© Siemens 2021

Machining channel, additional	**Option** Article No.: 6FC5800-0CC10-0YB0
	Function A machining channel must be regarded as a separate CNC with decoding, block preparation and interpolation. An extension up to 4 machining channels is possible. 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.

© Siemens 2021

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

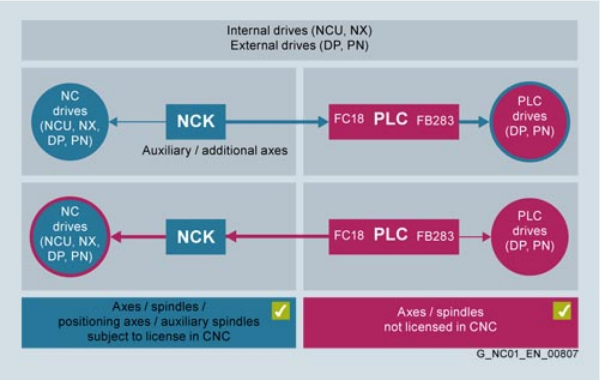
- Battery-buffered CNC user memory

© Siemens 2021

Axis/spindle	Option Article No.: 6FC5800-0CA00-0YB0
	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 → *Virtual axis* → *Internal/external drives*

© Siemens 2021

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 axes/spindles (NCK drives) are controlled by the NCK. A license is required for these axes/spindles and positioning axes/auxiliary spindles in the CNC.

Benefits

- Axes may optionally be controlled by the PLC

→ *Axis/spindle*
→ *Virtual axis*
→ *Internal/external drives*

© Siemens 2021

Internal/external drives

Function

All drives are connected via PROFINET.

- SINAMICS S120 drives are commissioned with SINUMERIK Operate.
- SINAMICS S210 drives are commissioned via Startdrive.

Benefits

- Freedom in the realization of machines

→ *Axis/spindle*

© Siemens 2021

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 2021

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 2021

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 2021

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 2021

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 2021

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 2021

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 2021

Measuring systems 1 and 2, selectable

Function

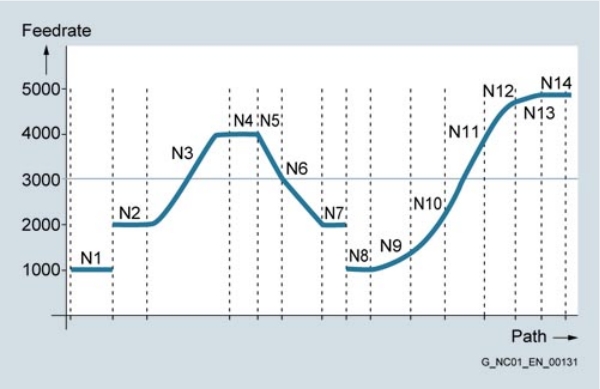
For special applications, 2 encoders can be assigned to one axis, such as for a direct measuring system for the machining process with high demands on accuracy, and an indirect measuring system for high-speed positioning tasks.
The switchover between measuring systems 1 and 2 is performed via the PLC.

Benefits

- Quicker positioning
- More precise production

© Siemens 2021

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 2021

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 2021

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.

Note

Can only be used in combination with SINAMICS S120.

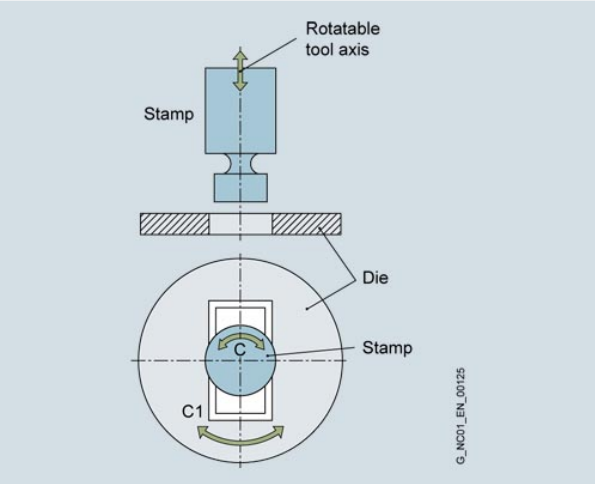
Benefits

- Clamping of workpieces

© Siemens 2021

Tangential control

In the SINUMERIK MC ECO package



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 tools 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

→ *SINUMERIK MC ECO package*

© Siemens 2021

Position switching signals/cam controller	In the SINUMERIK MC ECO package

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.

