© Siemens 2020

Mode group	Option Order code: C01 ... C09 Article No.: 6FC5800-0AC00-0YB0
	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.

© Siemens 2020

Machining channel, additional	Option Order codes: C11 ... C19 Article No.: 6FC5800-0AC10-0YB0

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*

© Siemens 2020

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

© Siemens 2020

CNC user memory, additional	Option Order codes: D01 ... D06 Article No.: 6FC5800-0AD00-0YB0

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
- Battery-buffered CNC user memory

→ *HMI user memory additionally on CF card of NCU (option P12)*

© Siemens 2020

CNC user memory, expanded	Option Order code: P77 Article No.: 6FC5800-0AP77-0YB0

Function

The function "CNC user memory expanded" increases the CNC user memory to 100 MB. With the additional HMI user memory on the CF card of the NCU, it is possible to increase the size of the CNC user memory up to 6 GB.
With a SINUMERIK PCU 50, the CNC user memory can be increased to 40 GB, for example.
With a SIMATIC IPC for SINUMERIK, the CNC user memory can be increased to 120 GB, for example.

Benefits

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

→ HMI user memory additionally on CF card of NCU (option P12)

© Siemens 2020

HMI user memory additionally on CF card of NCU	Option Order code: P12 Article No.: 6FC5800-0AP12-0YB0

Function

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

Note

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

Benefits

- Larger HMI user memory without SINUMERIK PCU or SIMATIC IPC

→ CNC user memory, additional (options D01 ... D06)

© Siemens 2020

Axis/spindle	Option Order codes: A01 ... A28 Article No.: 6FC5800-0AA00-0YB0

Function

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

- 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

Benefits

- Expansion of number of axes/number of spindles compared with the basic version

→ *Positioning axis/auxiliary spindle*
→ *Simulation axis/spindle*
→ *Virtual axis*
→ *Internal/external drives*

© Siemens 2020

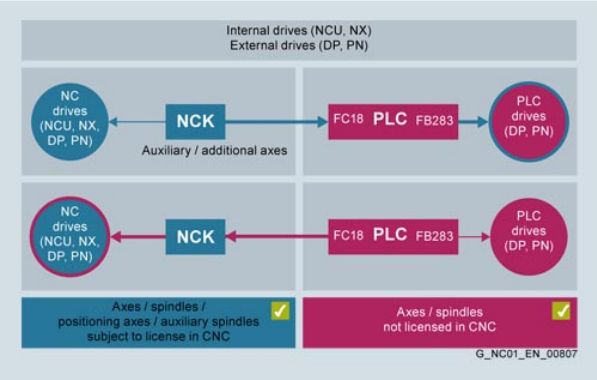
Positioning axis/auxiliary spindle		Option Order codes: B01 ... B28 Article No.: 6FC5800-0AB00-0YB0
	Function Positioning axes can execute movements in parallel to the actual machining process. They can be programmed with an axis-specific feedrate in the part program. Axis movement beyond block boundaries is also possible. Positioning axes can also be controlled by the PLC. This means that axis movements can be started independently of the part program without requiring an additional machining channel. Auxiliary spindles are speed-controlled spindle drives without an actual-position sensor, e.g. for tool drives.	
	POS/SPOS/M3, M4, M5 (from CNC block)	✔
	POSA/SPOSA (from CNC block, modally)	✔
	FC18/POS/SPOS/M3, M4, M5 (PLC axes)	✔
	PLC-VDI interface (M3, M4, M5 directly)	✔
	OSCILL (asynchronous oscillation)	✔
	OSCILL (synchronous oscillation)	✔
	do POS/SPOS/M3, M4, M5 (synchronized actions)	✔
	Couplings (TRAIL, gantry and master/slave (positioning axis as slave axis), LEAD, EG, CP, etc.)	✔
	Setpoint and actual value available	✔
	Commissioning with SINUMERIK Operate	✔
	Path axes/geometry axes/additional path axis/GEOAX()	–
	Tapping and thread cutting with compensating chuck	–
	Note An additional positioning axis/auxiliary spindle cannot interpolate with other axes.	

© Siemens 2020

Positioning axis/auxiliary spindle	Option Order codes: B01 ... B28 Article No.: 6FC5800-0AB00-0YB0
	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*

© Siemens 2020

PLC axes



Function

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

Benefits

- Axes may optionally be controlled by the PLC

→ *Axis/spindle*
→ *Simulation axis/spindle*
→ *Positioning axis/auxiliary spindle*
→ *Virtual axis*
→ *Internal/external drives*

© Siemens 2020

Multi-axis package	**Option** Order code: M10 Article No.: 6FC5800-0AM10-0YB0
	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

© Siemens 2020

Internal/external drives

Function

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

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*

© Siemens 2020

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 the 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

© Siemens 2020

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

© Siemens 2020

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

© Siemens 2020

Top Speed	Option Order code: S13 Article No.: 6FC5800-0AS13-0YB1
	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*

© Siemens 2020

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

© Siemens 2020

Jerk adaptation	Option Order code: S22 Article No.: 6FC5800-0AS22-0YB0

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

© Siemens 2020

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

© Siemens 2020

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

© Siemens 2020

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

© Siemens 2020

Measuring systems 1 and 2, selectable

Function

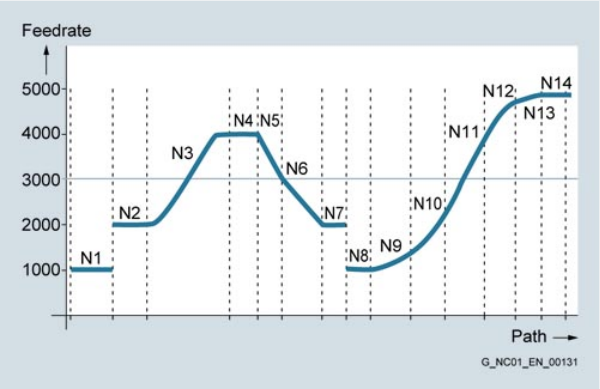
For special applications, 2 encoders can be assigned to one axis, such as for a direct measuring system for the machining process with high demands on accuracy, and an indirect measuring system for high-speed positioning tasks.
The switchover between measuring systems 1 and 2 is performed via the PLC.

Benefits

- Quicker positioning
- More precise production

© Siemens 2020

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*

© Siemens 2020

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

© Siemens 2020

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.

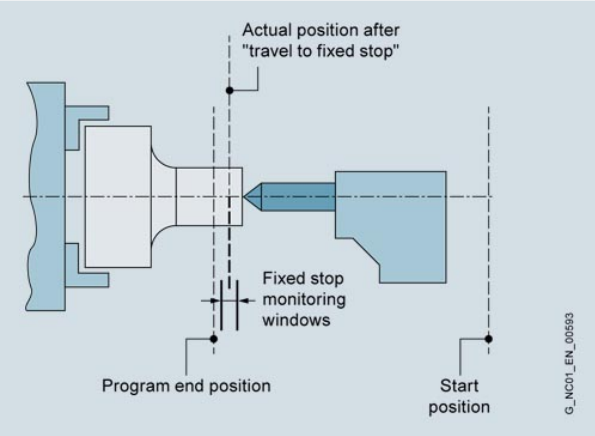
Benefits

- Clamping of workpieces

© Siemens 2020

Travel to fixed stop with Force Control

Option
Order code: M01
Article No.: 6FC5800-0AM01-0YB0



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

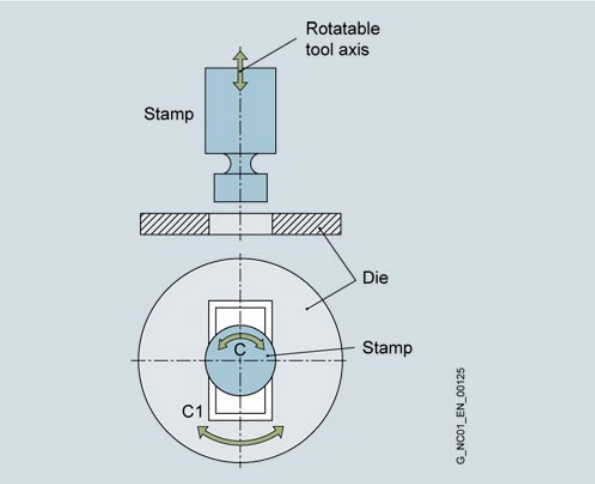
© Siemens 2020

Setpoint exchange	Option Order code: M05 Article No.: 6FC5800-0AM05-0YB0
	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

© Siemens 2020

Tangential control

Option
Order code: M06
Article No.: 6FC5800-0AM06-0YB0



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

© Siemens 2020

Position switching signals/cam controller	Option Order code: M07 Article No.: 6FC5800-0AM07-0YB0

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*

© Siemens 2020

Advanced Position Control ECO (APC ECO)	**Option** Order code: M12 Article No.: 6FC5800-0AM12-0YB0

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 were mechanical natural frequencies can be clearly identified in the motor signals

© Siemens 2020

Advanced Position Control APC	Option Order code: M13 Article No.: 6FC5800-0AM13-0YB0

Function

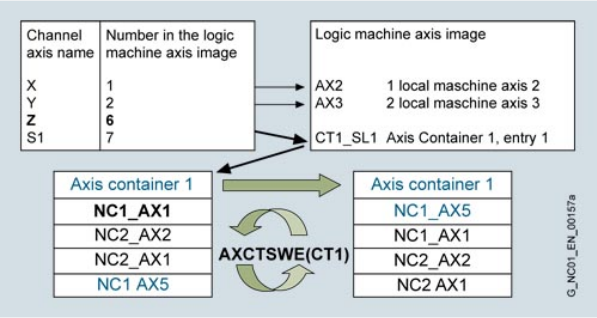
Mechanical natural frequencies of the machine axes can be actively damped with Advanced Position Control APC via a closed control loop. Dampening the oscillations is achieved by moving the motor to oppose the oscillation.
The status variable necessary for damping can be derived from the direct or/and 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.

© Siemens 2020

Axis container



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

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.

© Siemens 2020

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 SPOS[1)]
- Spindle monitoring with the functions[1)]:
  - Axis/spindle stationary ($n < n_{min}$)
  - Spindle in setpoint range
  - Spindle speed, max.
  - Programmable lower (G25) and upper (G26) spindle speed limitation
  - Speed of the gear stage 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).

© Siemens 2020

Spindle functions

- Thread cutting with constant pitch[1]:
  The following thread types can be produced with G33: cylindrical, taper and face thread, single-start or multiple-start, as left-hand or right-hand thread. In addition, multiple-block threads can be produced by concatenating threading blocks.
- Thread cutting with variable lead[1]:
  Threads can also be programmed with linearly progressive (G34) or linearly degressive (G35) lead.
- Programmable thread run-in and run-out path:
  When thread cutting, DITS/DITE (displacement thread start/end) can be used to program the path ramp for the acceleration or deceleration process as a distance. This makes it possible, to adjust e.g. 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).

© Siemens 2020

Scalable analog setpoint Run MyCC /SANS	**Option** Order code: N48 Article No.: 6FC5800-0AN48-0YB0

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.

© Siemens 2020

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.

© Siemens 2020

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

© Siemens 2020

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.

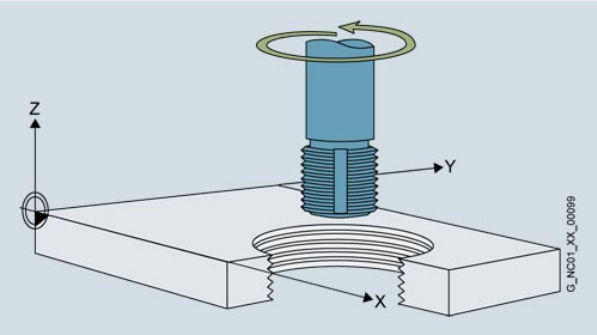
Three points are required to program the circle: the starting point, the intermediate point and the end point.

Benefits

- Circular-path programming obliquely in space

© Siemens 2020

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

© Siemens 2020

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:

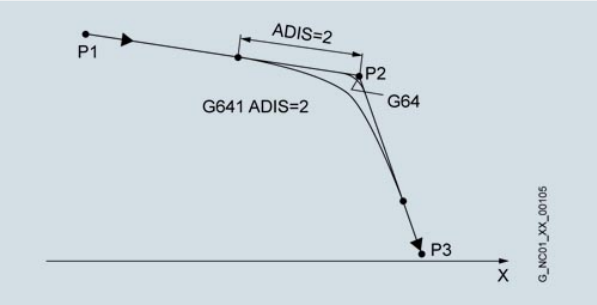
- Linear interpolation
- Circular interpolation
- Helical interpolation
- Involute interpolation
- Splines (A, B, C)
- Polynomial

Benefits

- Structured program layout

© Siemens 2020

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

© Siemens 2020

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

Option
Order code: M24
Article No.: 6FC5800-0AM24-0YB0

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.

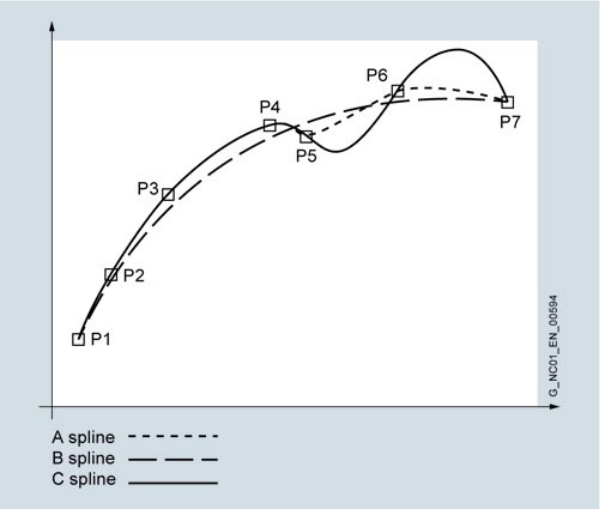
Benefits
Continue machining is used, for example, when the machine operator only notices the interruption in a 2D laser or water jet cutting operation a few blocks after the actual interruption. The head has usually already progressed further in the machining process, and must, therefore, be returned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

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

© Siemens 2020

Spline interpolation (A, B and C splines)

Option
Order code: S16
Article No.: 6FC5800-0AS16-0YB0



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.

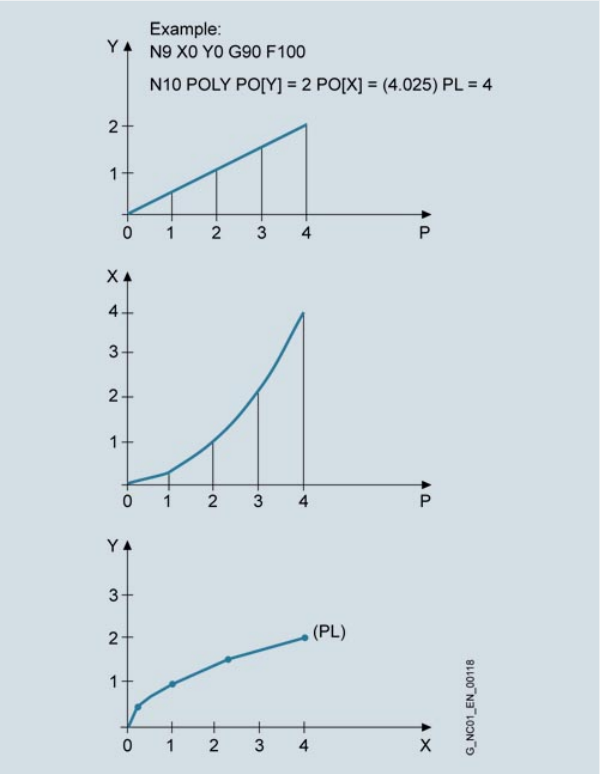
Benefits

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.

© Siemens 2020

Polynomial interpolation

Option
Order code: M18
Article No.: 6FC5800-0AM18-0YB0



Polynomial interpolation

Function

Curves can be interpolated using polynomial interpolation, whereby the CNC axes follow the function:
$f(p) = a_0 + a_1p + a_2p^2 + a_3p^3 + a_4p^4 + a_5p^5$ (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.

