contents by the machine manufacturer using SINUMERIK Create MyHMI /3GL

Requirements:

- SINUMERIK Operate /PCU or SINUMERIK Operate /PC
- Display with a resolution of 1920 × 1080 and a display diagonal of 22 inches or larger

→ *SINUMERIK Operate /PCU (option P88)*

→ *SINUMERIK Operate /PC (option P87)*

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

→ *Use HMI applications Run MyHMI /3GL (option P60)*

→ *www.siemens.com/sinumerik-operate*

SINUMERIK Operate /NCU

Option

Order code: S00

Article No.: 6FC5800-0BS00-0YB0

Function

SINUMERIK Operate /NCU permits the activation of SINUMERIK Operate in the CNC software for the NCU with SINUMERIK TCU.

- Optimized for multi-touch operation
- Paperless manufacturing, e.g. due to the simple data transfer using the integrated DXF reader
- Maximum convenience through interactive input using animated elements
- Data handling just like on a PC with program manager
- Graphic support in the intelligent JOG mode
- Very simple tool and workpiece measurement in JOG mode
- Clearly structured tool management
- Process safety through CNC simulation

Benefits

- High efficient usability on the machine tool

Restricted functionality for export versions:

Not possible.

→ www.siemens.com/sinumerik-operate

SINUMERIK Operate /universal client

Option

Order code: S87

Article No.: 6FC5800-0BS87-0YB0

Function

The SINUMERIK Operate /universal client options permits activation of SINUMERIK Operate in the CNC software for a connected universal Thin Client.

- Optimized for multi-touch operation
- Paperless manufacturing, e.g. due to the simple data transfer using the integrated DXF reader
- Maximum convenience through interactive input using animated elements
- Data handling just like on a PC with program manager
- Graphic support in the intelligent JOG mode
- Very simple tool and workpiece measurement in JOG mode
- Clearly structured tool management
- Process safety through CNC simulation

Benefits

- High efficient usability on the machine tool

→ www.siemens.com/sinumerik-operate

SINUMERIK Operate /PCU

Option

Order code: P88

Article No.: 6FC5800-0BP88-0YB0

Function

SINUMERIK Operate /PCU permits activation of SINUMERIK Operate in the CNC software for a connected PCU. SINUMERIK Operate makes machine tool operation extremely easy. SINUMERIK Operate therefore sets the standard for the efficient operation of machine tools.

Benefits

SINUMERIK Operate /PCU provides added performance for demanding operating and programming tasks, wherever a SINUMERIK PCU or a SIMATIC IPC is used for SINUMERIK.

→ www.siemens.com/sinumerik-operate

SINUMERIK PCU base software /IPC**Option**

Order code: P86

Article No.: 6FC5800-0BP86-0YB0

Functions:

- SINUMERIK PCU base software /IPC provides SINUMERIK PCU functionality on defined SIMATIC IPCs
- Integrated solution for backup of the SIMATIC IPCs for SINUMERIK
- TCU support for connection of SINUMERIK HT8, SINUMERIK OP and SIMATIC ITC, including focus switchover of up to 4 active panels
- Integrated mechanism for software installation

Benefits

- Connection of SINUMERIK HT8, SINUMERIK OP, and SIMATIC ITC
- SINUMERIK Operate /PCU can be used on SIMATIC IPCs
- No separate solution necessary for backup of the SIMATIC IPCs for SINUMERIK

SINUMERIK Operate /PC**Option**

Order code: P87

Article No.: 6FC5800-0BP87-0YB0

Function

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

- Optimized for multi-touch operation
- Paperless manufacturing, e.g. due to the simple data transfer using the integrated DXF reader
- Maximum convenience through interactive input using animated elements
- Data handling just like on a PC with Program Manager
- Graphic support in the intelligent JOG mode
- Very simple tool and workpiece measurement in JOG mode
- Clearly structured tool management
- Process safety through CNC simulation

Benefits

- High efficient usability on the machine tool

→ www.siemens.com/sinumerik-operate

Operator panel management

Functions:

- Operator panels with SINUMERIK TCU (thin client unit) allow the operator panel and SINUMERIK NCU/PCU or SIMATIC IPCs approved for use to be installed in separate locations
- Up to 4 operator panels with TCU can be connected to SINUMERIK NCU/PCU or SIMATIC IPCs approved for use
- The user interface is copied from one control device onto multiple operator panels

✓ = possible

– = not possible

Operator panel management

2/4 TCUs with interlocking of simultaneous operation by Veto mode and PLC
(4 × T : 1 × M)

TCU suppression, with more than 2/4 TCUs ($n \times T : 1 \times M$)

External HMIs ($n \times M : n \times N$)

One external HMI which can be switched over via several NCUs ($1 \times M : n \times N$)

One internal and one external HMI connected simultaneously to one NCU

NCU CNC SW	SINUMERIK PCU or SIMATIC IPC for SINUMERIK SINUMERIK Operate
✓ (2/4)	✓ (4)
✓	✓
–	✓ (1 × M : 4 × N)
–	✓ (4 × N)
✓ Tool management	✓ Tool management
–	–
✓	✓

Display

Down zoom

Dynamic resolution switchover

Operator panel management

✓ = possible
– = not possible

NCU
CNC SW

SINUMERIK PCU or
SIMATIC IPC for
SINUMERIK
SINUMERIK Operate

Activate/deactivate MCP

MCP PROFIBUS

–

–

MCP IE (IE mode)

✓

✓

MCP PN (PN mode)

–

–

One or several TCUs which can be switched over via several NCUs and PCUs (1 × T : n × M)

Using 2-key operation

✓

✓

Using the channel menu

✓

✓

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

T = number of TCUs

M = number of HMIs

N = number of NCUs

Benefits

- Low-oscillation mounting of the SINUMERIK PCU or the SIMATIC IPC in the control cabinet
- Effective operation of larger machines using up to 5 identical operator panels
- Signal transmission between SINUMERIK PCU, SIMATIC IPC and operator panel via Industrial Ethernet
- Operation at the active operator panel with the option to enable the passive operator panel on request
- Combined operation of operator panel with TCU and an operator panel front directly on the SINUMERIK PCU
- Distances between SINUMERIK PCU, SIMATIC IPC and operator panel of up to 100 m (maximum distance between two network nodes)

Screen blanking

Function

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.

Benefits

The service life of the screens is increased.

Electronic handwheels

Function

Using electronic handwheels, it is possible to move selected axes simultaneously in manual mode. The weighting of the handwheel graduations is dependent on the increment-size weighting. If coordinate offset or coordinate rotation is selected, it is also possible to move the axes manually in the transformed workpiece coordinate system.

The maximum input frequency of the handwheel inputs is 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 along a contour.

Once the Contour handwheel function has been activated, the handwheel has a velocity-generating effect in AUTO and MDI CNC operating modes, i.e. a feedrate specified via the part 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.

Benefits

- Manual traversing of axes

Electronic Key System EKS

Option

Order code: P53

Article No.: 6FC5800-0BP53-0YB0

Function

Support of the Electronic Key System (EKS) in SINUMERIK MPPs (machine push button panels).