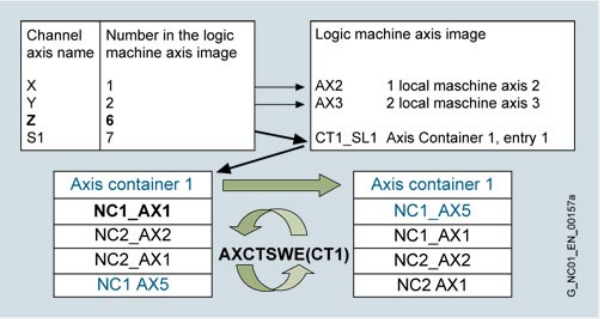
Benefits

- 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*
→ *SINUMERIK MC ECO package*

© Siemens 2021

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

Data structure for flexible assignment of machine axes to geometry or special axes.
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 2021

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 2021

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 2021

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 2021

Fast Interpolation Cycle		In the **SINUMERIK MC Fabrication PERFORMANCE package** Order code: Article No.: 6FC5800-0CM11-0YB0
	Function Reduction of the interpolation cycle to 2 ms Note Restriction: No spindle function The position-control cycle remains unchanged → *SINUMERIK MC Fabrication PERFORMANCE package*	

© Siemens 2021

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 2021

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.

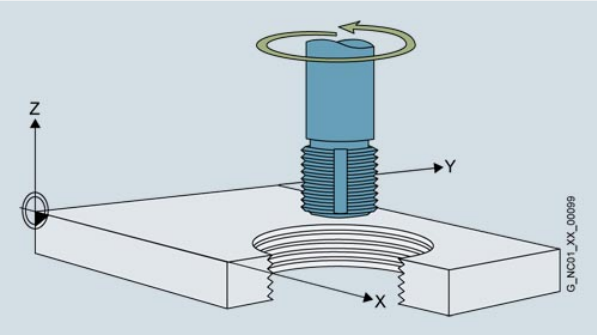
Three points are required to program the circle: the starting point, the intermediate point and the end point.

Benefits

- Circular-path programming obliquely in space

© Siemens 2021

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 2021

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:

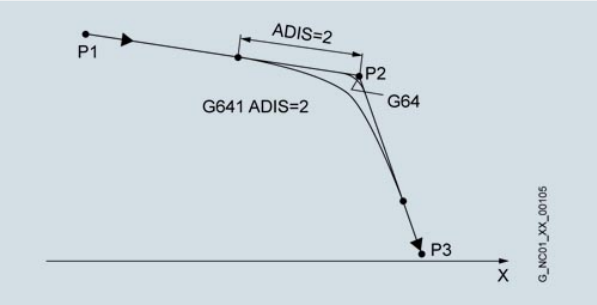
- Linear interpolation
- Circular interpolation
- Helical interpolation
- Involute interpolation
- Splines (A, B, C)
- Polynomial

Benefits

- Structured program layout

© Siemens 2021

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 2021

Continue machining at the contour (retrace support) Run MyCC /RESU	**In the SINUMERIK MC Fabrication ADVANCED package and Fabrication PERFORMANCE package**

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.

Requirement:

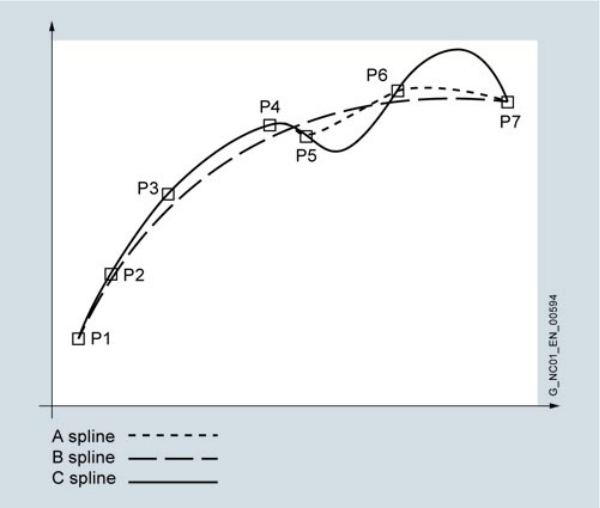
- Cross-mode actions (included in the ECO package)

→ *SINUMERIK MC Fabrication ADVANCED package*
→ *SINUMERIK MC Fabrication PERFORMANCE package*

© Siemens 2021

Spline interpolation (A, B and C splines)