© Siemens 2020

Involute interpolation	Option Order code: M21 Article No.: 6FC5800-0AM21-0YB0

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.

© Siemens 2020

Multi-axis interpolation > 4 interpolating axes	**Option** Order code: M15 Article No.: 6FC5800-0AM15-0YB0

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.

© Siemens 2020

Crank interpolation Run MyCC /CRIP	Option Order code: N04 Article No.: 6FC5800-0AN04-0YB0

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.

© Siemens 2020

Advanced Surface motion control	Option Order code: S07 Article No.: 6FC5800-0AS07-0YB0
	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.

© Siemens 2020

Top Surface motion control	Option Order code: S17 Article No.: 6FC5800-0AS17-0YB0

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

© Siemens 2020

Computational motion control, Run MyCC /EMC

Option
Article No.: 6FC5800-0AN47-0YB0
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
- NCU 720.3B PN or NCU 730.3B PN

Note
Includes the nodding compensation Run MyCC /NOCO

Restricted functionality for export versions:
Not possible.

→ *Nodding compensation Run MyCC /NOCO*

© Siemens 2020

Setting the jerk filter time Run MyCC /AJET	Option Article No.: 6FC5800-0AN82-0YB0 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.

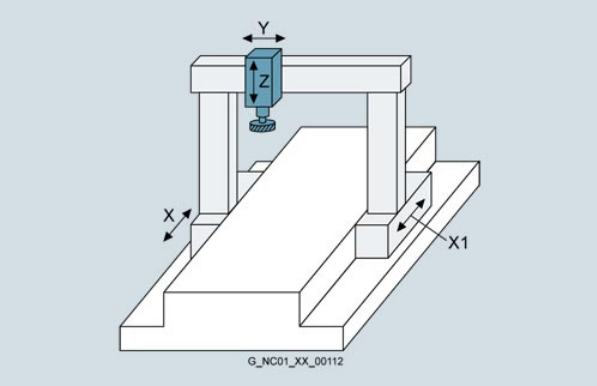
Benefits

- 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

© Siemens 2020

Pair of synchronized axes (gantry axes)

Option
Order code: M02
Article No.: 6FC5800-0AM02-0YB0



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.

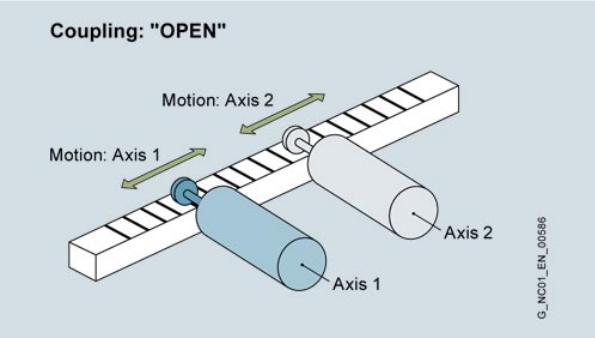
Benefits

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

© Siemens 2020

Master/slave for drives

Option
Order code: M03
Article No.: 6FC5800-0AM03-0YB0



Function

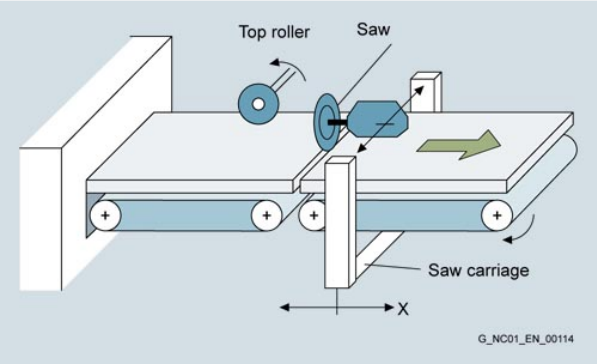
The "master/slave for drives" function is required when 2 electrical drives are mechanically linked to an axis. With this link, a torque controller ensures that both drives produce the exact same amount of torque, so that the motors do not work in opposition.
In order to attain tensioning between the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

© Siemens 2020

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.

© Siemens 2020

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 couplings*
→ *Measuring stage 2 (option M32)*
→ *Synchronous spindles, multi-edge turning*

© Siemens 2020

Virtual axis

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

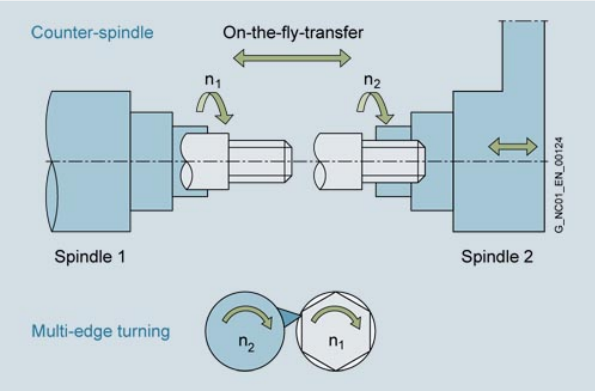
Benefits
Virtual axes are used in leading value couplings. Examples include:

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

→ *Axis/spindle*
→ *Positioning axis/auxiliary spindle*
→ *Simulation axis/spindle*

© Siemens 2020

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 couplings*

© Siemens 2020

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.

© Siemens 2020

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 couplings*

© Siemens 2020

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 couplings*

© Siemens 2020

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*

© Siemens 2020

Generic coupling CP-Standard

Function

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

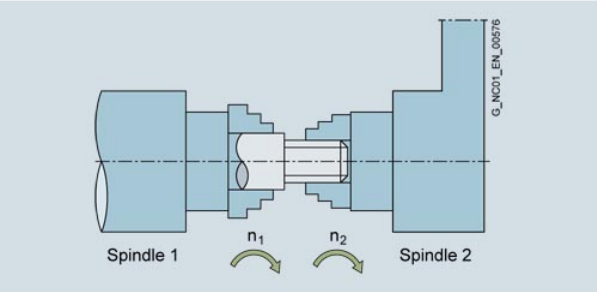
Benefits

- Coupled motion of axes

© Siemens 2020

Generic coupling CP-Static

Option
Order code: M75
Article No.: 6FC5800-0AM75-0YB0



Function
One simple synchronous spindle with coupling ratio 1:1.

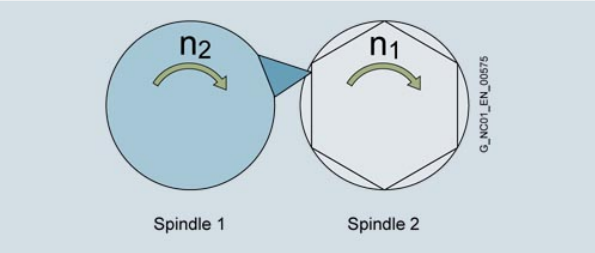
Note
- No multi-edge machining

Benefits
- Coupled motion of axes

© Siemens 2020

Generic coupling CP-Basic

Option
Order code: M72
Article No.: 6FC5800-0AM72-0YB0



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

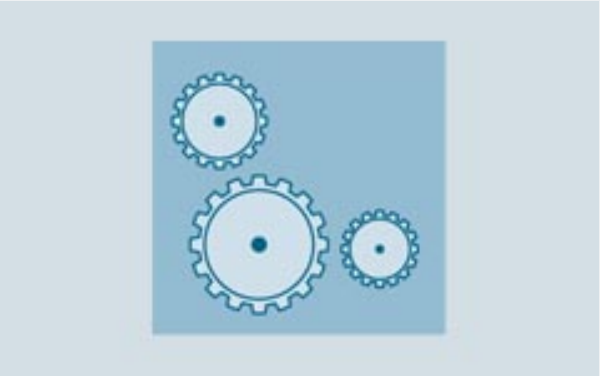
- Coupled motion of axes
- Multi-edge machining and synchronous spindle

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

© Siemens 2020

Generic coupling CP-Comfort

Option
Order code: M73
Article No.: 6FC5800-0AM73-0YB0



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*

© Siemens 2020

Generic coupling CP-Expert	**Option** Order code: M74 Article No.: 6FC5800-0AM74-0YB0

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*

© Siemens 2020

Coupling, transformation and sensor technology Run MyCC /PCTS	Option Order code: N21 Article No.: 6FC5800-0AN21-0YB0
	Function Run MyCC /PCTS is a package for coupling, transformation and sensor technology. Note The specific function Run MyCC /PCTS is not intended for general use. Restricted functionality for export versions: Not possible.

© Siemens 2020

Variable-based axis movement Run MyCC /AMOV	**Option** Order code: N62 Article No.: 6FC5800-0AN62-0YB0
Note The specific function Run MyCC /AMOV is not intended for general use. Restricted functionality for export versions: Not possible.	

© Siemens 2020

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.

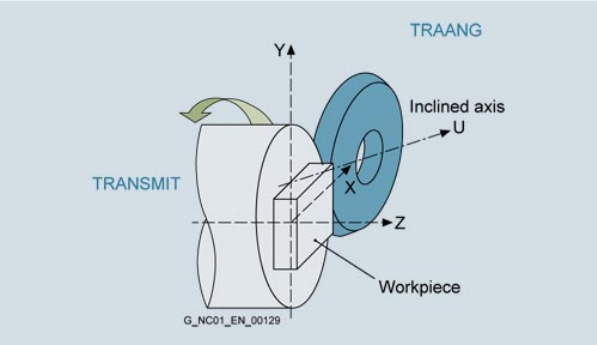
Cartesian PTP travel is also used for cylindrical grinding machines with an inclined axis: With active transformation, the infeed axis can be moved either according to Cartesian coordinates or at the angle of the inclined axis.

Benefits

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

© Siemens 2020

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

© Siemens 2020

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

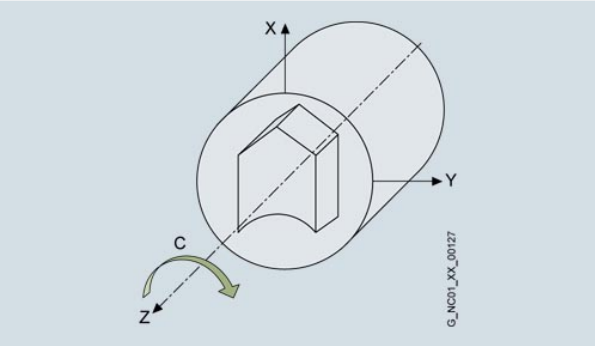
- Orientation of the tool in space can be arbitrarily defined

Restricted functionality for export versions:
Not possible.

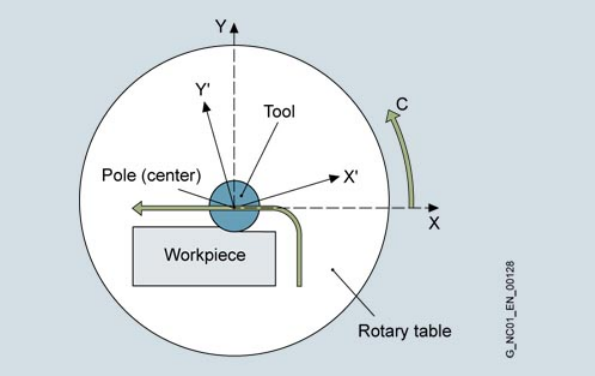
© Siemens 2020

TRANSMIT/cylinder surface transformation

Option
Order code: M27
Article No.: 6FC5800-0AM27-0YB0



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

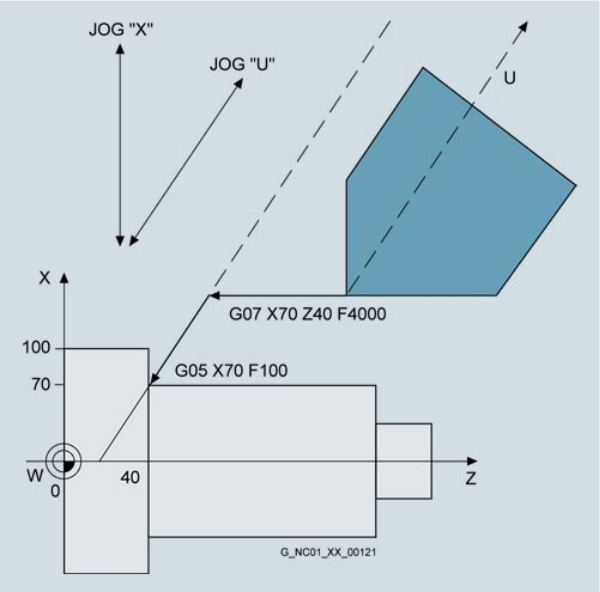
The cylinder surface transformation is used for turning machines and milling machines and allows machining of the peripheral surface, for example on turned parts.

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.

© Siemens 2020

Inclined axis

Option
Order code: M28
Article No.: 6FC5800-0AM28-0YB0



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

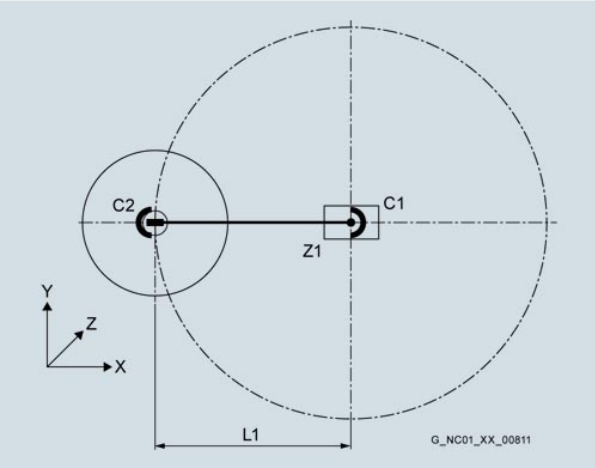
© Siemens 2020

Geometrically redundant axes Run MyCC /GRED		Option Order code: N73 Article No.: 6FC5800-0AN73-0YB0
	Note The specific function Run MyCC /GRED is not intended for general use. Restricted functionality for export versions: Not possible.	

© Siemens 2020

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

Option
Order code: N43
Article No.: 6FC5800-0AN43-0YB0



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:

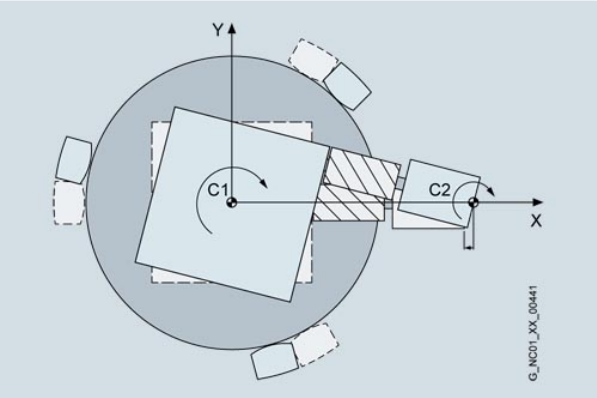
- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Transformation: DOUBLETRANSMIT Run MyCC /2TRA

Option
Order code: M25
Article No.: 6FC5800-0AM25-0YB0



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.

© Siemens 2020

Transformation: Double Generic, Run MyCC /DGEN	Option Order code: N34 Article No.: 6FC5800-0AN34-0YB0

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.

© Siemens 2020

Transformation: Dynamic swivel TRIPOD, Run MyCC /DSTT	Option Order code: M84 Article No.: 6FC5800-0AM84-0YB0

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

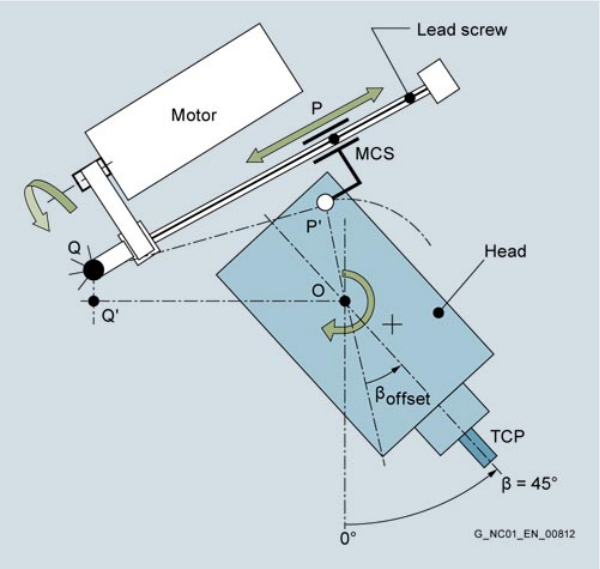
- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Transformation: Eccentric axis, Run MyCC /ECCA

Option
Order code: N44
Article No.: 6FC5800-0AN44-0YB0



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°.