Benefits

- Integration of the Electronic Key System EKS in SINUMERIK MPPs
- Secure and controlled access
- Increased product quality due to controlled production
- Increased operational reliability

Plain text display of user variables

Function

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

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

Definition: DEF INT NUMBER – Display: NUMBER

Definition: DEF REAL DEPTH – Display: DEPTH

Benefits

- Operator recognizes significance of the variables immediately

→ *CNC high-level language*

Multi-channel display

Function

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

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

Benefits

- Simultaneous display of multiple channels

Workpiece-related actual value system

Function

The workpiece-related actual value system of the SINUMERIK CNC enables the user to do the following:

- After the CNC has powered up, start machining in a workpiece coordinate system defined via machine data in JOG and AUTO CNC operating modes without any additional operator actions
- Retain the valid settings relating to the active plane, settable frames (G54 ... G57), kinematic transformations, and active tool offset at the end of the part program for use in the next part program
- Toggle between the workpiece coordinate system WCS and the machine coordinate system MCS
- Change the workpiece coordinate system, e.g. by changing the settable frames or the tool offset.

Benefits

- Displayed actual positions correspond to the values on the workpiece drawing

Access protection

Function

- Access to programs, data and functions is protected in a user-oriented hierarchical system of 7 access levels.
- Protection level 1 has the highest, protection level 7 the lowest access rights. A higher protection level automatically includes all protection levels below it.
- Access rights for the protection levels 1 to 3 are set by Siemens by default. Entry of a password will overwrite the keyswitch setting.

Protection level	Type	User	Access to (examples)
1	Password	Machine manufacturer: Development	Defined functions, programs and data: e.g. entering options
2	Password	Machine manufacturer: Commissioning engineer	Defined functions, programs and data: e.g. bulk of machine data
3	Password	End user: Service	Assigned functions, programs and data
4	Red key Switch position 3	End user: Programmer, machine setter	Less than protection level 0 to 3: specified by machine manufacturer or end user
5	Green key Switch position 2	End user: Qualified operator who does not program	Less than protection level 0 to 3: specified by end user
6	Black key Switch position 1	End user: Trained operator who does not program	Program selection only, tool wear entries and work offset entries
7	Switch position 0	End user: Semi-skilled operator	No inputs and program selection possible, only the machine control panel can be operated

Note

Passwords should be regularly changed from an industrial security perspective.

Access protection

Benefits

- Protection against unauthorized interventions
- Access rights for protection levels 4 to 7 can be altered by the machine manufacturer or end user.
- Subprograms can be completely protected against unauthorized reading and display

Monitoring functions

Function

The SINUMERIK CNCs contain continuously active monitoring functions which detect faults in the CNC, PLC or machine in time.

When a fault occurs, machine operation is interrupted and the drives brought to a standstill. The cause of the fault is saved and displayed as an alarm. At the same time, the PLC is notified that a CNC alarm has been triggered.

The following areas are monitored:

- Read in
- Format
- Position 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

Benefits

- Early detection of faults
- Preventing damage on workpiece, tool or on the machine

Working area limitation

Function

Working area limitations define the area in which machining is permitted. These limitations refer to the basic coordinate system BCS. The tool tip is monitored to determine whether it is inside the protected working area (taking into account the tool radius).

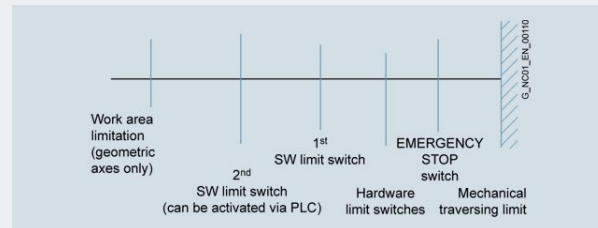
One value pair per axis (\pm) may be used to describe the protected working area. The upper and lower working area limits, which can be set and activated via setting data, may be modified using the G25/G26 commands.

Benefits

Working area limitations restrict the traversing range of the axes in addition to the limit switches. Protection areas in which tool motion is inhibited and which protect equipment such as tool turrets, measuring stations, etc., against damage, are thus set up in the machine working area.

→ [Work offsets](#)

Limit-switch monitoring



Overview of travel limits

Function

Preceding the emergency stop switch, hardware limit switches, which take the form of digital inputs controlled via the PLC interface, limit the traversing range of the machine axes. Deceleration is realized either as rapid deceleration with setpoint zero or along a braking characteristic. The axes must be retracted in the opposite direction in the JOG CNC operating mode. Software limit switches precede the hardware limit switches, are not passed, and are not active until reference point approach has been completed. Following preset, software limit switches are no longer effective. A second pair of plus/minus software limit switches can be activated via the PLC.

Benefits

- Machine protection

Position monitoring

Function

SINUMERIK CNCs provide extensive monitoring mechanisms for axis monitoring:

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

Position monitoring is always activated after termination of motion blocks according to the setpoint. To ensure that an axis is in position within a specified period of time, the timer configured in the machine data is started when a traversing block terminates; when the timer expires, a check is made to ascertain whether the following error fell below the limit value (machine data).

When the specified fine exact stop limit has been reached or following output of a new position setpoint other than zero (e.g. after positioning to coarse exact stop and subsequent block change), position monitoring is deactivated and replaced by standstill monitoring.

Position monitoring is effective for linear and rotary axes as well as for position-controlled spindles. Position monitoring is not active in the follow-up mode.

Benefits

- More precise manufacturing

Standstill monitoringFunction

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

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

Standstill monitoring is effective for linear and rotary axes as well as for position-controlled spindles. Standstill monitoring is inactive in the follow-up mode.

Benefits

- More precise manufacturing

→ *Position monitoring*

Clamping monitoring

Function

Clamping monitoring is one of the many extensive axis monitoring mechanisms implemented in SINUMERIK CNCs. When an axis is to be clamped on completion of the positioning action, it is possible to 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 target position during the clamping procedure.

The amount of deviation from the target position is set via the machine data. During the clamping procedure, clamping monitoring replaces standstill monitoring, and is effective for linear axes, rotary axes, and position-controlled spindles.

Clamping monitoring is not active in follow-up mode. When the monitoring function responds, its reactions are the same as those of the standstill monitoring.

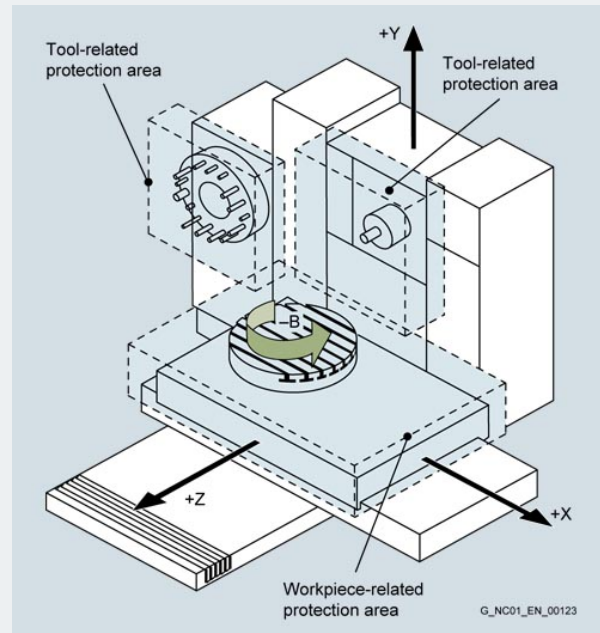
Benefits

- Monitors adherence to the parameterized clamping tolerance

→ *Position monitoring*

→ *Standstill monitoring*

2D/3D protection areas



Function

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

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

The 3-dimensionally programmed protection areas are displayed in 2D. This display also applies to the programmed working area limitations.