In the SINUMERIK MC TECHNOLOGY package



Spline interpolation

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

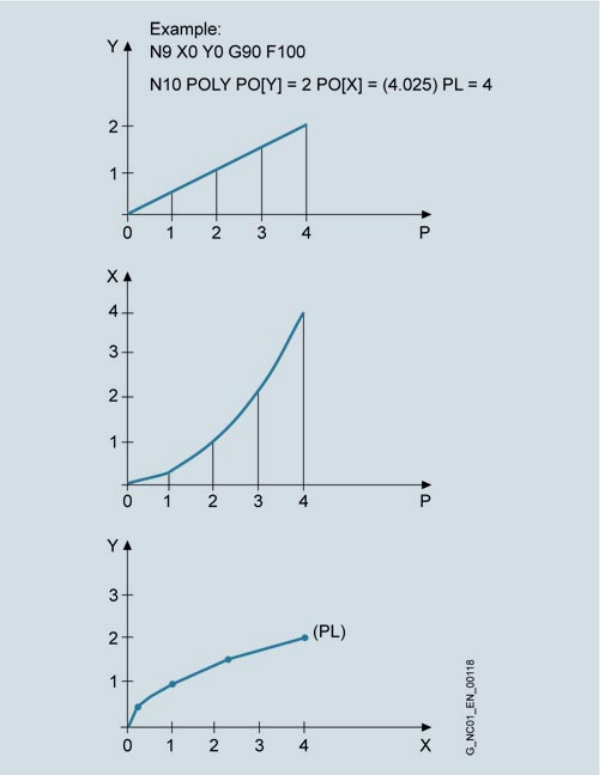
With the COMPCAD compressor, smooth curves of this kind can be approximated within the boundaries of compressor tolerance (parallel tool paths) so that surfaces of a high optical quality can also be obtained even in the case of increased tolerances.

→ *SINUMERIK MC TECHNOLOGY package*

© Siemens 2021

Polynomial interpolation

In the SINUMERIK MC PERFORMANCE package



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.

→ *SINUMERIK MC PERFORMANCE package*

© Siemens 2021

Multi-axis interpolation > 4 interpolating axes	In the SINUMERIK MC TECHNOLOGY package
	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. → *SINUMERIK MC TECHNOLOGY package*

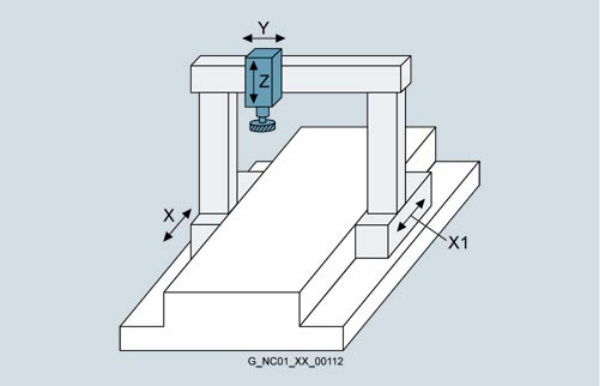
© Siemens 2021

Advanced Surface motion control	**Option** Article No.: 6FC5800-0CS07-0YB0 or in **SINUMERIK MC PERFORMANCE package** and **SINUMERIK MC TECHNOLOGY package**
	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. → *SINUMERIK MC PERFORMANCE package* → *SINUMERIK MC TECHNOLOGY package*

© Siemens 2021

Pair of synchronized axes (gantry axes)

Option
Order code: M02
Article No.: 6FC5800-0CM02-0YB0



Gantry axes (pair of synchronous 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.

Note
When ordering a gantry option, an additional NC axis is provided.

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

→ *Axis/spindle*

© Siemens 2021

Virtual axis

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

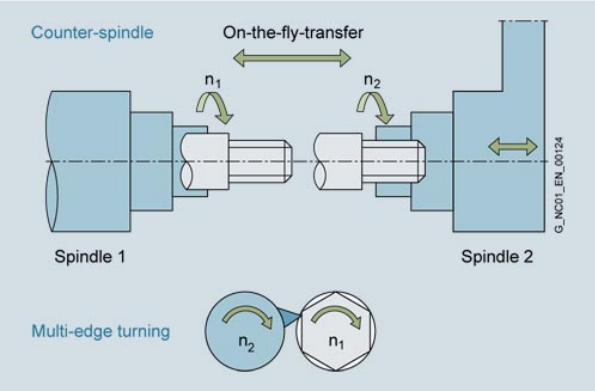
Benefits
Virtual axes are used in leading value couplings. Examples include:

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

→ *Axis/spindle*

© Siemens 2021

Synchronous spindles, multi-edge turning



Function

Precise angular synchronization between one leading and one or more following spindles enables on-the-fly workpiece transfer, particularly for turning machines, from spindle 1 to spindle 2, for example for the purpose of finishing.
In addition to the speed synchronism, the relative angular position of the spindles to one another, e.g., on-the-fly, position-oriented transfer of edged workpieces, is also specifiable.
On-the-fly transfer:

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

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

$n_2 = T \times n_1$

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

Benefits

- Avoidance of idle times due to re-clamping

→ Generic coupling
→ SINUMERIK MC ADVANCED package
→ SINUMERIK MC PERFORMANCE package

© Siemens 2021

Axes, coupled motion

Function
Coupling type for which the traversing movement of a coupled-motion axis is derived from a master value axis via a programmable coupling factor.
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.

→ *Generic coupling*

© Siemens 2021

Generic coupling

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-Basic*
→ *Generic coupling CP-Comfort*

© Siemens 2021

Generic coupling CP-Standard

Function

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

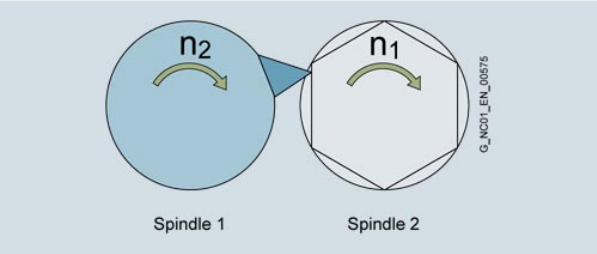
Benefits

- Coupled motion of axes

© Siemens 2021

Generic coupling CP-Basic

In the SINUMERIK MC ADVANCED package



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.

→ *SINUMERIK MC ADVANCED package*

© Siemens 2021

Generic coupling CP-Comfort	In the SINUMERIK MC PERFORMANCE package

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.

→ *SINUMERIK MC PERFORMANCE package*

© Siemens 2021

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.

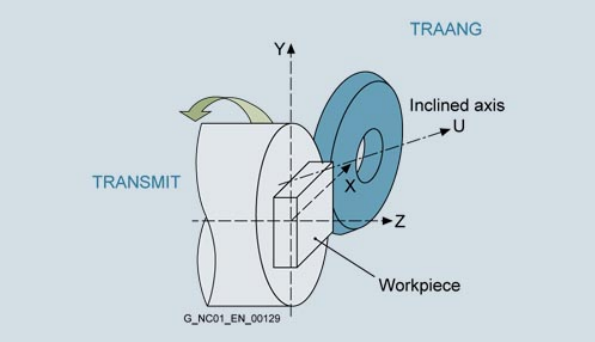
Cartesian PTP travel is also used for cylindrical grinding machines with an inclined axis: With active transformation, the infeed axis can be moved either according to Cartesian coordinates or at the angle of the inclined axis.

Benefits

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

© Siemens 2021

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 2021

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

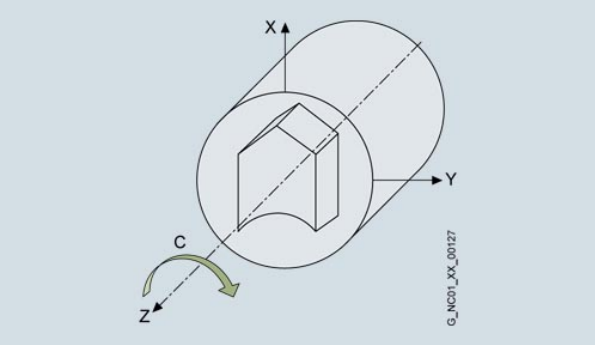
- Orientation of the tool in space can be arbitrarily defined

Restricted functionality for export versions:
Not possible.

© Siemens 2021

TRANSMIT/cylinder surface transformation

In the SINUMERIK MC TECHNOLOGY package



Face machining with TRANSMIT

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.

→ *SINUMERIK MC TECHNOLOGY package*

© Siemens 2021

Measuring functions/measuring cycles

Function

Function to acquire a quantity/variable The measuring functions can be executed 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 2021

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.