Benefits

- Implementation of machines with specific kinematics

Requirement:

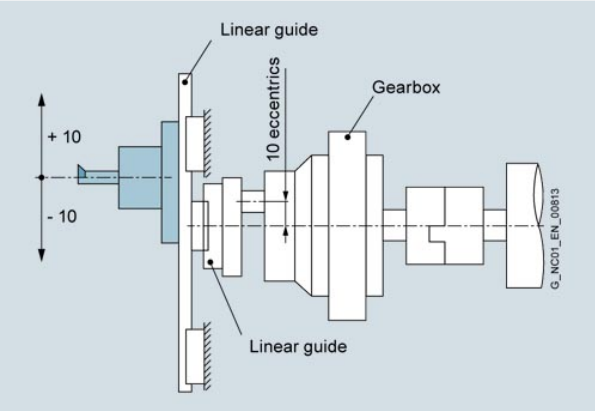
- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Transformation: Eccentric, Run MyCC /ECCE

Option
Order code: N41
Article No.: 6FC5800-0AN41-0YB0



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.

© Siemens 2020

Transformation: HEXAPOD, Run MyCC /HEX	Option Order code: M71 Article No.: 6FC5800-0AM71-0YB0

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.

© Siemens 2020

Transformation: PARACOP 3 axes, Run MyCC /PACO	Option Order code: M44 Article No.: 6FC5800-0AM44-0YB0

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)*

© Siemens 2020

Transformation: Swivel axis, Run MyCC /PIVA	Option Order code: N52 Article No.: 6FC5800-0AN52-0YB0

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:

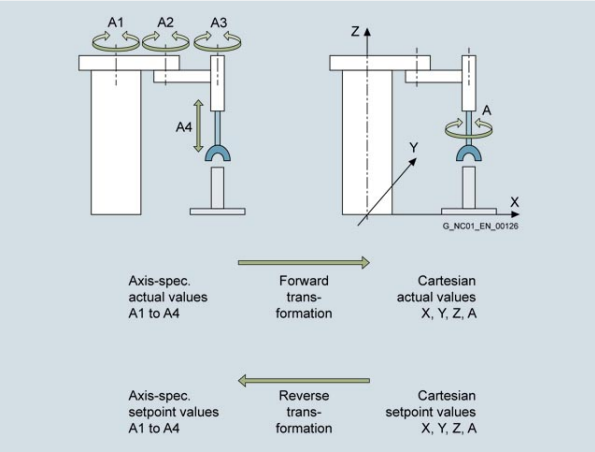
- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Transformation: Handling, Run MyCC /RCTRA

Option
Order code: M31
Article No.: 6FC5800-0AM31-0YB0



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)*

© Siemens 2020

Transformation: Robotic extended, Run MyCC /ROBX	Option Order code: N54 Article No.: 6FC5800-0AN54-0YB0

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. In order to improve accessibility to machining points on large workpieces, up to 3 additional linear axes, which move the robot or the workpiece, can be included in the transformation. The workpiece can be turned in space by an additional 3 rotary axes that are also included in the transformation.
Tool motion and orientation are still programmed with reference to the workpiece.

Benefits

- Implementation of robot kinematics with the SINUMERIK CNC

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Transformation for articulated arm robots Run MyRobot /ROBX AR

Option
Order code: R05
Article No.: 6FC5800-0AR05-0YB0

Function
Run MyRobot /ROBX AR allows a 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*

© Siemens 2020

Transformation: Rotating eccentric, Run MyCC /ROTE	Option Order code: N37 Article No.: 6FC5800-0AN37-0YB0

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.

© Siemens 2020

Transformation: Pantograph kinematics, Run MyCC /SCIS	Option Order code: M51 Article No.: 6FC5800-0AM51-0YB0

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:

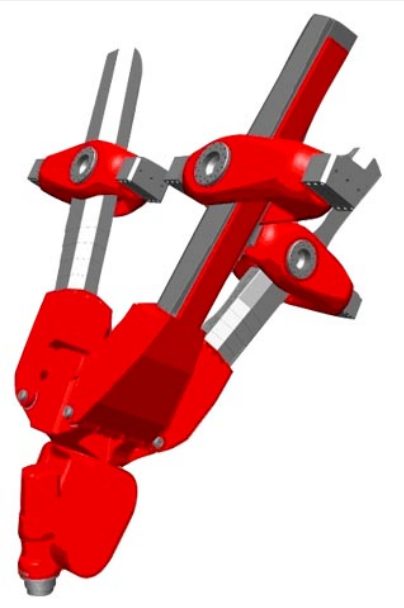
- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Transformation: TRIPOD hybrid kinematics, Run MyCC /THYK

Option
Order code: N36
Article No.: 6FC5800-0AN36-0YB0



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:

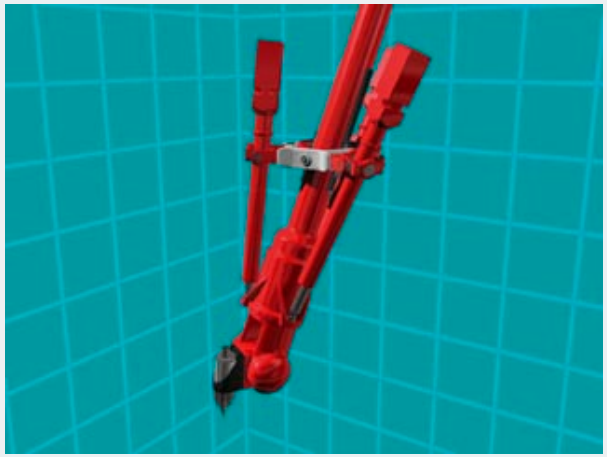
- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

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

Option
Order code: M46
Article No.: 6FC5800-0AM46-0YB0


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:

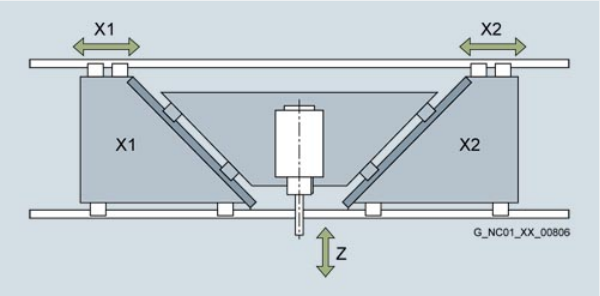
- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Transformation: Double slide, Run MyCC /SKID

Option
Order code: M80
Article No.: 6FC5800-0AM80-0YB0



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.

© Siemens 2020

Transformation: GRINDING, Run MyCC /GRIN		Option Order code: N40 Article No.: 6FC5800-0AN40-0YB0
	Function The specific function Run MyCC /GRIN is not intended for general use. Restricted functionality for export versions: Not possible.	

© Siemens 2020

Transformation: SCARA 2/3 axes, Run MyCC /SCRA		**Option** Order code: M68 Article No.: 6FC5800-0AM68-0YB0
	Note Run MyCC /SCRA is a customized transformation. Run MyCC /RCTRA represents a general solution. Restricted functionality for export versions: Not possible. → *Transformation handling Run MyCC /RCTRA (option M31)*	

© Siemens 2020

Transformation swiveling through 2 linear axes Run MyCC /SW2A

Option
Order code: N45
Article No.: 6FC5800-0AN45-0YB0

Note
The transformation swiveling through 2 linear axes Run MyCC /SW2A is a customized transformation.
Run MyCC /PACO represents a general solution.

Restricted functionality for export versions:
Not possible.

→ *Transformation PARACOP 3 axes Run MyCC /PACO (option M44)*

© Siemens 2020

Transformation: W-Z Tracking, Run MyCC /WZTR		Option Order code: N22 Article No.: 6FC5800-0AN22-0YB0
	Note The specific function Run MyCC /WZTR is not intended for general use. Restricted functionality for export versions: Not possible.	

© Siemens 2020

Transformation additional function 6th axis Run MyCC /TR-6A	**Option** Order code: M66 Article No.: 6FC5800-0AM66-0YB0
Note The specific function Run MyCC /TR-6A is not intended for general use. Restricted functionality for export versions: Not possible.	

© Siemens 2020

Transformation: Redundant axes at workpiece Run MyCC /RDCC

Option
Order code: N26
Article No.: 6FC5800-0AN26-0YB0

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)*

© Siemens 2020

Differential tower transformation, Run MyCC /DTOW	**Option** Order code: N68 Article No.: 6FC5800-0AN68-0YB0
Note The specific function Run MyCC /DTOW is not intended for general use. Restricted functionality for export versions: Not possible.	

© Siemens 2020

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

© Siemens 2020

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

© Siemens 2020

Measuring stage 2

Option
Order code: M32
Article No.: 6FC5800-0AM32-0YB0

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.

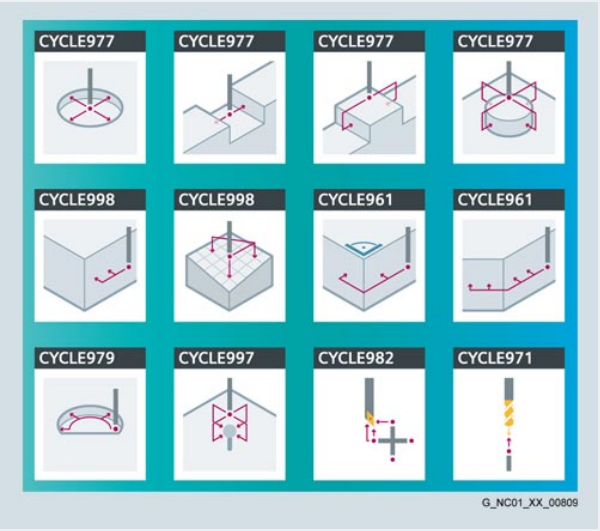
Benefits

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

© Siemens 2020

Measuring cycles for drilling/milling and turning

Option
Order code: P28
Article No.: 6FC5800-0AP28-0YB0



Measuring cycles for drilling/milling and turning

Function

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.

In order to measure tool and workpiece dimensions, a touch-trigger probe is required that supplies a constant signal (rather than a pulse) when deflected. The probe should switch without any bounce, and mechanical adjustment may be necessary.

Multidirectional probes can be used for all tool and workpiece measurements on turning and milling machines. Bidirectional probes are treated like a mono probe for workpiece measurements on milling and machining centers, but are not suitable for tool measurements.

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

© Siemens 2020

Measuring cycles for drilling/milling and turning

Option
Order code: P28
Article No.: 6FC5800-0AP28-0YB0

- 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

© Siemens 2020

Measure kinematics	Option Order code: P18 Article No.: 6FC5800-0AP18-0YB0

Function

The Measure kinematics function supports the calibration of kinematic structures of 5-axis machines. The SINUMERIK 840D sl is able to determine the parameters of kinematic transformations of the digitally or manually alignable rotary axes quickly and automatically.

Benefits

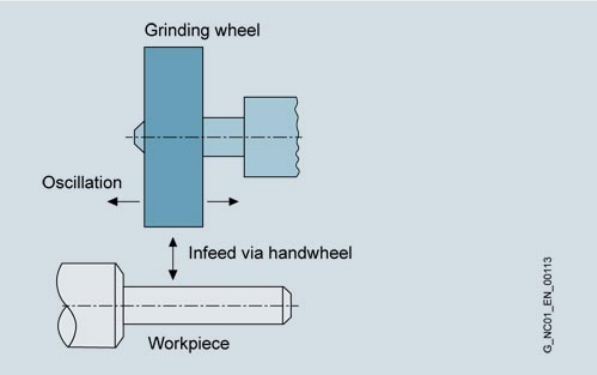
- First commissioning possible without dimensioned layout of the machine
- Regular check of the production process wherever high precision is required

© Siemens 2020

Drive current measurement Run MyCC /KPXT	**Option** Order code: M82 Article No.: 6FC5800-0AM82-0YB0
Note The specific function Run MyCC /KPXT is not intended for general use. Restricted functionality for export versions: Not possible. → *Auto Servo Tuning AST*	

© Siemens 2020

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

© Siemens 2020

Contour handwheel	Option Order code: M08 Article No.: 6FC5800-0AM08-0YB0

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)*

© Siemens 2020

Punching/nibbling	Option Order code: M33 Article No.: 6FC5800-0AM33-0YB0

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).

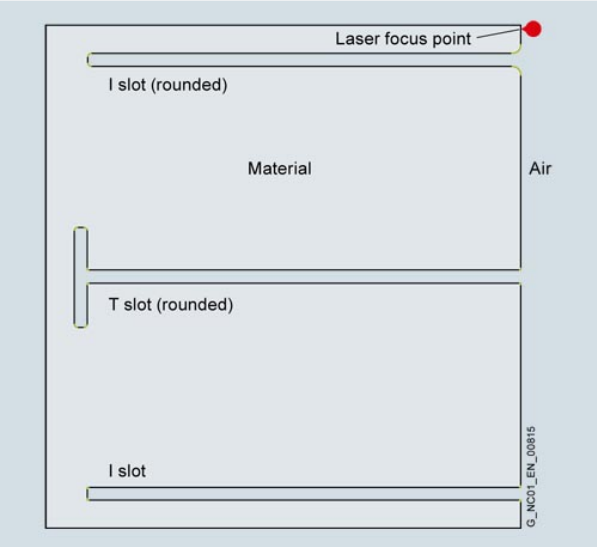
Benefits

- Implementation of machines for punching and nibbling

© Siemens 2020

Slot shape recognition

Option
Order code: S18
Article No.: 6FC5800-0AS18-0YB0



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.

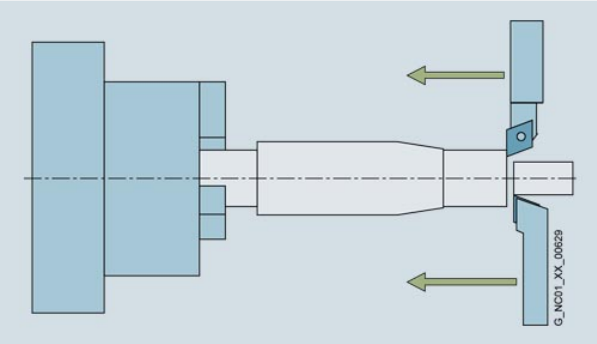
Benefits

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

© Siemens 2020

Balance cutting: 4-axis roughing

Option
Order code: S05
Article No.: 6FC5800-0AS05-0YB0



Function

Balance cutting (4-axis roughing) allows a workpiece contour to be machined with 2 tools arranged opposite to one another simultaneously.

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.

© Siemens 2020

Interpolation turning	Option Order code: P57 Article No.: 6FC5800-0AP57-0YB0

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.

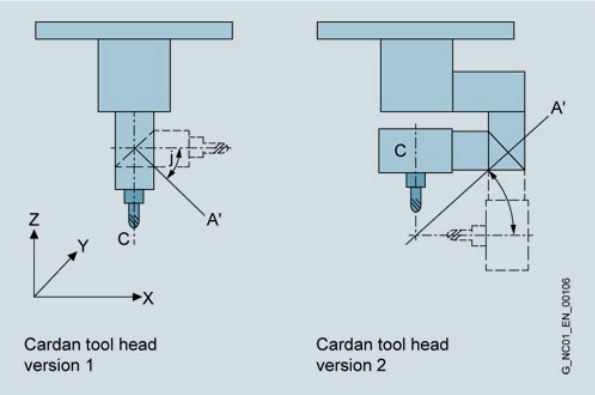
Benefits

- 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

© Siemens 2020

Machining package 5 axes

Option
Order code: M30
Article No.: 6FC5800-0AM30-0YB0



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.