Benefits

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

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

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

Path length evaluation

Option

Order code: M53

Article No.: 6FC5800-0BM53-0YB0

Product ID: MCS3100

Function

With path length evaluation, data in the CNC can be buffered so that conclusions can be drawn with respect to the maintenance status of the machine. In the first stage, the following data is recorded:

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

This data is stored in the SRAM and is not affected by power on/off. This data can also be read through system variables in the part program and in synchronized actions.

In addition, using an external service, consistent data can be achieved for the complete life cycle of a machine.

Benefits

- Implementation of functions which allow conclusions to be drawn with respect to the maintenance status of the machine

→ *Synchronized actions*

Contour monitoring

Function

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.

Benefits

- More precise manufacturing

→ *Travel to fixed stop*

Contour monitoring with tunnel function

Option

Order code: M52

Article No.: 6FC5800-0BM52-0YB0

Function

With contour monitoring with tunnel function, the absolute movement of the tool tip in space can be monitored in 5-axis machining or when complex workpieces are being machined. This function provides optimum protection for high-quality workpieces. A cylindrical tunnel (tolerance band) 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.

Benefits

- More precise manufacturing
- Protect high-quality workpieces

Axis limitation from the PLC

Function

The preactivation of protection areas with specification of a position offset is programmed in the part program. The preactivated protection areas can be made operative in the PLC user program via the PLC interface. As a result, the relevant protection area is activated, for example, before a tool probe is swiveled into position in the working area, to see whether the tool or a workpiece is in the path of the swiveling part.

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

Benefits

- Protection of machine elements
e.g. after swiveling a tool probe or tailstock into position

→ *2D/3D protection areas*

Spindle speed limitation

Function

A lower (G25) and upper (G26) spindle speed limitation can be programmed.

Requirement

Actual position value sensor (measuring system) with corresponding resolution and direct mounting onto the spindle.

→ *Spindle functions*

Integrated ISM spindle monitor (S-Monitor)

Option

Order code: P55

Article No.: 6FC5800-0BP55-0YB0

Function

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

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

Benefits

- Visualization of current information for the spindle facilitates the diagnosis of faults, provides support in scheduling service intervals and for preventative maintenance measures.
- Easy and quickly configurable tool change and minimal chip-to-chip times due to digital transfer of already prepared spindle signals directly to control variables of the SINUMERIK, without inclusion of the PLC and without clock time dependency on the PLC cycle.
- Increase of spindle service life due to improved planning of preventative measures for servicing of the spindle raises productivity of the machine.

Requirement:

- WEISS spindle sensor module

Collision avoidance Protect MyMachine /3D Primitives

Option

Article No.: 6FC5800-0BS03-0YB0

Product ID: MCS41201

Function

Protect MyMachine /3D Primitives is a control-integrated function for collision avoidance on machine tools. Protect MyMachine /3D Primitives monitors static and moving machine components as 3D bodies in space in relation to the kinematics and machine axis positions of machine tools. The monitoring function is supported in the CNC operating modes JOG, MDI and AUTO:

- Protection area elements include simple bodies (cubes, cylinders, spheres, ...)
- Protection elements ≤ 34
- Protection areas ≤ 17
- Collision pairs ≤ 10
- 1-channel
- Graphical visualization (requirement: simultaneous recording)
- Transparent model data
- Simple configuration with copy & paste
- Tool holder, work holder and workpiece are not monitored
- No operator protection

Benefits

- Prompt and powerful protection in all operating situations against accidental collisions between moving and static machine components
- Machine damage can be minimized

Requirements:

- SINUMERIK CNC software version 6.13 and higher
- SINUMERIK NCU 1750, NCU 1760, PPU 1740
- Simultaneous recording for graphical visualization

→ [Collision avoidance Protect MyMachine /3D STL](#)

→ [Collision avoidance Protect MyMachine /Open](#)

→ [Simultaneous recording \(option P22\)](#)

Collision avoidance Protect MyMachine /3D STL

Option

Article No.: 6FC5800-0BS02-0YB0

Product ID: MCS41001

Function

Protect MyMachine /3D STL is a control-integrated function for collision avoidance on machine tools. With Protect MyMachine /3D STL, SINUMERIK provides reliable protection against undesirable collisions between moving and static machine components in the machine's working area – in all operating situations. Protection area elements include simple bodies (cubes, cylinders, spheres, ...) and bodies in STL format.

Protect MyMachine /3D STL is focused on critical situations encountered in practice, e.g. during machine setup or machining interruptions – in other words, when the operator intervenes in the process:

- 3D collision detection in real time
- Monitoring static and moving machine components
- Efficient handling of collision bodies in SINUMERIK Operate and data transfer from a CAD system (e.g. CA Config tool)

Benefits

- Precise modeling of the monitored machine bodies
- Prompt and powerful protection in all operating situations against accidental collisions between machine components
- Machine damage can be minimized

Requirements:

- SINUMERIK CNC software version 6.13 and higher
- 1-channel machine with one NCU
- SINUMERIK NCU 1750, NCU 1760, with certain restrictions also PPU 1740
- Simultaneous recording for graphical visualization

→ *Collision avoidance Protect MyMachine /3D Primitives*

→ *Collision avoidance Protect MyMachine /Open*

→ *Simultaneous recording (option P22)*

Collision avoidance Protect MyMachine /Open

Option

Article No.: 6FC5800-0BS04-0YB0

Product ID: MCS41101

Function

Protect MyMachine /Open is a control-integrated interface for connection to the external CAS collision avoidance from ModuleWorks. The overall solution monitors static and moving machine components as 3D bodies in space, depending on the kinematics and machine axis positions of the machine tools. The workpiece and clamping device are also monitored.

The monitoring function is supported in the CNC operating modes JOG, MDI and AUTO:

- Protection area elements include simple bodies (cubes, cylinders, spheres, ...) and STL files
- Protection elements, protection areas, and collision pairs are only limited by the memory and the performance of the IPC
- Protection of workpiece, tool, tool holder, and clamping device
- 1-channel
- Graphic visualization including removal simulation
- Enables the interface from the SINUMERIK to the external SIMATIC so the necessary data can be transferred for collision calculation
- No operator protection

Benefits

- Prompt and powerful protection in all operating situations against accidental collisions between machine components as well as workpiece and clamping devices
- Damage to the machine, workpiece and fixtures can be easily minimized.