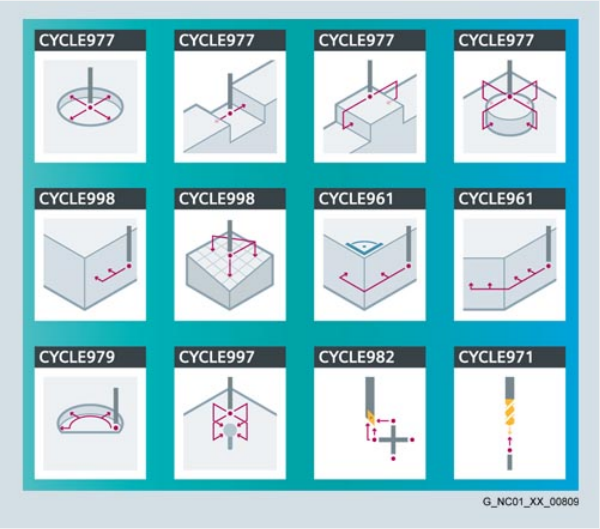
Benefits

- Influencing the process during machining and ensuring machining precision

© Siemens 2021

Measuring cycles for drilling/milling and turning

In the SINUMERIK MC TECHNOLOGY package



Function

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

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

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

The result of the workpiece measurement can either include an automatic work offset or a correction of the tool wear by the difference between the actual value and the setpoint. The measured results can be logged in a file. SINUMERIK measuring cycles offer a standard log, which can be freely configured by the user.

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 2021

Measuring cycles for drilling/milling and turning	In the SINUMERIK MC TECHNOLOGY package

- 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

→ *SINUMERIK MC TECHNOLOGY package*

© Siemens 2021

Measure kinematics	In the SINUMERIK MC TECHNOLOGY package
	Function The Measure kinematics function supports the calibration of kinematic structures of 5-axis machines. The SINUMERIK MC 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 → *SINUMERIK MC TECHNOLOGY package*

© Siemens 2021

SINUMERIK MC ECO package **Included in standard CNC software**

Function

The SINUMERIK MC ECO package contains the options:

- 3 NC axes + 1 channel
- Tangential control
- Position switching signals/cam controller
- Synchronized actions stage 2
- Path velocity-dependent analog output (laser power control)
- Adaptive Control (evaluation of internal drive variables)
- Cross-mode actions ASUB and synchronized actions
- Run MyHMI /3GL
- Program execution from external storage EES
- CNC user memory
- Oscillation functions

Benefits

- Simple machines for special technologies with
  - a customized user interface based on Windows or WinCC
  - path control based on G code with up to three axes
- E.g. for glass cutting, adhesive application, or flat-bed grinding

Restricted functionality for export versions:
Not possible.

→ *Tangential control*
→ *Position switching signals/cam controller*
→ *Synchronized actions stage 2*
→ *Path velocity-dependent analog output (laser power control)*
→ *Adaptive Control (evaluation of internal drive variables)*
→ *Cross-mode actions ASUB and synchronized actions*
→ *Use HMI applications Run MyHMI /3GL*
→ *Execution from external storage EES*
→ *Oscillation functions*

© Siemens 2021

SINUMERIK MC ADVANCED package	Option Article No.: 6FC5800-0CW61-0YB0

Function

The SINUMERIK MC ADVANCED package contains the options:

- Contour handwheel
- Sag compensation, multi-dimensional
- Generic coupling CP-Basic

Benefits

Implementation of machines for special technologies, such as woodworking

- with a user interface based on Windows or WinCC tailored to the corresponding machining operation
- with path control based on G code with up to three axes
- with coupled axes, higher precision and handwheel

Restricted functionality for export versions:
Not possible.

→ *Contour handwheel*
→ *Sag compensation, multi-dimensional*
→ *Generic coupling CP-Basic*

© Siemens 2021

SINUMERIK MC PERFORMANCE package	Option Article No.: 6FC5800-0CW62-0YB0

Function

The SINUMERIK MC PERFORMANCE package contains the options:

- Polynomial interpolation
- Leadscrew error compensation, bidirectional
- Generic coupling CP-Comfort
- Advanced Surface motion control

Benefits

Implementation of machines for special technologies, such as woodworking

- with a user interface based on Windows or WinCC tailored to the corresponding machining operation
- with path control based on G code with up to three axes for 2.5D applications
- with coupled axes, higher precision with synchronous spindles

Restricted functionality for export versions:
Not possible.

→ *Polynomial interpolation*
→ *Leadscrew error compensation, bidirectional*
→ *Generic coupling CP-Comfort*
→ *Advanced Surface motion control*

© Siemens 2021

SINUMERIK MC TECHNOLOGY package	Option Article No.: 6FC5800-0CW63-0YB0

Function

The SINUMERIK MC TECHNOLOGY package contains the options:

- TRANSMIT/cylinder surface transformation
- Multi-axis interpolation > 4 interpolating axes
- Machining package 5 axes
- 3D tool radius compensation
- Measuring cycles for drilling/milling and turning (calibrate workpiece probe, workpiece measurement, tool measurement)
- Measure kinematics
- Spline interpolation (A, B and C splines)
- Advanced Surface motion control

Benefits

Implementation of sophisticated machines for special technologies, such as woodworking or laser machining

- with a user interface based on Windows or WinCC tailored to the corresponding machining operation
- with path control based on G code and 5-axis machining in mold making
- machining centers for end face and cylinder surface machining

Restricted functionality for export versions:
Not possible.