Notes

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

© Siemens 2020

Machining package 5 axes	Option Order code: M30 Article No.: 6FC5800-0AM30-0YB0
	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*

© Siemens 2020

Machining package 5 axes additional function 7th axis

Option
Order code: S01
Article No.: 6FC5800-0AS01-0YB0

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.

© Siemens 2020

Milling technology package SINUMERIK MDynamics 3 axes

Option
Order code: S32
Article No.: 6FC5800-0AS32-0YB0

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
- HMI user memory additionally on the CompactFlash 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

© Siemens 2020

Milling technology package SINUMERIK MDynamics 5 axes

Option
Order code: S33
Article No.: 6FC5800-0AS33-0YB0

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
- HMI user memory additionally on the CompactFlash 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.

© Siemens 2020

Cartesian travel to fixed stop Run MyCC /FXSC	Option Order code: N38 Article No.: 6FC5800-0AN38-0YB0

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.

© Siemens 2020

Velocity adaptation for water jet cutting, Run MyCC /VADA

Option
Order code: N05
Article No.: 6FC5800-0AN05-0YB0

Function
For 2D water jet cutting applications, it is possible to reduce the path velocity at corners and along circular contours as a function of the corner angle or circle radius. CNC commands in the part program activate and deactivate the function and parameterize the velocity reduction.

Benefits
- Improved machining quality for the special water jet cutting technology

Requirement:
- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Extrapolated switching signals, Run MyCC /XOUT	Option Order code: N51 Article No.: 6FC5800-0AN51-0YB0

Function

Composite components are finding wider application, especially in the aircraft construction. The production of these components by fiber placement machines requires highly accurate output of the path-related switching positions required to cut and place fiber strands.

Run MyCC /XOUT switches up to 128 output signals as a function of the path. The outputs switch as soon as the machine axis reaches the preprogrammed switching positions.

Different modules can be used depending on the level of accuracy required:

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

Note

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

Benefits

- High-precision output of path-related switching positions

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Synchronous motion, Run MyCC /SYMO	Option Order code: N66 Article No.: 6FC5800-0AN66-0YB0

Function

Run MyCC /SYMO synchronous motion can be used to synchronize the traversing motion of 2 separate machines (e.g. 2 robots for testing components based on ultrasound transmission). In this case, each of the machines is configured in a separate channel.

The part program is executed in the master channel. The TCP and orientation of the second machine in the slave channel are coupled in real time to the motion of the machine in the master channel.

Benefits

- Synchronous operation of 2 machines with a shared Tool Center Point (TCP), relative to the respective component being tested

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Path-related pulse output, Run MyCC /PRIG	Option Order code: N76 Article No.: 6FC5800-0AN76-0YB0

Function

The number of pulses of an output signal (max. 256 kHz) supplies precise information about the traversed path. It is possible to program the required distance per pulse.
The pulses are output via the connector X520 of the SINAMICS S120 TM41 technology module.

Benefits

- Non-destructive testing of composite components using ultrasound transmission

Requirements:

- Loadable compile cycle
- SINAMICS S120 Terminal Module TM41

© Siemens 2020

Utility programs for the servo, Run MyCC /SUTI	**Option** Order code: N10 Article No.: 6FC5800-0AN10-0YB0

Function

Run MyCC /SUTI offers 3 real-time axis functions:

- Real-time modification of position control gain factor using Synchronous Action variables
- The Dual Position Feedback option dynamically uses the second encoder for position control when the speed of the axis changes
- Mixing master and slave encoders provides a Synchronous Action variable for flexible weighting of 2 encoders, which are used for closing of the position control loop of the master in a master-slave.

Each of these functions can be activated for a maximum of 4 machine axes.

Benefits

- Position controller gain factor k_v in real time influences the position of the slave encoder taking into account the master-slave speed coupling

Requirement:

- Loadable compile cycle

© Siemens 2020

Multiple feedrates in one block

Function

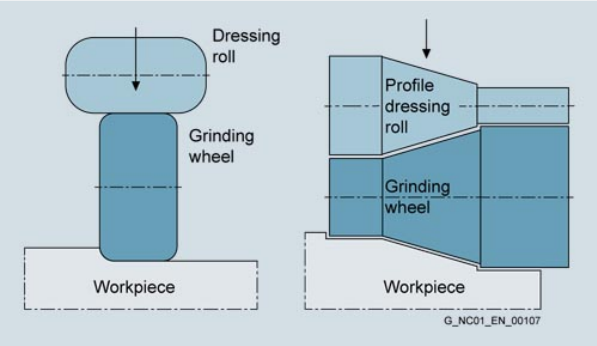
Depending on external digital and/or analog CNC inputs, this function can be used for motion-synchronous activation of up to 6 different feedrates, a dwell time, and a retraction in a single CNC block.
The input signals are combined in an input byte with a permanently assigned function. The retraction 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.

© Siemens 2020

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

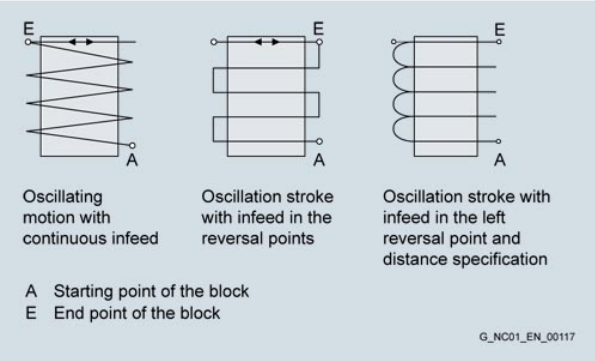
© Siemens 2020

Grinding Advanced	Option Order code: S35 Article No.: 6FC5800-0AS35-0YB0
	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

© Siemens 2020

Oscillation functions

Option
Order code: M34
Article No.: 6FC5800-0AM34-0YB0



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:
  - Cancelation 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

© Siemens 2020

Fast braking during grinding, Run MyCC /FABS	Option Order code: N81 Article No.: 6FC5800-0AN81-0YB0

Function

Run MyCC /FABS uses the edge of an input signal to decelerate the feed axis at a fast rate when grinding if the grinding wheel touches the workpiece.

The input signal can be generated by way of an acoustic emission sensor, for example. Programming of the motion sequence must be adapted for utilization of this function.

Benefits

- Fast deceleration of the feed axis during grinding

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

CCG compiler cam contour grinding

Option
Article No.: 6FC5800-0AP10-0YB0
Product ID: MCS2100

Function
This option is required to allow the execution of part programs which have been generated with the CCG compiler library. The OEM can integrate the corresponding library into the user interface on the SINUMERIK PCU/SIMATIC IPC or use it on an external PC.
The CCG compiler generates part programs in the polynomial format to allow the machining of non-circular contours on a cylindrical grinding machine. The lift curves commonly used today, which describe the desired final contour in polar coordinates, are used as the input data for programming and generating a complete part program.
The technology data, i.e. the number of infeed revolutions, allowance, sparking-out revolutions, angle of infeed and velocities, are parameterized and taken into account when the part program is generated. The generated contour is not dependent on the tool (grinding wheel) radius, because radius compensation (G41/G42) is active in the CNC.
A PC tool is included in delivery for test purposes, allowing the user to generate programs easily for test machining. This PC tool is not an integral part of the CCG compiler product.

Note
The current version of the library is provided once upon request. The option (runtime) is required for each CNC.

Benefits
- Easier programming
- Programming arbitrary closed non-circular contours
- Automatic transformation into the Cartesian coordinate system
- Retraction of a polynomial curve rather than individual points
- Diversely applicable for turning-milling, milling-turning, or grinding

Requirements:
- TRANSMIT
- Polynomial interpolation

→ *TRANSMIT/cylinder surface transformation (option M27)*
→ *Polynomial interpolation (option M18)*

© Siemens 2020

Handling package	Option Order code: S31 Article No.: 6FC5800-0AS31-0YB0

Function

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

- 3 additional axes
- 3 additional machining channels
- Transformation handling, Run MyCC /RCTRA
- Synchronized actions stage 2

Benefits

- Cost-effective option package for handling

Note

Tool offsets and spindles are not possible.

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

→ *Transformation handling Run MyCC /RCTRA (option M31)*
→ *Synchronized actions stage 2 (option M36)*

© Siemens 2020

Display external positions, Run MyCC /EXPD

Option
Order code: N64
Article No.: 6FC5800-0AN64-0YB0

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

Benefits
- Display of robot positions via CNC operator panel
- Specification of robot target position via part program

Requirement:
- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

→ *Run MyRobot /Handling*

© Siemens 2020

Run MyRobot /Handling

Option
Article No.: 6FC5800-0AP72-0YB0
Product ID: MCS1190

Function

Run MyRobot /Handling for SINUMERIK allows robots to be integrated in the SINUMERIK for the control of handing tasks without any knowledge of robots.
Operators of a machine tool are thus free to program, modify, and manage handling tasks in the familiar SINUMERIK Operate environment. Retooling operations, but also new programming tasks are thus considerably simplified and errors are avoided. No costly and time-consuming training is required.
Operation of the robots is mapped in SINUMERIK Operate:

- Programming of the traversing paths by way of cycles in the program editor
- Representation of positions (Cartesian or axis-related) in the position window
- Operation via machine control panel with override switch
- User-friendly diagnostics and status messages

Run MyRobot /Handling for SINUMERIK contains the necessary functions for communication with the robot.

Note

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

Benefits

- Increased process flexibility
- Individualized mass production
- Higher degree of cost effectiveness as a result of automatic loading/unloading
- Homogeneous production environment due to standardized solution with uniform look and feel

Requirements

- SINUMERIK CNC software version 4.7 SP2 and higher
- 1 additional machining channel (option C11)
- Run MyHMI /3GL
- Run MyScreens > 5 screens
- KUKA.mxAutomation 2.1 (from KUKA AG)

→ *Use HMI applications Run MyHMI /3GL (option P60)*
→ *Use HMI applications Run MyScreens (option P64)*

© Siemens 2020

Run MyRobot /Handling package

Option
Article No.: 6FC5800-0AP71-0YB0
Product ID: MCS1180

Function
The Run MyRobot /Handling package combines the necessary requirements for the use of Run MyRobot /Handling (formerly option P72) into a single package:

- Run MyRobot /Handling for SINUMERIK
- 1 additional machining channel (option C11)
- Run MyHMI /3GL
- Run MyScreens > 5 screens

Benefits

- Easier to order

Requirements

- SINUMERIK CNC software version 4.7 SP2 and higher
- KUKA.mxAutomation 2.1 (from KUKA AG)

→ *Run MyRobot /Handling*
→ *Use HMI applications Run MyHMI /3GL (option P60)*
→ *Use HMI applications Run MyScreens (option P64)*

© Siemens 2020

Technological functions with compressor Run MyCC /COTE	Option Order code: N50 Article No.: 6FC5800-0AN50-0YB0
	Note Run MyCC /COTE is a special function and has not been approved for general use. Restricted functionality for export versions: Not possible.

© Siemens 2020

Communication interface to a KUKA robot, Run MyCC /RODI

Option
Order code: N65
Article No.: 6FC5800-0AN65-0YB0

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

Benefits
- Communication with KUKA robots

Requirement
- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Electronic transfer CP

Option
Order code: M76
Article No.: 6FC5800-0AM76-0YB0

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 and PROFINET*

© Siemens 2020

Plastics package IME

Option
Order code: S40
Article No.: 6FC5800-0AS40-0YB0

Function
The plastics package IME comprises the functions necessary for plastics processing machines:

- 3 additional axes
- Travel to fixed stop with Force Control
- Pair of synchronized axes (gantry axes)
- Master/slave for drives
- Position switching signals/cam controller
- Polynomial interpolation
- Transformation Handling
- Synchronized actions stage 2

Note
No tool offsets or spindles can be activated.

Benefits
Facilitated ordering of all functions required for plastics processing machines.

Restricted functionality for export versions:
Not possible.

→ *3 additional axes (option A03)*
→ *Travel to fixed stop with Force Control*
→ *Pair of synchronized axes (gantry axes) (option M02)*
→ *Master/slave for drives (option M03)*
→ *Position switching signals/cam controller (option M07)*
→ *Polynomial interpolation (option M18)*
→ *Transformation handling Run MyCC /RCTRA (option M31)*
→ *Synchronized actions stage 2 (option M36)*

© Siemens 2020

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

© Siemens 2020

High-speed CNC inputs/outputs

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

Benefits

- Rapid response to special events

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

© Siemens 2020

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

© Siemens 2020

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 SYNFCT 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*

© Siemens 2020

Synchronized actions stage 2	Option Order code: M36 Article No.: 6FC5800-0AM36-0YB0

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).

© Siemens 2020

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.

© Siemens 2020

Analog value control in the interpolation cycle

Function

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

Benefits

- Implementation of machine-specific functions by the machine manufacturer

Requirement:

- Analog input

© Siemens 2020

Path velocity-dependent analog output (laser power control)	Option Order code: M37 Article No.: 6FC5800-0AM37-0YB0

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

© Siemens 2020

High-speed laser switching signal, Run MyCC /HSLC	Option Order code: M38 Article No.: 6FC5800-0AM38-0YB0

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

© Siemens 2020

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

© Siemens 2020

1D/3D clearance control in position control cycle, Run MyCC /CLC	Option Order code: M40 Article No.: 6FC5800-0AM40-0YB0

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.

© Siemens 2020

Clearance control 1D/3D in position control cycle with free direction Run MyCC /CLC-FD	Option Order code: M65 Article No.: 6FC5800-0AM65-0YB0

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.

© Siemens 2020

Clearance control Run MyCC /CLCX	**Option** Order code: N78 Article No.: 6FC5800-0AN78-0YB0

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

© Siemens 2020

Evaluation of internal drive variables

Option
Order code: M41
Article No.: 6FC5800-0AM41-0YB0
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

© Siemens 2020

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)

© Siemens 2020

Interrupt routines with fast retraction from the contour

Option
Order code: M42
Article No.: 6FC5800-0AM42-0YB0

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 the fast retraction.

Benefits

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

© Siemens 2020

Cross-mode actions ASUB and synchronized actions	Option Order code: M43 Article No.: 6FC5800-0AM43-0YB0

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)*

© Siemens 2020

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

© Siemens 2020

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*

© Siemens 2020

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

© Siemens 2020

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/IPC 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.

© Siemens 2020

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

© Siemens 2020

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

© Siemens 2020

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

© Siemens 2020

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

© Siemens 2020

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.

© Siemens 2020

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

© Siemens 2020

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.

© Siemens 2020

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.

© Siemens 2020

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

© Siemens 2020

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.

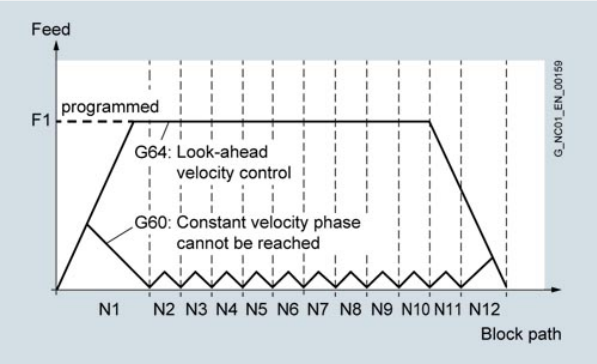
The preprocessing memory is constantly reloaded during execution. Block execution can be interrupted with the STARTFIFO command until the preprocessing memory has been filled, or STOPFIFO (start high-speed machining section) or STOPRE (stop preprocessor) can be programmed.