Requirements:

- SINUMERIK CNC software version 6.13 and higher
- SIMATIC IPC427E / IPC627E
- ModuleWorks package (ModuleWorks GmbH)
- OPC UA

→ [Collision avoidance Protect MyMachine /3D Primitives](#)

→ [Collision avoidance Protect MyMachine /3D STL](#)

Collision protection Protect MyMachine /CC Axis Monitor

Option

Article No.: 6FC5800-0BN06-0YB0

Product ID: MCS41401

Function

Protect MyMachine /CC Axis Monitor supports collision protection of up to 20 axis pairs that, for example, move along a common guide rail and that could collide with each other. The axes concerned can also be active in different channels. The traversing directions of the axes of an axis pair can differ.

A maximum spacing can also be monitored.

Benefits

- Simpler and more cost-effective axis pair protection by way of permanent axis collision protection with activation of fewer parameters
- Ensuring productivity through availability of the machine, because costly repairs, machine downtimes, and production outages are avoided

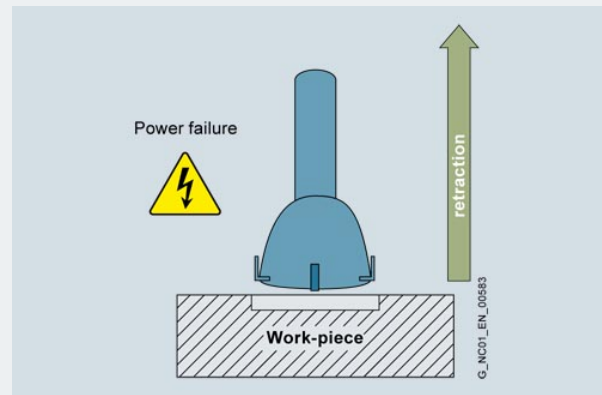
Restricted functionality for export versions:

Not possible.

Extended stop and retract ESR, drive-autonomous**Option**

Order code: M60

Article No.: 6FC5800-0BM60-0YB0

Function

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

Benefits

- Protection of workpiece and tool

Stop and retract ESR, extended (CNC-controlled and drive-autonomous)

Option

Order code: M61

Article No.: 6FC5800-0BM61-0YB0

Function

A safe position is assumed from the machining level without any collision between tool and workpiece. As well as the drive-autonomous stop and retract function, the CNC-controlled stop and retract function is also available.

To permit gentle interpolated retraction along the path or contour, the path interpolation can be processed further for a definable period following the trigger event.

The retraction axes are subsequently traversed in synchronism to an absolute or incremental position as programmed.

Benefits

- Protection of workpiece and tool
- Primarily used for gearing and grinding technologies

Integrated process monitoring Run MyCC /IPM**Option**

Order code: M70

Article No.: 6FC5800-0BM70-0YB0

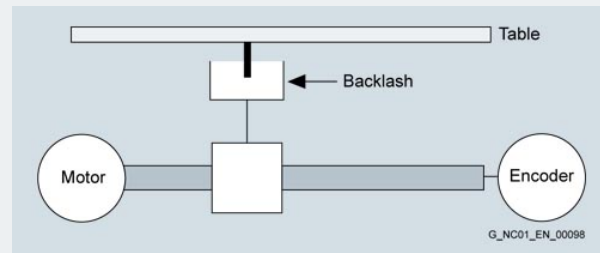
Function

The specific function Run MyCC /IPM is not intended for general use.

Restricted functionality for export versions:

Not possible.

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.

Function

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 the 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 second measuring system is available, the relevant backlashes must be entered for each of the two measuring systems.

Backlash compensation is always active in all modes following reference point approach.

Benefits

- Electronic compensation of mechanical conditions for precise manufacturing

Leadscrew/measuring system error compensationFunction

Compensation of thread pitch errors of the ball screw or errors of the measuring system, e.g. due to inaccuracies of the code disk/code ruler.

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

- Leadscrew error compensation (LEC) and 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. This means that the actual position of the axis can be derived from the position of the drive spindle (ideal situation).

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

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

Benefits

- More precise workpiece machining

Feedforward control (following error compensation)

Function

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

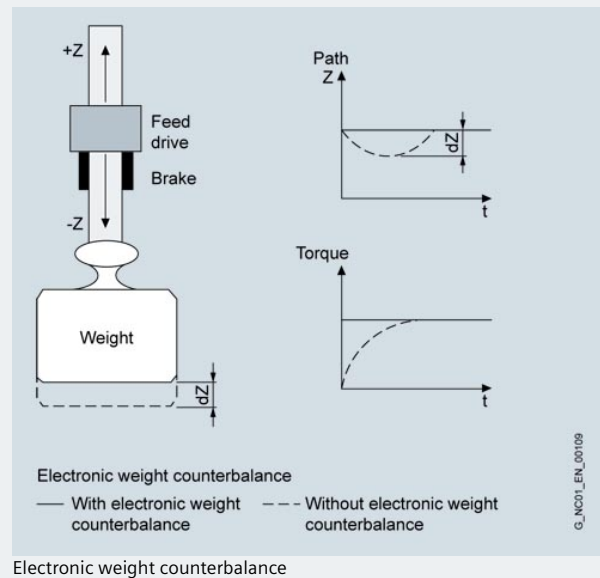
- Velocity-dependent speed feedforward control
The following error can be reduced almost to zero at constant velocity with this mode of feedforward control.
- Acceleration-dependent torque feedforward control
In order to achieve precise contours even when the demand for dynamics is at its highest, the torque feedforward control – provided the settings are right – can compensate the following error almost completely, even during acceleration.

Benefits

- Reduction of unwanted, speed-dependent contour errors during acceleration processes on contour curvatures, e.g. circles and corners
- Excellent machining precision, even at high path velocities

Weight counterbalance, electronic

Basic function of SINAMICS S120



Function

For weight-loaded axes without mechanical or hydraulic weight counterbalance, the vertical axis drops when the brake is released and the servo enable is switched on. This lowering dZ of the Z axis can be compensated by activating an electronic weight counterbalance.

After releasing the brake, the constant weight counterbalance torque now 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

Benefits

- Prevents lowering of weighted axes without mechanical or hydraulic counterbalance. After releasing the brake, the constant weight counterbalance torque now maintains the position of the vertical axis.
- More precise manufacturing

Temperature compensation

Function

Heat causes machine parts to expand. This expansion depends, among other things, on the temperature and thermal conductivity of the machine parts. The actual positions of the individual axes, which change depending on the temperature, have a negative effect on the precision with which workpieces are machined. These actual value changes can be corrected using temperature compensation.