→ *TRANSMIT/cylinder surface transformation*
→ *Multi-axis interpolation > 4 interpolating axes*
→ *Machining package 5 axes*
→ *3D tool radius compensation*
→ *Measuring cycles for drilling/milling and turning*
→ *Measure kinematics*
→ *Spline interpolation (A, B and C splines)*
→ *Advanced Surface motion control*

© Siemens 2021

SINUMERIK MC Fabrication ADVANCED package	Option Article No.: 6FC5800-0CW64-0YB0
	Function The SINUMERIK MC Fabrication ADVANCED packages comprises the following options: • Run MyCC /CLCX • Run MyCC /RESU Benefits Implementation of sophisticated machines for, e.g., water jet machining → *Clearance control Run MyCC /CLCX* → *Continue machining at the contour Run MyCC /RESU*

© Siemens 2021

SINUMERIK MC Fabrication PERFORMANCE package	Option Article No.: 6FC5800-0CW65-0YB0

Function

The SINUMERIK MC Fabrication PERFORMANCE package comprises the following options:

- Run MyCC /CLCX
- Run MyCC /RESU
- Run MyCC /HSLC
- Fast Interpolation Cycle

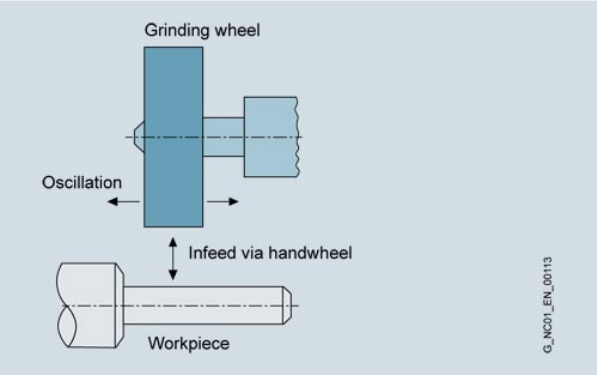
Benefits

Implementation of sophisticated machines for, e.g., lasers

→ *Clearance control Run MyCC /CLCX*
→ *Continue machining at the contour Run MyCC /RESU*
→ *High-speed laser switching signal Run MyCC /HSLC*
→ *Fast Interpolation Cycle*

© Siemens 2021

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 2021

Contour handwheel	In the SINUMERIK MC ADVANCED package

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

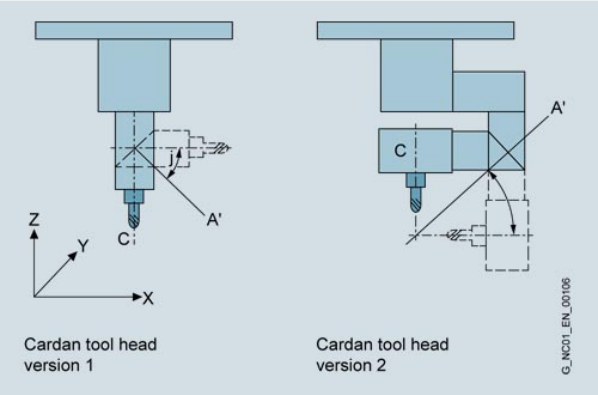
- Used with conventional turning and grinding machines for setup/scratching
- More user-friendly operation of the machine in setup mode

→ *Feedrate interpolation*
→ *SINUMERIK MC ADVANCED package*

© Siemens 2021

Machining package 5 axes

In the SINUMERIK MC TECHNOLOGY package



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 length compensation for 5-axis machining
  When machining with the 4th/5th axis, the lengths of the selected tool are automatically included and compensated in the axis movement.
- Oriented tool retraction
  If machining is interrupted e.g. because of tool breakage, a program command can be used to carry out defined, oriented tool retraction.
- Tool-oriented RTCP
  With the Remote Tool Center Point RTCP function, the tool swivel axes can be positioned in the manual mode, as long as there is compliance with the tool center point marked by the tool tip. The RTCP function simplifies the inclusion of program interpolation points in manual mode with orientation of the tool.
- Cardan milling head/nutating head
  Requirements: Machining package 5 axes with 5-axis transformation.
  Using a cardan milling head in conjunction with the Nutating head function, it is possible to machine outside contours of spatially shaped parts at high feedrates. To do this, the CNC performs a 5-axis transformation. The 3 translatory main axes X/Y/Z determine the tool operating point; 2 rotary axes, one of which is an inclined axis (angle can be set in the machine data), permit virtually any orientation in the working area. Cardan milling heads, versions 1 and 2 are supported. In the case of version 2, the position of the operating point does not change when the tool is swiveled; the compensating movements required for orientation changes are minimal.