Benefits

- Increased productivity due to fast execution

© Siemens 2020

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*

© Siemens 2020

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

© Siemens 2020

Frame concept

Benefits

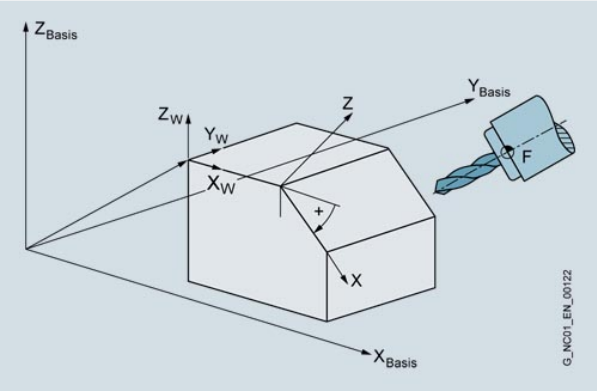
The frame concept makes it possible to transform Cartesian coordinate systems very simply by offsetting, rotating, scaling and mirroring.

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

© Siemens 2020

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*

© Siemens 2020

Axis/spindle interchange

Function

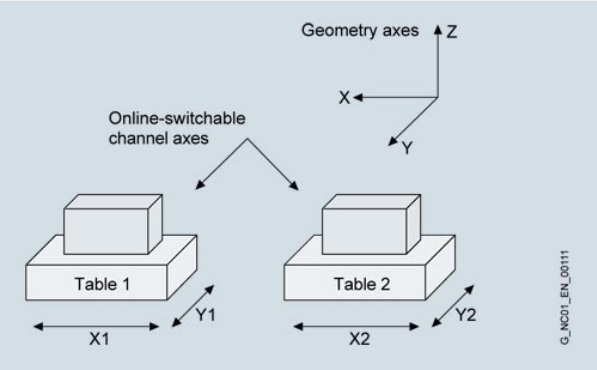
An axis/a spindle is permanently assigned to a specific channel via machine data. The axis/spindle replacement function can be used to release an axis/a spindle (RELEASE) and to assign it to another channel (GET), i.e. to replace the axis/spindle. The relevant axes/spindles are determined via machine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

Benefits

- Implementation of special machine tools

© Siemens 2020

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.

© Siemens 2020

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.

© Siemens 2020

Online ISO dialect interpreter

Function

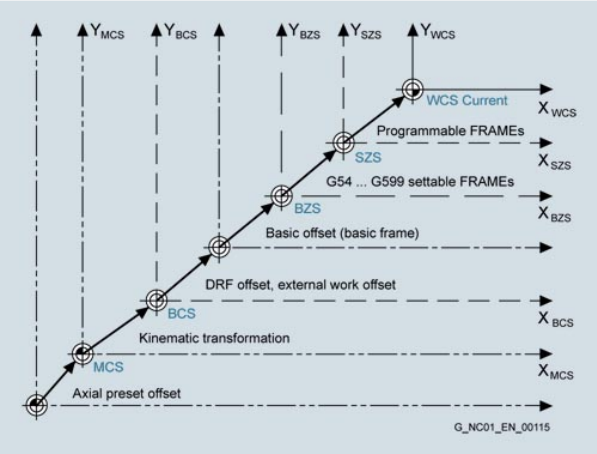
In general, part programs for SINUMERIK CNCs are programmed according to DIN 66025 and relevant expansions. Part programs created according to the ISO standard (e.g., G codes from other manufacturers) can be read in, edited and executed on SINUMERIK CNCs using the online ISO dialect interpreter.

Benefits

- Part programs from other manufacturers can be used

© Siemens 2020

Work offsets



Function

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.

© Siemens 2020

Work offsets

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].

Benefits

- Easier programming of workpieces

→ *Frame concept*

© Siemens 2020

Scratching, determining work offset

Function

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

© Siemens 2020

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*

© Siemens 2020

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

© Siemens 2020

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)	$\pm(10^{-300} \ldots 10^{+300})$
BOOL	Boolean values: TRUE (1) and FALSE (0)	1.0
CHAR	ASCII character specified by the code	0 ... 255
STRING	Character string, number of characters in [...], maximum of 200 characters	Sequence of values with 0 ... 255
AXIS	Axis names (axis addresses) only	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.

© Siemens 2020

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

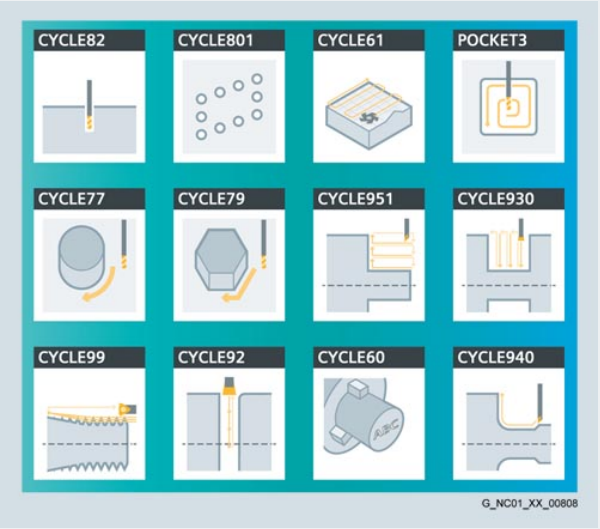
The auxiliary function output may be carried out either with velocity reduction and PLC acknowledgement up to the next block, or before and during travel without velocity reduction and without block change delay. Following blocks are then traversed without a time-out.

Benefits

- Timely coordination between part program and PLC

© Siemens 2020

Technology cycles



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

© Siemens 2020

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*

© Siemens 2020

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

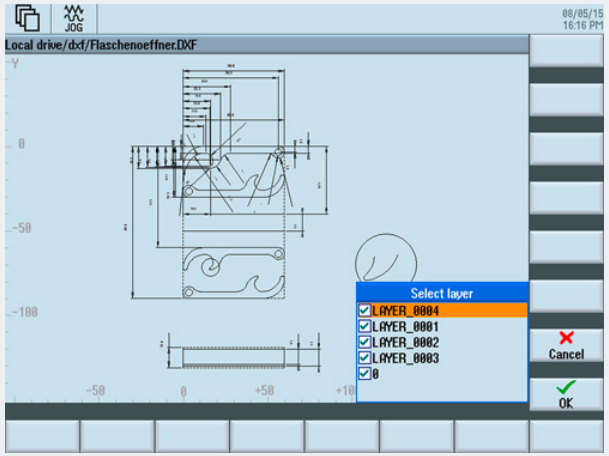
- Help for fast programming of part programs
- Dynamic programming graphics
- Animated elements

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

© Siemens 2020

DXF Reader

Option
Article No.: 6FC5800-0AP56-0YB0
Product ID: MCS4130



Function

The DXF Reader allows DFX 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

© Siemens 2020

Manual machine

Function

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.

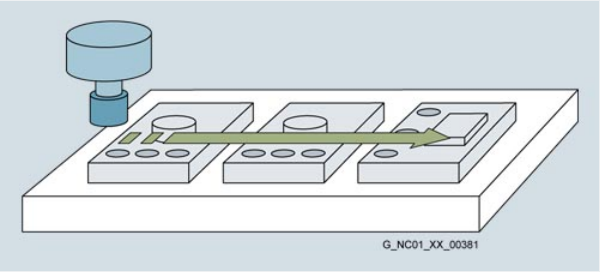
Benefits

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)*

© Siemens 2020

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

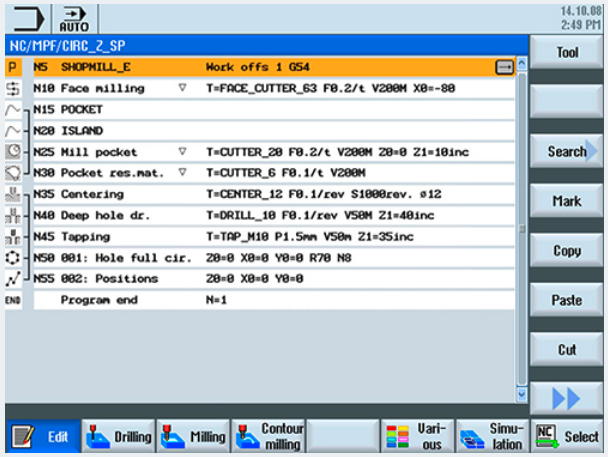
- Time-saving production of identical and different workpieces
- Idle times are reduced due to fewer tool changes because a tool performs as many machining operations as possible in all clampings before the next tool change is initiated.

→ *ShopTurn/ShopMill (option P17)*

© Siemens 2020

ShopTurn/ShopMill

Option
Order code: P17
Article No.: 6FC5800-0AP17-0YB0



ShopMill: Machining step programming

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

© Siemens 2020

programSYNC

Option
Order code: P05
Article No.: 6FC5800-0AP05-0YB0



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.

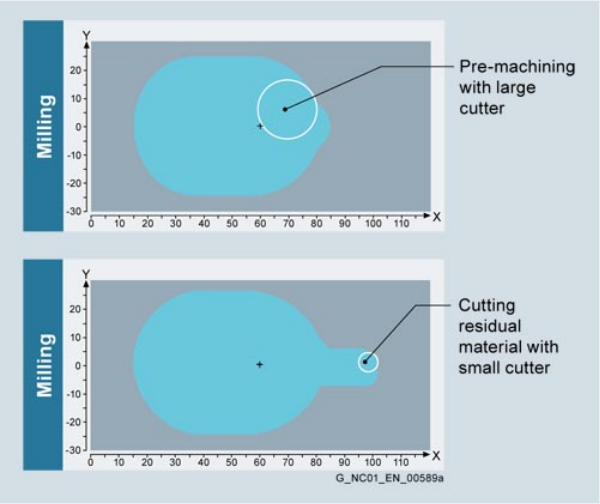
Benefits

- Creation of time-optimized programs

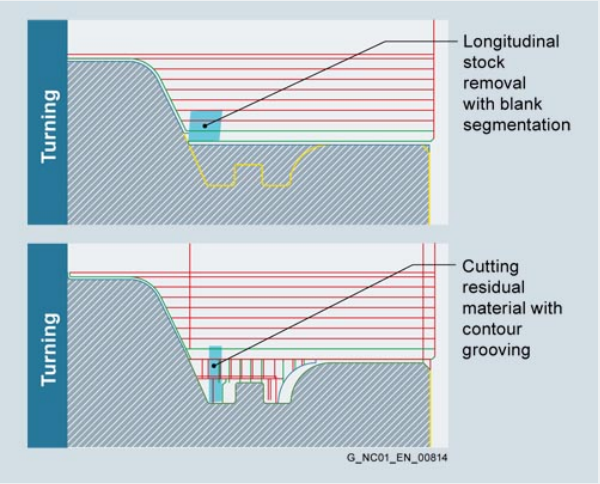
© Siemens 2020

Residual material detection and machining for contour pockets and stock removal

Option
Order code: P13
Article No.: 6FC5800-0AP13-0YB0



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

© Siemens 2020

Access protection for cycles Lock MyCycles	Option Order code: P54 Article No.: 6FC5800-0AP54-0YB0 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

© Siemens 2020

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.

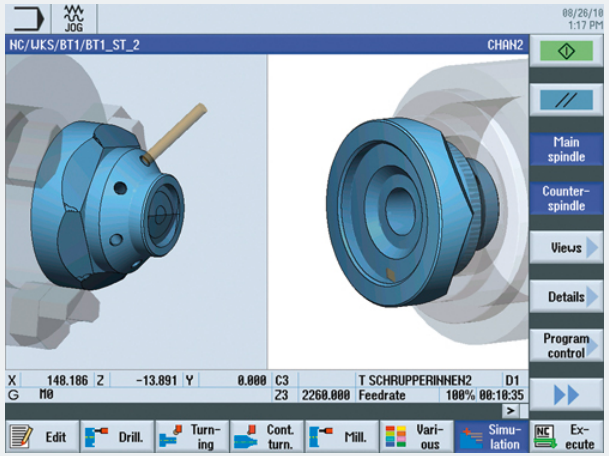
This software protects technological knowledge against unauthorized access and modification. It prevents the reverse engineering of machines, thus safeguarding the investments made.

Benefits

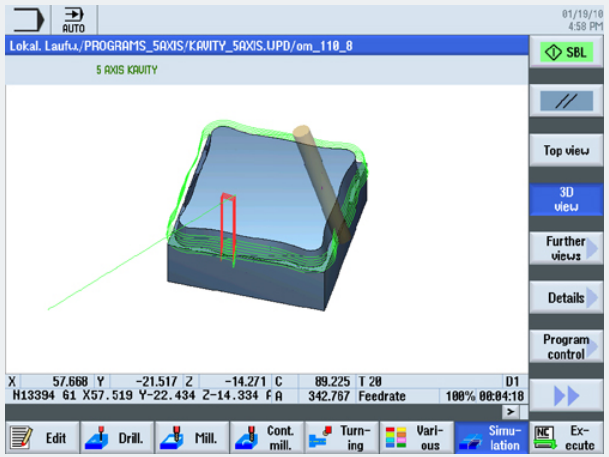
- Internal know-how can be protected

© Siemens 2020

Simulation SINUMERIK Operate



SINUMERIK Operate: Turning/drilling simulation



SINUMERIK Operate: Milling simulation

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.

Notes:

- Very fast simulation result, but consequently low dynamic resolution
- Fine recording is too slow for complex displays/fine recording fails to run for very complex parts
- Quickview for mold-making part programs
- Simulation of special kinematics not available when compile cycles are used
- Blank CAD data cannot be imported, e.g. for castings

© Siemens 2020

Simulation SINUMERIK Operate

- 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

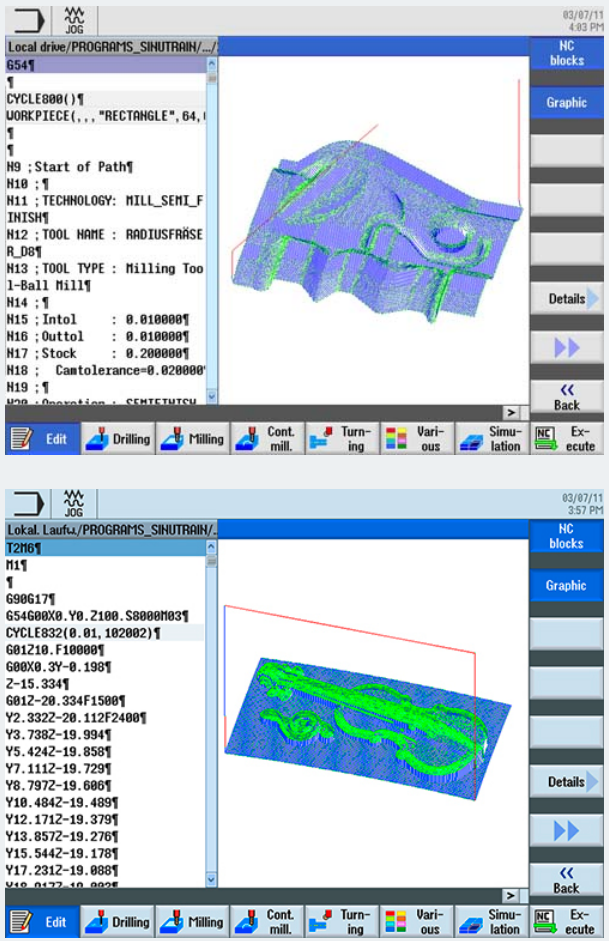
Benefits

- Program control without machining

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

© Siemens 2020

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

© Siemens 2020

3D simulation 1 (finished part)

Option
Order code: P25
Article No.: 6FC5800-0AP25-0YB0

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

© Siemens 2020

Simultaneous recording (real-time simulation of the current machining operation)	**Option** Order code: P22 Article No.: 6FC5800-0AP22-0YB0

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*

© Siemens 2020

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*

© Siemens 2020

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

© Siemens 2020

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

© Siemens 2020

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.

© Siemens 2020

Execution from external storage EES

Option
Order code: P75
Article No.: 6FC5800-0AP75-0YB0
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.