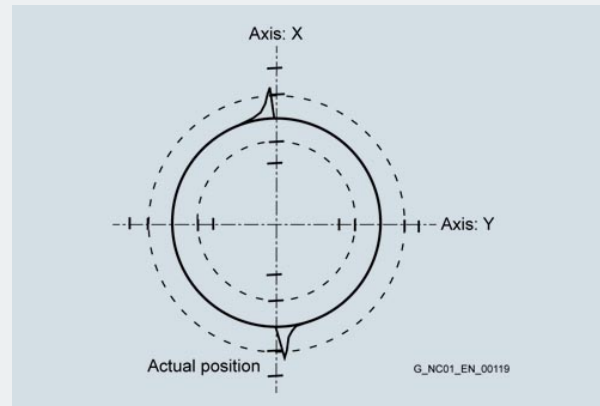
At a specific temperature, the actual-value offset over the positioning range of the axis is determined to create the error curve for this temperature value. Error curves for different temperatures can be defined for each axis. In order to ensure proper compensation of thermal expansion at all times, the temperature compensation value, reference position and gradient (angle) parameters must be transferred from the PLC to the SINUMERIK CNC system via function blocks each time the temperature changes.

Abrupt changes in these parameters are automatically smoothed by the SINUMERIK CNC system in order to prevent machine overload and avoid inadvertent triggering of monitoring functions.

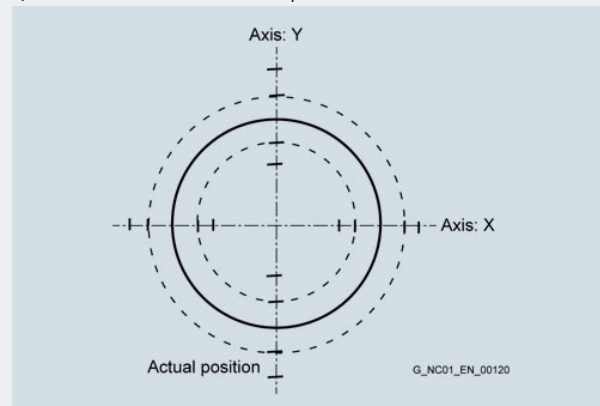
Benefits

- Compensation of thermal expansion of machine parts
- Consistent manufacturing accuracy of machined workpieces

Quadrant error compensation



Quadrant transitions without compensation



Quadrant transitions with quadrant error compensation

Function

Quadrant error compensation (friction compensation) ensures a much higher degree of contour precision, particularly when machining circular contours. At the quadrant transitions, one axis traverses at maximum path velocity while the second axis is stationary. The different friction conditions can cause contour errors.

Quadrant error compensation reliably compensates for this behavior and produces excellent results, without contour errors, in the very first machining operation. For operator-controlled quadrant error compensation, the intensity of the compensation pulse is set according to an acceleration-based characteristic. This characteristic is determined and parameterized during commissioning with the aid of the circularity test.

Benefits

- Considerably higher contour precision
- More precise manufacturing

→ *Circularity test*

→ *Friction compensation with adaptive characteristics (option S06)*

Circularity test

Function

During the circularity test, deviations of the actual position from the programmed radius – particularly at the quadrant transitions – are measured and graphically displayed while the circular contour is being traversed.

Benefits

- Creation and parameterization of a characteristic for quadrant error compensation.

Friction compensation with adaptive characteristics

Option

Order code: S06

Article No.: 6FC5800-0BS06-0YB0

Function

This function provides optimized friction compensation. The required setting parameters are calculated automatically.

Note

- Replaces Run MyCC /FRIC (option N67)

Benefits

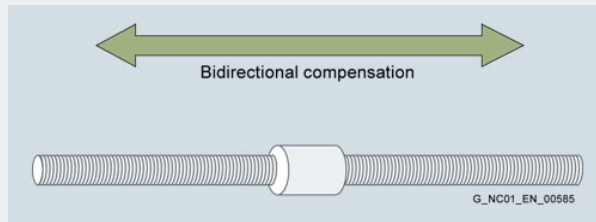
- Friction-dependent path deviations are eliminated even more effectively
- Consistently high contour accuracy and precision on the workpiece
- Simplified optimization

Leadscrew error compensation, bidirectional

Option

Order code: M54

Article No.: 6FC5800-0BM54-0YB0



Function

Bidirectional compensation is an expansion of the leadscrew error compensation function (LEC) and the measuring system error compensation function (MSEC). In contrast to LEC/MSEC, bidirectional compensation works in both directions.

Benefits

- Correction of mechanical spindle inaccuracies
- More precise manufacturing

Restricted functionality for export versions:

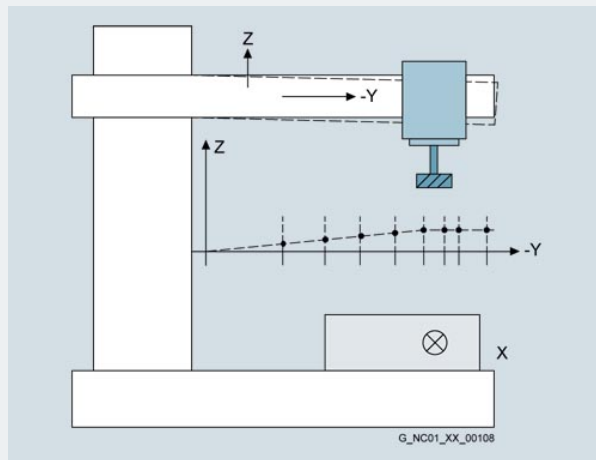
The correctable tolerance band is restricted to 1 mm (0.039 in).

Sag compensation, multi-dimensional

Option

Order code: M55

Article No.: 6FC5800-0BM55-0YB0



Sag compensation

Function

Multi-dimensional compensation is also possible for the effects of physical influences and manufacturing tolerances, such as sag or leadscrew pitch errors. The compensation tables can be switched over from the PLC. When the reference axis and the compensating axis are identical, leadscrew pitch errors can be compensated. By transferring weighting factors (PLC interface), stored compensating characteristics can be adapted to different conditions, e.g. tools.

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

- Independent error curves can be defined: number = $2 \times (\text{number of axes}_{\text{max}})$
- Freely selectable compensation positions: configurable number (depending on the CNC user memory configuration)
- Interpolating inclusion of compensation values
- Weighting factor for compensation of tool weights
- Reference axis and compensating axis are selectable

The correctable tolerance band is restricted to 10 mm (0.39 in) for versions with full functional scope.

Benefits

- Multi-dimensional compensation for the effects of physical influences and manufacturing tolerances such as sag or leadscrew pitch errors
- More precise manufacturing

Restricted functionality for export versions:

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

Nodding compensation Run MyCC /NOCO

Option

Article No.: 6FC5800-0BN63-0YB0

Product ID: MCS2130

Function

Run MyCC /NOCO allows workpieces to be machined with greater precision by compensating for mechanical effects between axes. Run MyCC /NOCO is a useful function for traveling column machines, for example. Sometimes, the acceleration and braking of other axes can lead to a "nodding" motion in the tower, causing the tool tip to make marks in the material as it is lowered into the workpiece.

The Run MyCC /NOCO nodding compensation function monitors the other axes and models the nodding behavior. Since the acceleration levels of all axes are known, the effects of the nodding motion can be compensated by feedforward control of the other axes.