Note

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

Benefits

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

© Siemens 2021

Machining package 5 axes	In the SINUMERIK MC TECHNOLOGY package
	Restricted functionality for export versions: Not possible. → *Multi-axis interpolation > 4 interpolating axes* → *Axis/spindle* → *SINUMERIK MC TECHNOLOGY package*

© Siemens 2021

Multiple feedrates in one block

Function

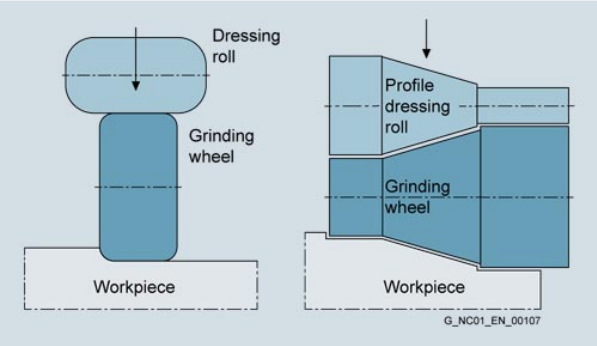
Depending on external digital and/or analog CNC inputs, this function can be used for motion-synchronous activation of up to 6 different feedrates, a dwell time, and a retraction in a single CNC block.
The input signals are combined in an input byte with a permanently assigned function. The retraction 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 2021

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).

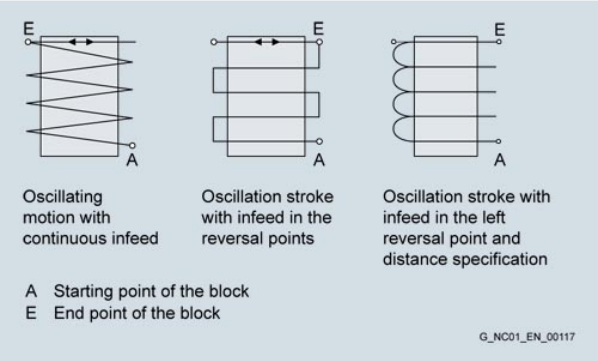
Benefits

- Simultaneous grinding and dressing possible

© Siemens 2021

Oscillation functions

In the SINUMERIK MC ECO package



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

→ *SINUMERIK MC ECO package*

© Siemens 2021

High-speed CNC inputs/outputs

Function

The high-speed CNC inputs/outputs function supports the input or output of signals in the position-control/interpolation cycle. The high-speed CNC inputs/outputs can be used for machines, such as those used for grinding and laser machining, as well as in SINUMERIK Safety Integrated.

Input signals are possible for the following:

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

Output signals are possible for the following:

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

© Siemens 2021

High-speed CNC inputs/outputs	
	Benefits • Rapid response to special events → *Position switching signals/cam controller*

© Siemens 2021

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 2021

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 2021

Synchronized actions stage 2	In the SINUMERIK MC ECO package

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).

→ *SINUMERIK MC ECO package*

© Siemens 2021

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 2021

Analog value control in the interpolation cycle

Function

This function makes it possible to directly specify values from the part program via analog outputs.
The value specified by the NCK can be changed by the PLC before it is output to the hardware of a SIMATIC DP ET 200 analog module.
The hardware outputs are written in the interpolation cycle.

Benefits

- Implementation of machine-specific functions by the machine manufacturer

Requirement:

- Analog input

© Siemens 2021

Path velocity-dependent analog output (laser power control)	In the SINUMERIK MC ECO package
	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 → *SINUMERIK MC ECO package*

© Siemens 2021

High-speed laser switching signal, Run MyCC /HSLC	In the SINUMERIK MC Fabrication PERFORMANCE package

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 defferential in relation to the position setpoint, are entered.