© Siemens 2020

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*

© Siemens 2020

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:

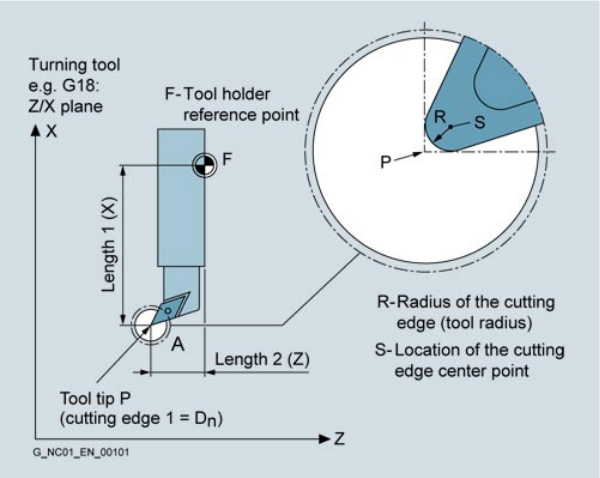
- With calculation at the contour
  During the block search, the same calculations are executed as during normal program operation. The target block is then traversed true-to-contour until the end position is reached. Using this function it is possible to approach the contour again from any situation.
- 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

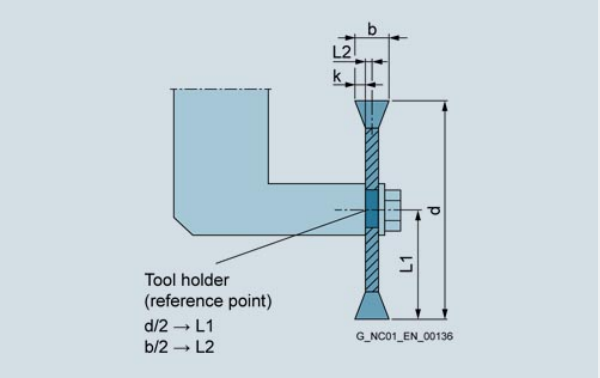
- Testing of part programs or restart following a cancellation in machining

© Siemens 2020

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

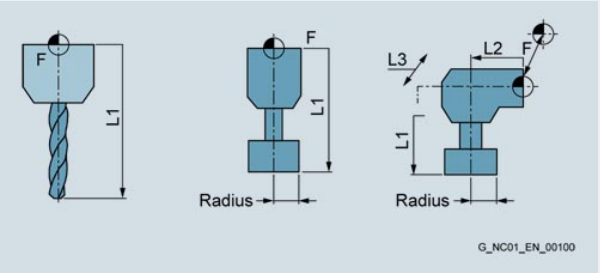
Group 1xy	Milling tools	From spherical head cutters to bevel cutters
Group 2xy	Drills	From twist drills to reamers
Group 4xy	Grinding tools	From surface grinding wheels to dressers
Group 5xy	Turning tools	From roughing tools to threading tools
Group 700	Slotting saws	Wood technology
Group 900	Special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

Benefits

The tool types define how the geometry specifications are computed.

© Siemens 2020

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.

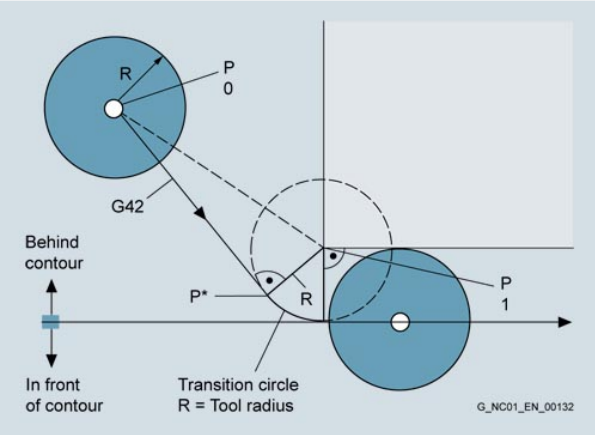
Tool offset D always has a reference to tool number T when Siemens tool management is active, e.g. with monitoring functions and management of sister tools.

Benefits

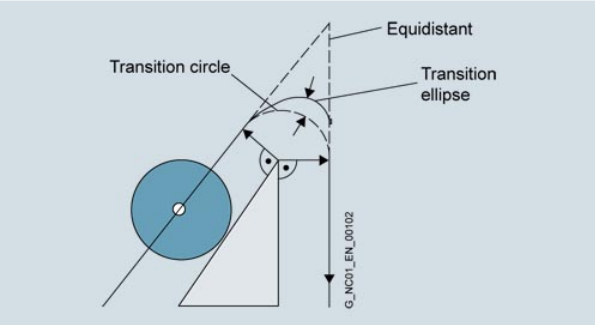
- Easier programming, as the tools are only taken into consideration upon execution

© Siemens 2020

Tool radius compensation



KONT 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.

KONT

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

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

You can implement various strategies spatially or in the plane to achieve 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*

© Siemens 2020

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*

© Siemens 2020

3D tool radius compensation	Option Order code: M48 Article No.: 6FC5800-0AM48-0YB0

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

© Siemens 2020

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*

© Siemens 2020

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.

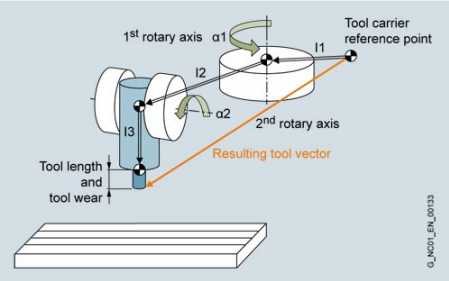
In turret magazines on turning machines, the T command carries out the entire tool change, that is, locates and inserts the tool. The tool change mode can be set using machine data.

Benefits

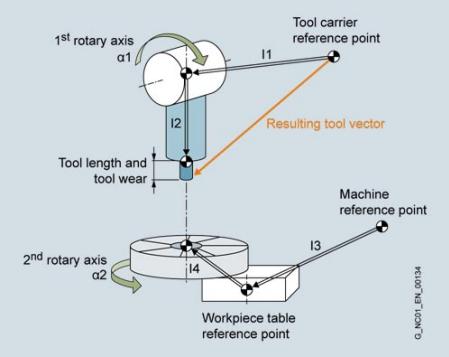
- Adapting the tool change to the machine

© Siemens 2020

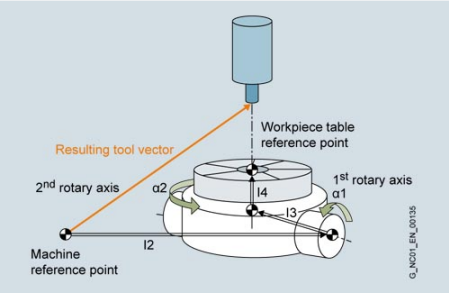
Tool carrier with orientation capability



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 α1 and α2 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:

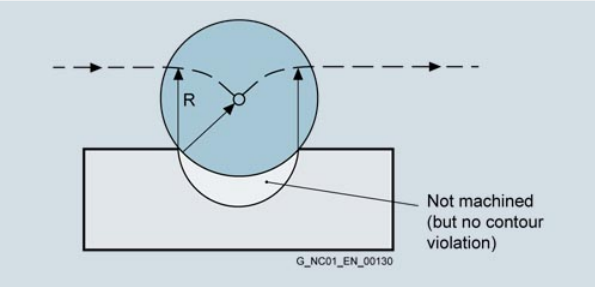
- Rotatable tool type T (tool)
- Rotatable tool/rotatable workpiece table type M (mixed)
- Rotatable workpiece table type P (part)

Benefits

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

© Siemens 2020

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

© Siemens 2020

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

© Siemens 2020

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)*

© Siemens 2020

Orientation offset static/dynamic	Option Order code: S14 Article No.: 6FC5800-0AS14-0YB0

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

© Siemens 2020

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.

© Siemens 2020

Tool management with more than 4 magazines		Option Order code: M88 Article No.: 6FC5800-0AM88-0YB0
	Function Four magazines are included in the basic version of SINUMERIK 840D sl: • Magazine • Spindle • Tool buffer Benefits • The number of magazines is increased	

© Siemens 2020

Monitoring for maximum tool speed/acceleration	Option Order code: S08 Article No.: 6FC5800-0AS08-0YB0

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

© Siemens 2020

Identify tool demand

Option
Order code: M77
Article No.: 6FC5800-0AM77-0YB0

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

© Siemens 2020

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*

© Siemens 2020

Tool Ident Connection

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

© Siemens 2020

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

© Siemens 2020

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*

© Siemens 2020

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 CBA (component based automation) for networking distributed plants, component engineering
- 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 CBA*
→ *PROFINET IO*
→ *www.siemens.com/profinet*

© Siemens 2020

PROFINET CBA

Function

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

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

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

In this context, "engineering" means:

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

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

A PROFINET component always consists of:

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

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

Benefits

- Automation concept for plants with distributed intelligence

© Siemens 2020

PROFINET IO

Function

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. ET200 distributed I/O or a SINAMICS drive are examples of PROFINET IO devices. When compared to PROFIBUS, an IO device is a slave.
- An IO Supervisor is typically a programming device, a PC or an HMI unit for commissioning or diagnostics. It features an engineering tool, which can be used to parameterize and diagnose individual IO devices. When compared to PROFINET, this would be a class 2 master in terms of function.

PROFINET IO provides protocol definitions for the following functions:

- 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

© Siemens 2020

Axis data output via PROFIBUS Run MyCC /ADAS

Option
Article No.: 6FC5800-0AN07-0YB0
Product ID: MCS3110

Function
Run MyCC /ADAS enables axis and spindle data to be output to a special PROFIBUS or PROFINET device module.
The required axis and signal type are selected by transferring a selection command (length: 8 bytes) to the CNC. Run MyCC /ADAS in the CNC kernel then sends up to 30 axis data items (of 4 bytes each) to the slave in each PROFIBUS cycle.
The transfer cycle can be equal to the position controller cycle, or a multiple thereof. Consistent receipt of data from each cycle can be guaranteed only when the slave is operating in isochronous mode (or with PROFINET IRT).

Benefits
- Process or machine monitoring functions in real time outside the CNC

Requirement:
- Loadable compile cycle

© Siemens 2020

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

Option
Order code: N61
Article No.: 6FC5800-0AN61-0YB0

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

© Siemens 2020

Master computer connection Access MyMachine /OPC UA

Option
Order code: P67
Article No.: 6FC5800-0AP67-0YB0

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*

© Siemens 2020

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.

© Siemens 2020

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.

© Siemens 2020

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.

© Siemens 2020

SINUMERIK extended touch	Option Order code: P80 Article No.: 6FC5800-0AP80-0YB0

Function

The SINUMERIK extended touch function expands SINUMERIK Operate to include multi-touch operation. Siemens operator panels for SINUMERIK do not require this option.

Note

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

Benefits

- Multi-touch operation in SINUMERIK Operate

© Siemens 2020

SINUMERIK Operate Display Manager	Option Order code: P81 Article No.: 6FC5800-0AP81-0YB0

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*

© Siemens 2020

SINUMERIK Operate /NCU	Option Order code: S00 Article No.: 6FC5800-0AS00-0YB0

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*

© Siemens 2020

SINUMERIK Operate /universal client	Option Order code: S87 Article No.: 6FC5800-0AS87-0YB0

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*

© Siemens 2020

SINUMERIK Operate /PCU	Option Order code: P88 Article No.: 6FC5800-0AP88-0YB0
	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*

© Siemens 2020

SINUMERIK PCU base software /IPC	Option Order code: P86 Article No.: 6FC5800-0AP86-0YB0

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

© Siemens 2020

SINUMERIK Operate /PC	Option Order code: P87 Article No.: 6FC5800-0AP87-0YB0

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*

© Siemens 2020

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

	NCU CNC SW	**SINUMERIK PCU or SIMATIC IPC for SINUMERIK SINUMERIK Operate**
Operator panel management		
2/4 TCUs with interlocking of simultaneous operation by Veto mode and PLC (4 × T : 1 × M)	✔ (2/4)	✔ (4)
TCU suppression, with more than 2/4 TCUs (*n* × T : 1 × M)	✔	✔
External HMIs (*n* × M : *n* × N)	–	✔ (1 × M : 4 × N)
One external HMI which can be switched over via several NCUs (1 × M : *n* × N)	–	✔ (4 × N)
One internal and one external HMI connected simultaneously to one NCU	✔ Tool management	✔ Tool management
Display		
Down zoom	–	–
Dynamic resolution switchover	✔	✔

© Siemens 2020

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/SIMATIC IPC for SINUMERIK
- Distances between SINUMERIK PCU, SIMATIC IPC and operator panel of up to 100 m (maximum distance between two network nodes)

© Siemens 2020

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.

© Siemens 2020

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

© Siemens 2020

Electronic Key System EKS	Option Order code: P53 Article No.: 6FC5800-0AP53-0YB0

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

© Siemens 2020

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*

© Siemens 2020

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

© Siemens 2020

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

© Siemens 2020

Operating software languages	Option Order code: N00 Article No.: 6FC5800-0AN00-0YB0
	Function The operating software supports the following languages as standard for user interface display texts: English, French, German, Italian, Simplified Chinese, and Spanish. Further languages such as Bulgarian, Chinese Traditional, Croatian, Czech, Danish, Dutch, Finnish, Greek, Hindi, Hungarian, Indonesian, Japanese, Korean, Malaysian, Polish, Portuguese/Brazilian, Romanian, Russian, Swedish, Slovakian, Slovenian, Tamil, Thai, Turkish, Vietnamese can be activated with the option N00. Benefits • Easier interaction with the operator interface by the operator

© Siemens 2020

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.

© Siemens 2020

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

© Siemens 2020

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

© Siemens 2020

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 (±) 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.

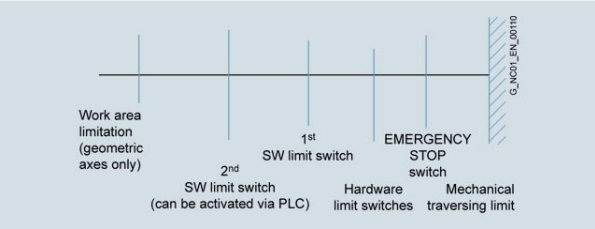
Benefits

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*

© Siemens 2020

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

© Siemens 2020

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

© Siemens 2020

Standstill monitoring

Function

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*

© Siemens 2020

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.

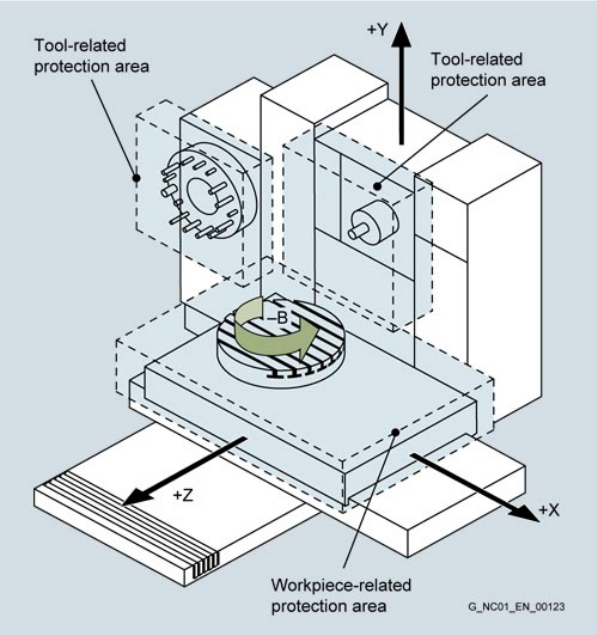
Benefits

- Monitors adherence to the parameterized clamping tolerance

→ Position monitoring
→ Standstill monitoring

© Siemens 2020

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

© Siemens 2020

Metal spinning protection area Run MyCC /MSPZ		Option Order code: N42 Article No.: 6FC5800-0AN42-0YB0
	Note The specific function Run MyCC /MSPZ is not intended for general use. Restricted functionality for export versions: Not possible.	