Benefits

- Increases machining precision and improves surface quality
- Improves productivity because nodding motion of the machine is compensated, allowing for increased acceleration
- Saves costs because often more cost-effective than mechanical measures to increase stiffness

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

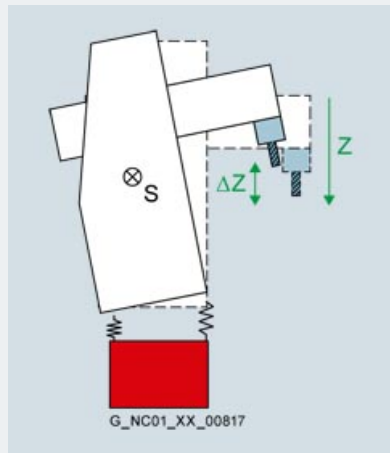
Not possible.

Nodding compensation ECO

Option

Order code: S20

Article No.: 6FC5800-0BS20-0YB0



Function

ECO nodding compensation is used to offset dynamic position deviations along a linear machine axis occurring due to acceleration along this or another linear machine axis.

The position deviation results from mechanical compliance within the machine. ECO nodding compensation counteracts the relevant position deviation with correction movements along a machine axis, e.g. compensation of ΔZ .

Benefits

- Improved machining quality with simultaneously increased jerk and acceleration values
- Increased productivity can be achieved without costly mechanical reinforcement of the machine
- Saves costs because often more cost-effective than mechanical measures to increase stiffness
- Can be used, for milling, multitasking, tapping, and laser and water jet machining, for example

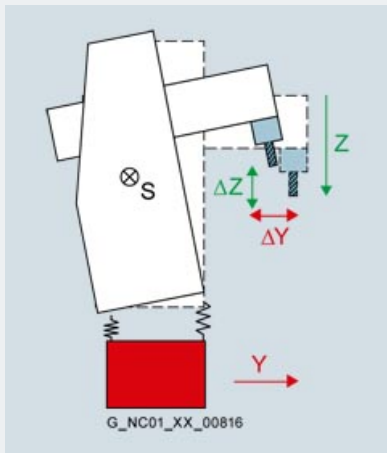
→ *Nodding compensation ADVANCED (option S21)*

Nodding compensation ADVANCED

Option

Order code: S21

Article No.: 6FC5800-0BS21-0YB0



Function

ADVANCED nodding compensation counteracts dynamic position deviations along linear machine axes that occur due to acceleration processes along this or another linear machine axis, or result from mechanical compliance within the machine. ADVANCED nodding compensation counteracts multiple deviations with correction movements along multiple machines axes, e.g. compensation of ΔY and ΔZ .

Benefits

- Improved machining quality with simultaneously increased jerk and acceleration values
- Increased productivity can be achieved without costly mechanical reinforcement of the machine
- Saves costs because often more cost-effective than mechanical measures to increase stiffness
- Can be used, for milling, multitasking, tapping, and laser and water jet machining, for example

→ *Nodding compensation ECO (option S20)*

Universal spatial compensation interface Run MyCCI /UCI

Option

Order code: N75

Article No.: 6FC5800-0BN75-0YB0

Function

Run My CCI /UCI provides an interface for the integration of customized compensation algorithms.

Note

- Replaces Run My CCI /UCI (option N74)

Benefits

- Integration of customer-specific compensation algorithms

Requirement:

- Create MyCC

Restricted functionality for export versions:

Not possible.

Spatial compensation Run MyCC /VCS-ECO

Option

Order code: N18

Article No.: 6FC5800-0BN18-0YB0

Function

Run MyCC /VCS-ECO is a version of the Volumetric Compensation System with reduced performance capacity.

Run MYCC /VCS-ECO enables compensation of the position deviations of rotary axes (=deviation deriving from non-ideal rotation vectors) and the position deviations of linear axes (=deviations from perpendicularity).

However, a position-specific compensation of deviations is not possible.

Benefits

- With the entry-level version Run MyCC /VCS-ECO position deviations can be compensated easily and reliably
- The position deviations can also be determined using conventional measuring instruments without the need for laser-based instruments,
for example they can be measured using kinematics CYCLE996 or using help from Solution Partners

Requirement:

- Loadable compile cycle

→ *Spatial compensation Run MyCC /VCS-A3*

→ *Spatial compensation Run MyCC /VCS-A5*

→ *Spatial compensation Run MyCC /VCS-A5 plus*

→ *Spatial compensation Run MyCC /VCS ROT*

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

Option

Order code: M57

Article No.: 6FC5800-0BM57-0YB0

Note

The specific function Run MyCC /SEC-KT is not intended for general use.

Restricted functionality for export versions:

Not possible.

For Cartesian machines, refer to:

→ *Spatial compensation Run MyCC /VCS-A3*

→ *Spatial compensation Run MyCC /VCS-A5*

→ *Spatial compensation Run MyCC /VCS-A5 plus*

Vibration extinction Run MyCC /VIBX

Option

Order code: N11

Article No.: 6FC5800-0BN11-0YB0

Function

The Run MyCC /VIBX vibration extinction function supports axis-specific damping of machine vibrations. Up to 8 axes can be parameterized in the SINUMERIK CNC system, each with 2 machine data for the filter frequency and the required damping factor.

Benefits

- Significantly reduces disturbing oscillations that follow positioning

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Not possible.

Cogging torque compensation axis/spindle

Option

Order code: D51 to D59

Article No.: 6FC5800-0BD50-0YB0

Function

The cogging torque compensation compensates for motor cogging torques at low speeds of rotary or linear synchronous motors. Cogging torques are a property of the respective motor. For this reason, each motor requires a separate compensation table which must be filled during a learning cycle.

Cogging torque compensation is a drive function.

Benefits

- More precise manufacturing
- Alternative to Run MyCC /COCO

→ *Magnetic cogging torque compensation Run MyCC /COCO (option N46)*

Magnetic cogging torque compensation, Run MyCC /COCO

Option

Order code: N46

Article No.: 6FC5800-0BN46-0YB0

Function

Run MyCC /COCO magnetic cogging torque compensation is used particularly for electric direct drives where it is necessary to reduce the torque ripple at low speeds in order to obtain a better machining quality.

Benefits

- Improved synchronous operation behavior due to compensation of the disturbance variable: Cogging torque
- Better machining result

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Not possible.

→ *Cogging torque compensation axis/spindle (options D51 to D59)*

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

Option

Order code: M81

Article No.: 6FC5800-0BM81-0YB0

Function

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

Benefits

- Auxiliary function for the operation of robots with forced axis coupling
- Allows the operation of special robots in conjunction with Run MyCC /ROBX or Run MyCC /ROBX AR

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:

Not possible.

→ *Transformation robotic extended Run MyCC /ROBX (option N54)*

→ *Transformation for articulated arm robots Run MyRobot /ROBX AR (option R05)*

Learning error compensation Run MyCC /LECO

Option

Order code: N77

Article No.: 6FC5800-0BN77-0YB0

Function

The Run MyCC /LECO learning error compensation implements a learning algorithm which takes the position error resulting from the effect of cyclic distortion during a machining process into consideration. Within a few learning periods, LECO learns how these effects can be compensated. It evaluates and adds compensation values to the speed output of the position controller.