© Siemens 2020

Path length evaluation

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

© Siemens 2020

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*

© Siemens 2020

Contour monitoring with tunnel function	Option Order code: M52 Article No.: 6FC5800-0AM52-0YB0

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

© Siemens 2020

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*

© Siemens 2020

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*

© Siemens 2020

Integrated ISM spindle monitor (S-Monitor)	Option Order code: P55 Article No.: 6FC5800-0AP55-0YB0

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

© Siemens 2020

Collision avoidance Protect MyMachine /3D Primitives

Option
Article No.: 6FC5800-0AS03-0YB0
Product ID: MCS4120

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 4.8 SP1 and higher
- NCU 710.3B (NCU 720.3B or NCU 730.3B recommended)
- Simultaneous recording for graphical visualization

→ *Collision avoidance Protect MyMachine /3D STL*
→ *Collision avoidance Protect MyMachine /Open*
→ *Simultaneous recording (option P22)*

© Siemens 2020

Collision avoidance Protect MyMachine /3D STL

Option
Article No.: 6FC5800-0AS02-0YB0
Product ID: MCS4100

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 4.5 SP2 and higher
- 1-channel machine with one NCU
- SINUMERIK NCU 720.3B PN (recommendation: NCU 730.3B PN)
- Simultaneous recording for graphical visualization

→ *Collision avoidance Protect MyMachine /3D Primitives*
→ *Collision avoidance Protect MyMachine /Open*
→ *Simultaneous recording (option P22)*

© Siemens 2020

Collision avoidance Protect MyMachine /Open

Option
Article No.: 6FC5800-0AS04-0YB0
Product ID: MCS4110

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 IPC 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 4.8 SP2 and higher
- SIMATIC IPC427E / IPC627E
- ModuleWorks package (ModuleWorks GmbH)
- OPC UA

→ *Collision avoidance Protect MyMachine /3D Primitives*
→ *Collision avoidance Protect MyMachine /3D STL*

© Siemens 2020

Collision protection Protect MyMachine /CC Axis Monitor

Option
Article No.: 6FC5800-0AN06-0YB0
Product ID: MCS4140

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.

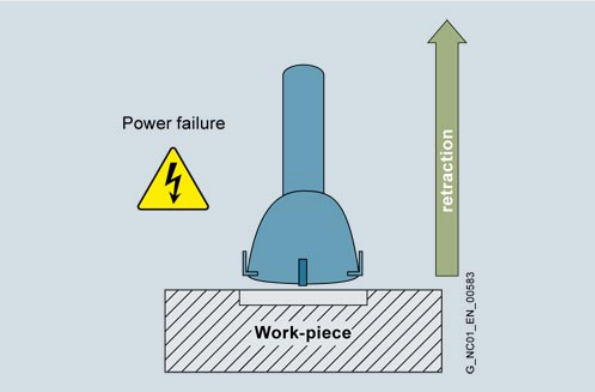
© Siemens 2020

Geometry monitoring interface Run MyCCI /GPM	Option Order code: M83 Article No.: 6FC5800-0AM83-0YB0
	Note Run MyCC /GPM is a specific function and is not intended for general use. Restricted functionality for export versions: Not possible. → *Collision avoidance Protect MyMachine /3D STL*

© Siemens 2020

Extended stop and retract ESR, drive-autonomous

Option
Order code: M60
Article No.: 6FC5800-0AM60-0YB0



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

© Siemens 2020

Stop and retract ESR, extended (CNC-controlled and drive-autonomous)	Option Order code: M61 Article No.: 6FC5800-0AM61-0YB0

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

© Siemens 2020

Integrated tool monitoring and diagnostics IMD

Function

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

Benefits

- Monitors tools
- Fast response to faults
- Improves productivity

© Siemens 2020

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

Option
Article No.: 6FC5800-0AN12-0YB0
Product ID: MCS3120

Function
Run MyCC /IMD-L supports a "missing tool monitoring" function within the IMD package. For this purpose, cuts (transition from rapid traverse to creep feed) must be selected and parameterized (CC_START_TASK ("MissingTool", ..., ...)) in the part program. For example, select "torque of spindle" or "torque of a specific axis". The torque can be learned in an initial program run (teach-in).
In productive operation, the tool is assumed to be present when the actual signal is equal to the taught signal. If not, an individually parameterized alarm response is triggered (RESET, CANCEL, Stop Spindle, Stop Axes, etc.).

Benefits
- Missing tool monitoring
- Fast response to faults
- Improves productivity

Requirement:
- Loadable compile cycle

© Siemens 2020

Integrated tool monitoring and diagnostics IMD BASE Run MyCC /IMD

Option
Article No.: 6FC5800-0AN13-0YB0
Product ID: MCS3130

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

Benefits
- Missing tool monitoring
- Programming of a permissible overload limit for a tool
- Fast response to faults
- Improves productivity

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

Requirement:
- Loadable compile cycle

© Siemens 2020

PROFIBUS tool and process monitoring Run MyCC /TPM-PB

Option
Article No.: 6FC5800-0AM62-0YB0
Product ID: MCS5120

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

Benefits
- Detect errors before they occur
- Active power monitoring keeps an eye on such things as breakage, wear and missing tools
- Precise operating status recognition and process optimization

Note
The hardware and software for evaluation are offered, for example, by the Product Partner SINUMERIK Systems.

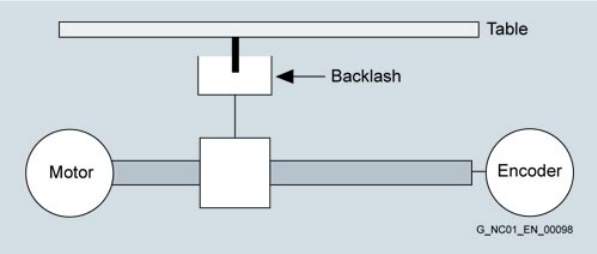
Requirement:
- Loadable compile cycle

© Siemens 2020

Integrated process monitoring Run MyCC /IPM	Option Order code: M70 Article No.: 6FC5800-0AM70-0YB0
Function Run MyCC /IPM is a specific function and is not intended for general use. Restricted functionality for export versions: Not possible.	

© Siemens 2020

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

© Siemens 2020

Leadscrew/measuring system error compensation

Function

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

© Siemens 2020

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.

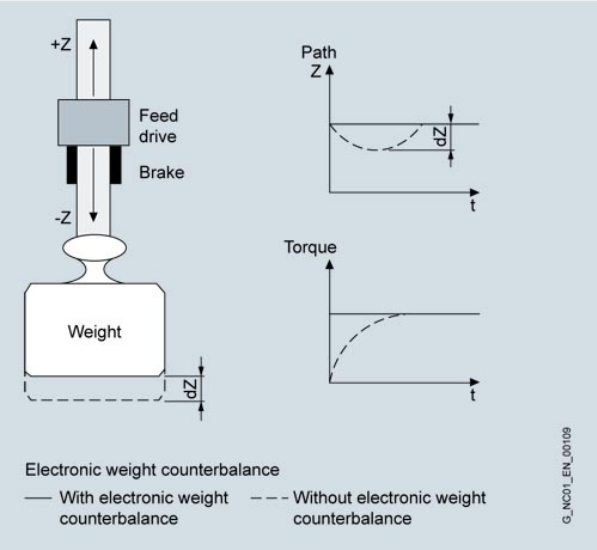
Benefits

- 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

© Siemens 2020

Weight counterbalance, electronic

Basic function of SINAMICS S120



Electronic weight counterbalance

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

© Siemens 2020

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.

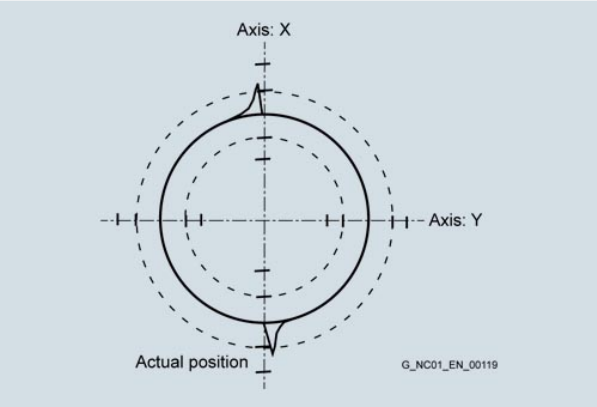
Abrupt changes in these parameters are automatically smoothed by the SINUMERIK CNC system in order to prevent machine overload and avoid inadvertent triggering of monitoring functions.

Benefits

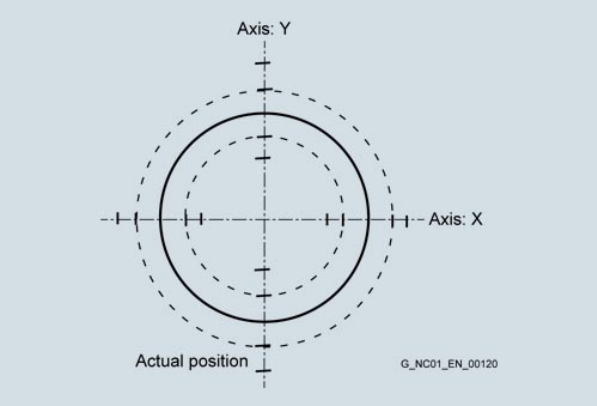
- Compensation of thermal expansion of machine parts
- Consistent manufacturing accuracy of machined workpieces

© Siemens 2020

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)*

© Siemens 2020

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.

© Siemens 2020

Friction compensation with adaptive characteristics	Option Order code: S06 Article No.: 6FC5800-0AS06-0YB0

Function

This function provides optimized friction compensation. The required setting parameters are calculated automatically.

Note:

- Replaces Run MyCC /FRIC (option N67)

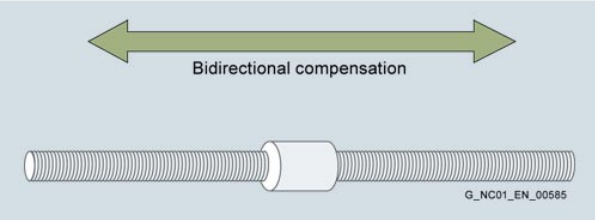
Benefits

- Friction-dependent path deviations are eliminated even more effectively
- Consistently high contour accuracy and precision on the workpiece
- Simplified optimization

© Siemens 2020

Leadscrew error compensation, bidirectional

Option
Order code: M54
Article No.: 6FC5800-0AM54-0YB0



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

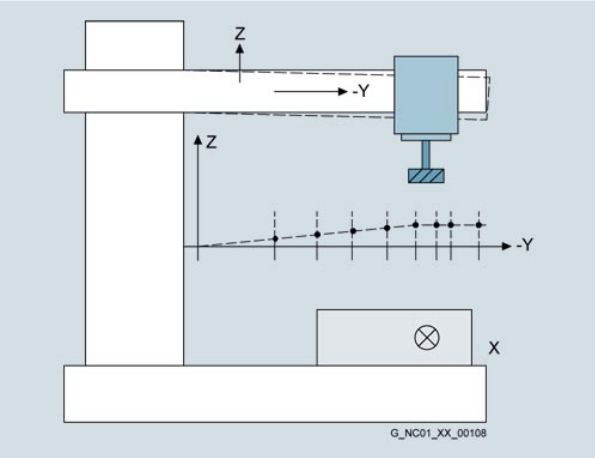
- Correction of mechanical spindle inaccuracies
- More precise manufacturing

Restricted functionality for export versions:
The correctable tolerance band is restricted to 1 mm (0.039 in).

© Siemens 2020

Sag compensation, multi-dimensional

Option
Order code: M55
Article No.: 6FC5800-0AM55-0YB0



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 × (number of $axes_{max}$)
- 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).

© Siemens 2020

Nodding compensation Run MyCC /NOCO

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

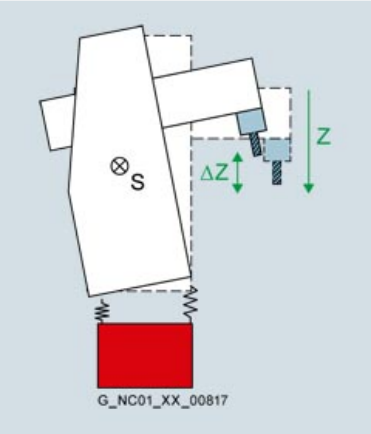
Requirement:
- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Nodding compensation ECO

Option
Order code: S20
Article No.: 6FC5800-0AS20-0YB0



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

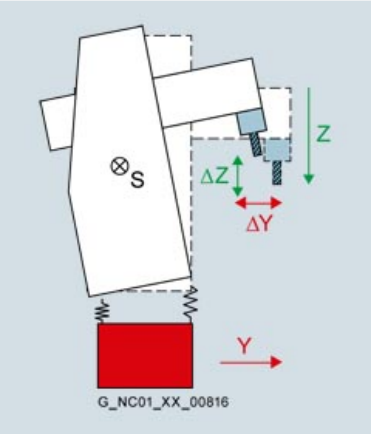
- Improved machining quality with simultaneously increased jerk and acceleration values
- Increased productivity can be achieved without costly mechanical reinforcement of the machine
- Saves costs because often more cost-effective than mechanical measures to increase stiffness
- Can be used, for milling, multitasking, tapping, and laser and water jet machining, for example

→ *Nodding compensation ADVANCED (option S21)*

© Siemens 2020

Nodding compensation ADVANCED

Option
Order code: S21
Article No.: 6FC5800-0AS21-0YB0



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)*

© Siemens 2020

Spatial compensation interface Run MyCCI /VCI	Option Order code: N74 Article No.: 6FC5800-0AN74-0YB0

Note
Do not use for new applications.

Requirement:
- Create MyCC

Restricted functionality for export versions:
Not possible.

→ *Universal spatial compensation interface Run MyCCI /UCI (option N75)*

© Siemens 2020

Universal spatial compensation interface Run MyCCI /UCI	Option Order code: N75 Article No.: 6FC5800-0AN75-0YB0

Function

Run My CCI /UCI provides an interface for the integration of customized compensation algorithms.

Note:

- Replaces Run My CCI /VCI (option N74)

Benefits

- Integration of customer-specific compensation algorithms

Requirement:

- Create MyCC

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Spatial compensation Run MyCC /VCS-ECO	Option Order code: N18 Article No.: 6FC5800-0AN18-0YB0

Function

Run MyCC /VCS-ECO is a version of the Volumentic 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*

© Siemens 2020

Spatial compensation Run MyCC /VCS-A3	Option Order code: N15 Article No.: 6FC5800-0AN15-0YB0

Function

Run MyCC /VCS-A3 implements compensation on Cartesian machines with 3 linear axes in cases where the tool orientation is fixed and the tool lengths are effective in parallel to the coordinates of the linear axes. Gantry axes can be active in parallel to the 3 linear axes.

Benefits

- More precise manufacturing

Requirement:
Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Spatial compensation, Run MyCC /VCS-A5	Option Order code: N16 Article No.: 6FC5800-0AN16-0YB0

Function

Run MyCC /VCS-A5 implements compensation on 5-axis Cartesian machines on which the tool can be oriented relative to the workpiece using 2 rotary axes. Compensation of the measured errors of the linear axes is computed with reference to the tool center point as a function of the current position of the tool.
The machine kinematics can be implemented as pure head kinematics, or as table kinematics with both swivel axes in the table, or as mixed kinematics with one swivel axis in the table – generic transformation types 24, 40 and 56.

Benefits

- More precise manufacturing

Requirement:
Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Spatial compensation for 5 axes plus: Run MyCC /VCS-A5 plus

Option
Order code: N17
Article No.: 6FC5800-0AN17-0YB0

Function

Run MyCC /VCS-A5 plus is designed for use on large machines (3-axis and 5-axis gantry milling machines). The requirements for positioning accuracy of these machines over the entire working area (volumetric accuracy) increases constantly and has become impossible to achieve by mechanical means in many cases.
Run MyCC /VCS-A5 plus implements volumetric compensation of all 21 aspects of freedom from errors that the 3 basic axes of a Cartesian machine tool can have – per axis linearity, 2 degree errors, rolling, nodding, yawing, and deviations from perpendicularity between the axes. These geometrical machine errors result in offset of the tool center point and an orientation error in the tool.
On a 5-axis machine, the tool center point and the orientation error of the tool are compensated with Run MyCC /VCS-A5 plus with TRAORI active. In comparison to LEC leadscrew error compensation LEC and sag error compensation, Run MyCC /VCS-A5 plus supports the total compensation of the geometric errors of a Cartesian machine tool. Run MyCC /VCS-A5 plus can be superimposed on already set LEC and sag error compensations.
The geometric errors of a machine tool are recorded with external laser-based instruments that determine them as effectively as possible. The measurement results are supplied to the SINUMERIK CNC as a file in the specified readable VCS format.
Measuring machine errors is the responsibility of the OEM or machine operator.
Measurement and support as a service are offered by several SINUMERIK system Product Partners who can create the necessary machine-specific files in VCS format.

Benefits

- More precise manufacturing

Requirement:

- Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Spatial compensation for kinematic transformations, Run MyCC /SEC-KT	**Option** Order code: M57 Article No.: 6FC5800-0AM57-0YB0
Note Run MyCC /SEC-KT is a special function and 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*	

© Siemens 2020

Spatial compensation for rotary axes, Run MyCC /VCS ROT	Option Order code: N31 Article No.: 6FC5800-0AN31-0YB0

Function

This static compensation function Run MyCC /VCS ROT enables the user to achieve more accurate tool orientation.
The two swivel axes in the head must be measured once in the working area. The compensation function then uses the table values to perform a real-time calculation of the offsets for the 3 linear axes referred to the current tool center point.

Benefits

- More precise manufacturing

Requirement:
Loadable compile cycle

Restricted functionality for export versions:
Not possible.

© Siemens 2020

Vibration extinction Run MyCC /VIBX	Option Order code: N11 Article No.: 6FC5800-0AN11-0YB0

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.

© Siemens 2020

Cogging torque compensation axis/spindle	Option Order code: D51 to D59 Article No.: 6FC5800-0AD50-0YB0

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)*

© Siemens 2020

Magnetic cogging torque compensation, Run MyCC /COCO

Option
Order code: N46
Article No.: 6FC5800-0AN46-0YB0

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)*

© Siemens 2020

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

Option
Order code: M81
Article No.: 6FC5800-0AM81-0YB0

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)*

© Siemens 2020

Learning error compensation Run MyCC /LECO	Option Order code: N77 Article No.: 6FC5800-0AN77-0YB0

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 leans 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.

© Siemens 2020

Intelligent Load Control	Option Order code: S11 Article No.: 6FC5800-0AS11-0YB0

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.

© Siemens 2020

Intelligent Dynamic Control, intelligent dynamic response adjustment	Option Order code: S23 Article No.: 6FC5800-0AS23-0YB0

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

© Siemens 2020

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.

© Siemens 2020

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.

© Siemens 2020

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)*

© Siemens 2020

PLC user memory, expanded	Option Order codes: D11 ... D18 Article No.: 6FC5800-0AD10-0YB0
	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

© Siemens 2020

User machine data

Function

The NC kernel makes machine data available for configuring the PLC user program. These user machine data are stored in the NCK-PLC interface during CNC power-up, prior to PLC power-up.

The PLC basic program reads these data from the NCK-PLC interface during its initialization phase.

Benefits

This means that specific machine configurations, machine expansions and user options can be activated.

© Siemens 2020

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. This allows not only for simple and economic implementation of the main requirements for functional safety, but also the realization of practical operating and security concepts.
Available functions include, among others:

- 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

A distinction is made between Safety Integrated (system-integrated safety functions with safe Programmable Logic) and Safety Integrated plus (drive-based safety functions and integrated failsafe PLC).

Benefits

- Highly efficient personnel and machine protection

→ *Axis/spindle*
→ *Positioning axis/auxiliary spindle*

© Siemens 2020

NCU-integrated safety functions with Safe Programmable Logic

Benefits

- Highly efficient personnel and machine protection

© Siemens 2020

Safety Integrated – SI axis/spindle, extra for each additional axis/spindle	**Option** Order codes: C71 ... C78 Article No.: 6FC5800-0AC70-0YB0
	Function Extended motion monitoring functions for an axis/spindle.

© Siemens 2020

Safety Integrated – SI axis/spindle package, additional 15 axes/spindles	**Option** Order codes: C61 ... C62 Article No.: 6FC5800-0AC60-0YB0
Function Extended motion monitoring functions for 15 axes/spindles.	

© Siemens 2020

Safety Integrated – SI Basic	Option Order code: M63 Article No.: 6FC5800-0AM63-0YB0
	Function SINUMERIK Safety Integrated for one NCU including 1 axis/spindle, 4 SPL inputs and 4 SPL outputs.

© Siemens 2020

Safety Integrated – SI Comfort	Option Order code: M64 Article No.: 6FC5800-0AM64-0YB0
	Function SINUMERIK Safety Integrated for one NCU including 1 axis/spindle, 64 SPL inputs and 64 SPL outputs.

© Siemens 2020

Safety Integrated – SI Connect	Option Order code: S67 Article No.: 6FC5800-0AS67-0YB0
	Function This option extends secure communication to up to 16 connections.

© Siemens 2020

Safety Integrated – SI High Feature	Option Order code: S68 Article No.: 6FC5800-0AS68-0YB0
	Function SINUMERIK Safety Integrated for one NCU including 1 axis/spindle, 192 SPL inputs and 192 SPL outputs. Requirement: • SINUMERIK NCU 720.xB PN or NCU 730.xB PN

© Siemens 2020

Drive-based safety functions and integrated F-PLC

Benefits

- Highly efficient personnel and machine protection

© Siemens 2020

Safety Integrated plus – SI axis/spindle	Option Order codes: K01 ... K09 Article No.: 6FC5800-0AK00-0YB0
Function Extended motion monitoring functions for an axis/spindle.	

© Siemens 2020

Safety Integrated plus – SI logic	Option Order code: S60 Article No.: 6FC5800-0AS60-0YB0
	Function This option activates the F-PLC integrated in SINUMERIK for failsafe programming.

© Siemens 2020

Safety Integrated plus – SI multi-axis package	Option Order code: S61 Article No.: 6FC5800-0AS61-0YB0
	Function Extended motion monitoring functions for all NCK drives (axes/spindles and positioning axes/auxiliary spindles) activated in the SINUMERIK.

© Siemens 2020

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*

© Siemens 2020

Auto Servo Tuning AST

Function

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)*

© Siemens 2020

Call Auto Servo Tuning from part program AST call	Option Order code: S10 Article No.: 6FC5800-0AS10-0YB0

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*

© Siemens 2020

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

© Siemens 2020

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/IPC 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 or IPC, 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.

© Siemens 2020

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.

© Siemens 2020

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
SinuCom NC	–	SinuCom NC-SI acceptance test	SinuCom NC Trace	SinuCom NC-Trace → Diagnostics

© Siemens 2020

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*

© Siemens 2020

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

© Siemens 2020

Action log

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

Benefits
- Retracing of operator actions

© Siemens 2020

SIMATIC ProDiag S7-1500 for WinCC	Option Order code: P68 Article No.: 6FC5800-0AP68-0YB0

Function

Thanks to the ProDiag functionality which is integrated in the TIA Portal you can configure customized monitoring for your machine or plant and diagnose process disturbances.

The SINUMERIK options 6FC5800-0AP68-0YB0 or 6FC5800-0BP68-0YB0 include the necessary software licenses.

- SIMATIC ProDiag S7-1500 for 256 Supervision Runtime Options in the PLC of the SINUMERIK
- SIMATIC WinCC Runtime Advanced 128 PowerTags from V15 and higher
- SIMATIC ProDiag for WinCC Runtime Advanced from V14 and higher

Benefits

- Automatic code generation in the CPU
- Guaranteed data consistency
- Message texts are derived automatically from information in the project
- Simple visualization on the HMI thanks to prepared controls
- Reaction-free, i.e. the monitoring sessions are defined independently of the user program

→ https://new.siemens.com/global/en/products/automation/industry-software/automation-software/tia-portal/software/tia-portal-options.html

© Siemens 2020

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.

© Siemens 2020

Access MyMachine /P2P	Option Order code: P30 Article No.: 6FC5800-0AP30-0YB0 Product ID: MCS3140

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

© Siemens 2020

Use HMI applications Run MyScreens	Option Order code: P64 Article No.: 6FC5800-0AP64-0YB0

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 6FC5800-0AP64-0YB0 is required to add more than 5 screen forms with Run MyScreens.

© Siemens 2020

Use HMI applications Run MyHMI /3GL	Option Order code: P60 Article No.: 6FC5800-0AP60-0YB0 Product ID: MCS1110

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*

© Siemens 2020

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)*

© Siemens 2020

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

Function

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 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)*

© Siemens 2020

Program screens and user interfaces Run MyHMI /WinCC Frame IPC

Option
Article No.: 6FC5800-0AP62-0YB0
Product ID: MCS1170

Function
Run MyHMI /WinCC Frame IPC includes the options for the operating concept with SINUMERIK Operate and WinCC Runtime Advanced:

- Run MyHMI /WinCC
- SINUMERIK Operate on PCU
- SINUMERIK PCU base software /IPC

Benefits
Configuration of manufacturer-specific operating concepts for optimum usability of the operator panel on a machine tool.

→ *Program screens, operating areas and user interfaces Run MyHMI /WinCC (option P61)*
→ *SINUMERIK Operate /PCU*
→ *SINUMERIK PCU base software /IPC*

© Siemens 2020

Program screens and user interfaces, Run MyHMI /WinCC Frame universal

Option
Order code: P63
Article No.: 6FC5800-0AP63-0YB0

Function
Run MyHMI /WinCC Frame universal includes the options for the operating concept with SINUMERIK Operate and WinCC Runtime Advanced:

- Run MyHMI /WinCC
- SINUMERIK Operate on PC

Benefits
Configuration of manufacturer-specific operating concepts for optimum usability of the operator panel on a machine tool.

→ *Program screens, operating areas and user interfaces Run MyHMI /WinCC (option P61)*
→ *SINUMERIK Operate /PC*

© Siemens 2020

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

Option
Article No.: 6FC5800-0AP61-0YB0
Product ID: MCS1160

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

© Siemens 2020

User interface on SIMATIC Panel Run MyHMI /SIMATIC OP for SIMATIC Comfort and Mobile Panels

Option
Article No.: 6FC5800-0AP03-0YB0
Product ID: MCS1150

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.

© Siemens 2020

Use standardized and freely configurable operating screens, Run MyHMI /PRO

Option
Order code: P47
Article No.: 6FC5800-0AP47-0YB0
Product ID: MCS1130

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.

© Siemens 2020

Execute compile cycles Run MyCC	Option Order code: M04 Article No.: 6FC5800-0AM04-0YB0

Function
Run MyCC enables users to execute their own compile cycles.

Benefits
- Machine manufacturer may supplement his own functions

Restricted functionality for export versions:
Not possible.

→ *Develop compile cycles Create MyCC*

© Siemens 2020

Develop compile cycles Create MyCC	On request
	Function The openness in the NC kernel permits the user to expand the real-time area of the SINUMERIK CNC through the development of customized compile cycles using Create MyCC. Benefits • In-house technological knowledge can be incorporated • Technological knowledge remains with the machine manufacturer Restricted functionality for export versions: Not possible. → *Execute compile cycles Run MyCC (option M04)*

© Siemens 2020

Develop compile cycles on a user-specific interface Create MyCCI

Article No.: 6FC5863-1YP00-0YB8
Product ID: MCS1014

Function
Create MyCCI supports the development of loadable compile cycles based on application-specific interfaces with a predefined development environment. For this special application, the customer uses development tools (GNU compiler/linker) in a Cygwin software shell on a Windows PC.

Benefits
This concept allows the OEM to develop his own applications in C/C++ and to load them as customized compile cycles.

→ *Compile cycles interface Run MyCCI /xy*

© Siemens 2020

Compile cycles interface Run MyCCI /xy

Function

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

For this special application, the customer uses software (GNU compiler/linker) in a Cygwin software shell on a Windows PC.

Benefits

This concept of loadable interfaces therefore allows the OEM to develop real-time applications in C/C++ and to load them as customized compile cycles.

→ *Develop compile cycles Create MyCC*
→ *Integrated tool monitoring and diagnostics IMD BASE Run MyCC /IMD*
→ *Universal spatial compensation interface Run MyCCI /UCI (option N75)*
→ *COA interface for compiled OEM cycles Run MyCCI /COOC (option M67)*

© Siemens 2020

COA interface for compiled OEM cycles Run MyCCI /COOC

Option
Order code: M67
Article No.: 6FC5800-0AM67-0YB0

Function
Create MyCCI /COOC is an interface for the OEM's own developments relating to the CNC Interpreter. Special calculations that are too complex to implement in the CNC language (e.g. intensive matrix algebra) can be programmed in C++ with a mathematics library to be provided by the user.
CNC commands cannot be executed on the C++ level. The development of customized compile cycles in the Interpreter is dependent upon a COA agreement and one-off purchase of the option Create MyCCI /INT.

Benefits
- Implementation of protected, complex mathematical algorithms possible

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

© Siemens 2020

Access MyMachine /Ethernet Article No.: 6FC5864-4AP41-0YB0

Function

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

Benefits

Access MyMachine /Ethernet enables worldwide, secure remote control/monitoring of a machine tool automated with SINUMERIK.

© Siemens 2020

Manage MyPrograms

Article No.: 6FC5864-1AP41-0YB0
Product ID: MCS5010

Function

Manage MyPrograms provides a central system of management and archiving for CNC programs and automatic backup of SINUMERIK CNC data. For example, part programs can simply be transferred from Teamcenter.
It can be linked into the SINUMERIK Operate user interface without requiring the addition of supplementary external equipment on the CNC.

Benefits

- Simple, centralized organization of CNC data with integration of the CNC programmers
- High-speed CNC program provision via networks

© Siemens 2020

Manage MyTools

Article No.: 6FC5864-2AP41-0YB0
Product ID: MCS5020

Function

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

Benefits

- Closed tool data circuit from procurement to use: procurement > Tool setting > Provision > Setup
- Potential for rationalization becomes recognizable, e.g. by cost cutting, optimization and availability

© Siemens 2020

Run MyVNCK

Article No.: 6FC5868-0YF45-0YB0
Product ID: MCS40711

Function
With VNCK we are offering a virtual NC kernel which is implemented at the machine level with the machine OEM directly or in a CAM system.
Run MyVNCK integrates CNC functions into the simulation process. The kernel uses the same source code as the CNC kernel in the CNC. This allows CNC algorithms, language scope, commissioning, data management and communication to be handled in the same way for both Run MyVNCK and CNC kernel.

Benefits
The VNCKs offers machine OEMs the flexibility they need to implement customized machine applications.

© Siemens 2020

Analyze MyCondition

Article No.: 6FC5684-7LP41-0YB
Article No.: 6FC5684-7AP41-0YB
Article No.: 6FC5684-7BP41-0YB
Article No.: 6FC5684-7CP41-0YB

Function

Analyze MyCondition provides test cycles for equability, circularity and universal axis tests, and also offers functionality for continuous data acquisition during the production process. It also supports the reporting of key parameters regarding the wear of mechatronic components.

Benefits

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

© Siemens 2020

Diagnostic functions

Function

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

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

For test purposes, output signals, input signals, and bit memories can be set. All alarms and messages are displayed in plain text on the operator panel along with the corresponding delete criterion. Alarms and messages are displayed separately.
In the Service display menu, it is possible to call up important information about the axis and spindle drives, such as:

- Absolute actual position
- Position setpoint
- Following error
- Speed setpoint
- Actual speed value
- Trace of CNC and drive variables

Benefits

- Diagnostics support, ensuring that the machine is productive again as quickly as possible

© Siemens 2020

Integrate CNC into production Create MyInterface Article No.: 6FC6000-7AC00-0AA8

Function

With Create MyInterface, it is possible to easily integrate SINUMERIK CNCs into the customer's manufacturing network using defined interfaces.

All essential machine data is exchanged between a control system and the CNC, e.g. status information, quantities, alarms, messages, part programs, and tool service life.

Benefits

- Reduced machine downtimes
- Increase machine utilization