The function can implement a compensation feedforward control pulse to the real-time pulse input of the SINAMICS drive. The function includes a trace, with which various signals from up to 5000 positions can be recorded.

Note

The specific function Run MyCC /IPM is not intended for general use.

Benefits

- It takes the position error resulted from the effect of the cyclic distortion during a machining process into consideration

Restricted functionality for export versions:

Not possible.

Intelligent Load Control (ILC)

Option

Order code: S11

Article No.: 6FC5800-0BS11-0YB0

Function

Intelligent Load Control (ILC) makes it possible to adapt dynamic and control parameters to the actual load of a linear or rotary axis using predefined adaptation characteristic curves.

With the support of operating cycle Cycle782 the current load of an axis can be measured or specified.

Deployment is recommended in particular for direct-driven axes where load inertia can be significant with regard to the intrinsic load inertia.

Benefits

- An optimum dynamic response of the machine axes is ensured by adjusting dynamic parameters such as jerk and acceleration, as well as adapting the position and speed control parameters.
- The immediate result is shorter machining times and better control quality in conjunction with increased precision.

Intelligent Dynamic Control (IDC)

Option

Order code: S23

Article No.: 6FC5800-0BS23-0YB0

Function

The "Intelligent Dynamic Control" function adjusts the control parameters for axes with a stability that changes at different positions or at different traversing velocities, depending on the position or velocity. By adapting control parameters, increased dynamics and better control quality can be achieved.

Benefits

- The best possible machining conditions throughout the whole working area
- Increased machining accuracy thanks to situation-dependent parameter adaptation

PLC

Function

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 statement list STL, function block diagram FBD and ladder diagram LAD programming languages are available. In addition, system function blocks SFB and system functions SFC integrated in the operating system can also be called. The PLC can also be programmed in other SIMATIC S7 high-level languages, such as S7-Graph or Structured Control Language SCL.

Benefits

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

- Controlling positioning axes
- Executing synchronized actions (auxiliary functions)
- Reading and writing NCK system/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/IPC 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 cyclic exchange of data is performed by the basic program at the start of the PLC cycle (OB1). This ensures, for example, that the signals from the NCK remain constant throughout a PLC cycle.
- **Event-driven signal transfer NCK → PLC**
PLC functions executed depending on the workpiece program are initiated via auxiliary functions in the workpiece program. If a block with auxiliary functions is executed, the type of auxiliary function determines whether the NCK has to wait for this function to execute (e.g. tool change) or whether the function will be executed together with the 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. Data transfer is, therefore, interrupt-driven and acknowledgement-driven. The basic program evaluates the signals and data, sends an acknowledgment to the NCK, transfers some of the data to OB40 and the rest to the user interface at the beginning of the cycle. CNC machining is not affected if the data do not require an acknowledgment from the user.

PLC

- **Event-driven signal exchange PLC → NCK**
Whenever the PLC sends a request to the NCK (such as a request to traverse an auxiliary axis), a PLC → NCK event-driven signal exchange takes place. In this case, data transfer is also acknowledgment-driven. When performed from the user program, this type of signal exchange is triggered using a function block (FB) or function call (FC). The associated FBs and FCs are provided together with the basic program.
- **Messages**
The acquisition and editing of user messages is handled by the basic program. The message signals are 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 user memory

Function

In the PLC user memory of the PLC CPU, the PLC user program and the user data are stored together with the PLC basic program. The memory of the PLC CPU is segmented according to load memory, work memory and system memory. Load memory is retentive, and takes the form of either integrated RAM or a RAM module (plug-in memory card). It contains data, program and decompiling information. The load memory and the high-speed work memory for execution-relevant program tests provide sufficient space for user programs.

Benefits

- Memory for PLC user program and the user data

→ *PLC user memory, expanded (options D11 ... D18)*

Programmable logic controller PLC

PLC user memory, expanded

Option

Order codes: D11 ... D18

Article No.: 6FC5800-0BD10-0YB0

Function

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

Benefits

- Expansion of the memory for the PLC user program and the user data

Safety Integrated

Function

SINUMERIK Safety Integrated provides integrated safety functions that support the implementation of highly effective personnel and machine protection. These safety functions meet the requirements of EN 61508 for use up to and including SIL2 and Category 3, as well as PL d according to EN ISO 13849. The safe logic is implemented via the integrated SIMATIC failsafe PLC S7-1500. This permits not only simple and economic implementation of the main requirements for functional safety, but also the realization of practical operating and security concepts.

Available functions include, among others:

- Safe Speed Monitor and Safe Stop functions
- Functions for establishing safe boundaries in working and protection areas, and for range recognition
- Direct connection of all safety-related signals and their internal logical linking
- Safe Brake Management
- Safe communication via PROFIBUS or PROFINET
- Operator-controlled acceptance test – integrated in SINUMERIK Operate

Benefits

- Highly efficient personnel and machine protection

→ *Safety Integrated - F-PLC*

→ *Safety Integrated - axis/spindle*

→ *Safety Integrated - multi-axis package*

→ *Axis/spindle*

→ *Positioning axis/auxiliary spindle*

Drive-based safety functions and integrated F-PLC

Benefits

- Highly efficient personnel and machine protection

→ *Safety Integrated*

Safety Integrated – axis/spindle

Option

Order code: K01 ... K09

Article No.: 6FC5800-0BK00-0YB0

Function

Extended motion monitoring functions for an analog axis/spindle.

→ *Safety Integrated*

Safety Integrated – F-PLC

Option

Order code: S60

Article No.: 6FC5800-0BS60-0YB0

Function

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

→ *Safety Integrated*

Safety Integrated – multi-axis package

Option

Order code: S61

Article No.: 6FC5800-0BS61-0YB0

Function

Extended motion monitoring functions for all NCK drives (axes/spindles and positioning axes/auxiliary spindles) activated in the SINUMERIK.

→ *Safety Integrated*

Industrial security

Function

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

Siemens (Business Unit Motion Control) is the first company worldwide to employ mandatory consideration of their security processes for product development in accordance with standard IEC 62443-4-1, and to allow them to be certified by an external auditor.

Benefits

Security-related aspects such as safety of third-party components, assurance of processes and quality, as well as safe architecture and the handling of weak points, all the way to security patch management are key aspects of the development process.

→ www.siemens.com/industrialsecurity

Auto Servo Tuning ASTFunction

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

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

Benefits

- High stability and quality for motion control involving a wide range of different workpiece weights or clamping operations, and for linear and torque motors
- Optimum productivity of the machine tool due to controllable dynamic response adaptation to production conditions

→ *Call Auto Servo Tuning from part program AST call (option S10)*

Call Auto Servo Tuning from part program AST call

Option

Order code: S10

Article No.: 6FC5800-0BS10-0YB0

Function

The AST call function allows machines to be automatically optimized again from the part program.

Benefits

- Operator-friendly and easy-to-use for the machine programmer by calling the automatic controller optimization directly from the part program

→ *Auto Servo Tuning AST*

Automated commissioning Create MyConfig

Function

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

Benefits

- Simplified commissioning or upgrades
- Less time required for commissioning or upgrading
- Automated process steps prevent errors

Commissioning series machines

Function

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

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

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

Commissioning drives and backing up drive data

Function

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

	Internal drives (NCU, NX) and external drives (DP, PN) (with bus system)			Via terminals (without bus system)
	NCK drives	PLC axes (FC18)	PLC axes (FB283)	Traversing commands via terminals I/Os
Commissioning				
• With SINUMERIK Operate	✓	✓	✓	–
• With Starter	✓ ¹⁾	✓ ¹⁾	✓	✓
Back up drive data in drive archive				
• With SINUMERIK Operate	✓	✓	✓	–
• With Starter	✓	✓	✓	✓

¹⁾ Current/speed controller level only.

Configuring, commissioning, optimization and service

Function

Preferred tools for configuring, commissioning, optimization and service.

	Configuring (without machine)	Commissioning (at the machine)	Optimization	Service
SINUMERIK Operate	–	1. Axis, axis-drive allocation, compensation 2. Safety Integrated incl. acceptance test and DB-SI	Optimize axes manually, current, speed and position controller, filter settings, trace, etc.	Troubleshooting
SINUMERIK Operate Auto Servo Tuning	Optimize individual or grouped axes subsequently offline using customer data	–	Automatically optimize individual or grouped axes	Automatically optimize individual or grouped axes subsequently
Create MyConfig	Modular machines, series machines with variations	Edit data with UPDiff, axis/drive allocation, direct adaptation of topology	–	Expand machine
Access MyMachine	Adapt user alarms, integrate user screens, tool management, boot stick	Access SINUMERIK Operate and the active file system	Adapt optimization data, possibly via remote access	Remote access via TS adapter
SIMATIC STEP 7	Configuring, e.g. tool management	Read in hardware configuration, PLC, e.g. configure DB18, transfer user program	–	Troubleshooting, optimize processes

CNC program messages

Function

All messages programmed in the part program and all alarms recognized by the system are displayed on the operator panel in plain text. Alarms and messages are displayed separately.

Messages can be programmed to give the user information on the current processing status while the program is executing.

Benefits

- Information can be provided directly to the operator

→ *Alarms and messages*

Alarms and messages

Function

All messages and alarms are displayed on the operator panel in plain text with date and time as well as the appropriate symbol for the delete 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. The contents of variables can also be displayed in message texts. Alarms may also be used in the part program. An alarm always goes hand in hand with a response from the CNC according to the alarm category. A large number of alarm numbers are available to the user for alarms and messages from the part program. Please refer to the programming instructions for the alarm numbers that can be used for this purpose.
- Alarms and messages from the PLC:
Machine-specific alarms and messages can be displayed directly from the PLC program in plain text. Messages comprise status messages and error messages. In the case of status messages, the display is immediately deleted when the condition is no longer active, error messages must be acknowledged. User-specific alarm numbers of the PLC can be assigned to general, channel-specific, axis-specific and spindle-specific user alarms and messages. Please refer to the PLC documentation for the alarm numbers that can be used. The response of the CNC to alarms or messages can be configured. The configured alarm and message texts are saved in application-specific text files.
- Specific evaluation of alarms:
A channel-specific signal can be used to decide whether other channels may continue to be used when an alarm is issued.

Benefits

- The operator receives information regarding the current machining situation during execution of the program

Action log

Function

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

Benefits

- Retracing of operator actions

PLC status

Function

The user can check and modify PLC status signals in the Diagnostics operating area via the operator panel.

Benefits

- 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-named signals can be changed for tests. Signal combinations are also possible, and as many as 10 operands can be modified simultaneously.

Access MyMachine /P2P

Option

Order code: P30

Article No.: 6FC5800-0BP30-0YB0

Product ID: MCS31401

Function

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

Benefits

- Increased machine availability due to speedy online presence
- Better preparation of actually required service operations

Use HMI applications Run MyScreens

Option

Order code: P64

Article No.: 6FC5800-0BP64-0YB0

Function

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

This function is implemented via an integrated interpreter and via configuring files containing the description of the user interface. The screen forms can be designed directly on the CNC itself.

Users can also write their own cycle forms that can also be called from the part program editor.

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

The following functions can be implemented with Run MyScreens:

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

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.

Benefits

- The user interface can be expanded even in the basic version by up to 5 screen forms using predefined softkeys with the integrated editor.

Requirements

- The option 6FC5800-0BP64-0YB0 is required to add more than 5 screen forms with Run MyScreens.

Use HMI applications Run MyHMI /3GL

Option

Order code: P60

Article No.: 6FC5800-0BP60-0YB0

Product ID: MCS11101

Function

Run MyHMI /3GL is used to execute programmed HMI applications. The applications are programmed with either Qt/C++, .net (C#, VB) or C++.

This option is also needed if third-party software or background functions with data communication (application without HMI components) are to be integrated in SINUMERIK Operate.

Benefits

- Use of expanded or in-house user interfaces

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

Program screens, operating areas and user interfaces Create MyHMI

Function

Create MyHMI offers 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).

Benefits

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

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

→ *Use HMI applications Run MyHMI /3GL (option P60)*

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

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

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

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

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

The following functions can be implemented, for example, with the 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

Notes:

- Use of the Create MyHMI /3GL package requires an OEM agreement.
- The Run MyHMI /3GL runtime license is required to run the programmed user screens.

Benefits

Configuration of machine manufacturer or end user-specific functional expansions or proprietary user interfaces to stand out from the competition.

→ *Use HMI applications Run MyHMI /3GL (option P60)*

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

Option

Article No.: 6FC5800-0BP61-0YB0

Product ID: MCS11601

Function

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

Benefits

- Configuration of machine manufacturer or end user-specific functional expansions to stand out from the competition

User interface on SIMATIC Panel Run MyHMI /SIMATIC OP for SIMATIC Comfort and Mobile Panels

Option

Article No.: 6FC5800-0BP03-0YB0

Product ID: MCS11501

Function

Run MyHMI /SIMATIC OP allows operation of the SINUMERIK NCU with its own user interface on SIMATIC Comfort Panels. The user interface required on the SIMATIC Comfort Panel can be created simply using the SIMATIC WinCC ES Advanced Engineering System and the add-on Create MyHMI /WinCC.

Benefits

The machine manufacturer enjoys even more freedom in the selection of panels. Thus, the machine manufacturer also fall back on the wide variety of SIMATIC Comfort Panels.

Use standardized and freely configurable operating screens, Run MyHMI /PRO

Option

Order code: P47

Article No.: 6FC5800-0BP47-0YB0

Product ID: MCS11301

Function

The runtime license Run MyHMI /PRO allows users to run configurations on the SINUMERIK PCU (SIMATIC IPC for SINUMERIK) or the NCU that have been created with the HMI PRO CS configuration system.

The option includes the standardized operating screens within HMI PRO as well as the freely configurable user screens that have been created using the variable layout.

Benefits

- The HMI PRO user interface provides cross-vendor, uniform machine operation and diagnostics for mechanical production and assembly.
- Operators and maintenance personnel enjoy the benefits of standardization through uniform operation and diagnostics for all machines and plants.