Benefits

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

→ *SINUMERIK MC Fabrication PERFORMANCE package*

© Siemens 2021

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 2021

Clearance control Run MyCC /CLCX	**In the SINUMERIK MC Fabrication ADVANCED package and Fabrication PERFORMANCE package**
	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. Note Without Run MyCC /CLCX, the number of interpolating axes is limited to 4, and with Run MyCC /CLCX to 3. Benefits • More precise manufacturing → *SINUMERIK MC Fabrication ADVANCED package* → *SINUMERIK MC Fabrication PERFORMANCE package*

© Siemens 2021

Adaptive Control (evaluation of internal drive variables)	In the SINUMERIK MC ECO package

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

→ *SINUMERIK MC ECO package*

© Siemens 2021

Asynchronous subprograms ASUB

Function

An asynchronous subprogram (ASUB) 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

© Siemens 2021

Cross-mode actions ASUB and synchronized actions	In the SINUMERIK MC ECO package
	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 → *SINUMERIK MC ECO package*

© Siemens 2021

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 2021

Working plane

Function

Plane in the coordinate system defined by 2 coordinate axes.

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 2021

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 2021

Job list

Function

This can be used to create a job list (loading list) for every workpiece to be machined, for extended workpiece selection. This job list contains instructions on making the following preparations for executing part programs, even when multiple channels are involved:

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

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

Benefits

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

© Siemens 2021

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 2021

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 2021

Skip blocks

Function

Program block that is executed by the PLC user program depending on default settings.
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

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

© Siemens 2021

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 2021

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 2021

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 2021

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 2021

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 2021

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 2021

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.

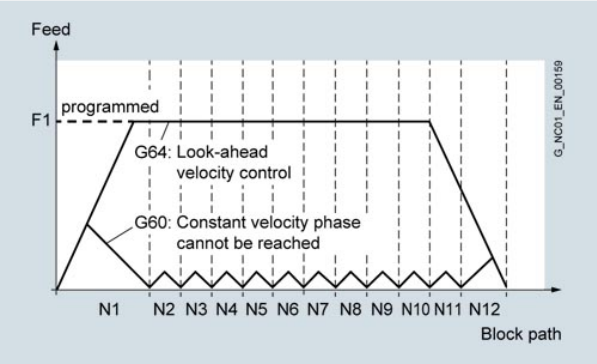
The preprocessing memory is constantly reloaded during execution. Block execution can be interrupted with the STARTFIFO command until the preprocessing memory has been filled, or STOPFIFO (start high-speed machining section) or STOPRE (stop preprocessor) can be programmed.

Benefits

- Increased productivity due to fast execution

© Siemens 2021

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 2021

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:

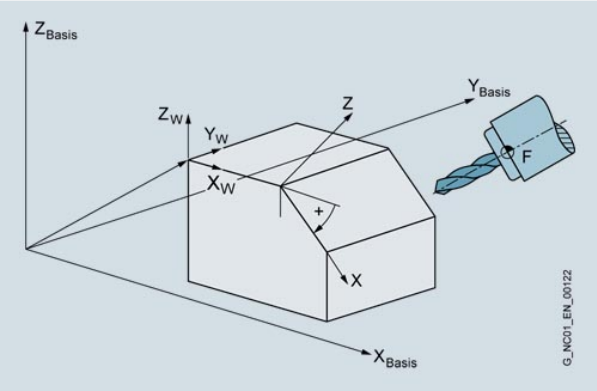
- ATRANS additive programmable work offset
- AROT additive rotation in space or in a plane
- ASCALE scale factor (multiplication)
- AMIRROR repeated mirroring
- AROTS additive rotation referred to the solid angles projected into the planes

© Siemens 2021

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 2021

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 2021

Axis/spindle interchange

Function

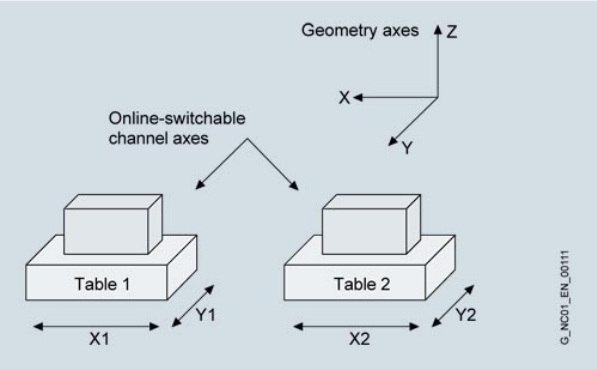
An axis/a spindle is permanently assigned to a specific channel via machine data. The axis/spindle replacement function can be used to release an axis/a spindle (RELEASE) and to assign it to another channel (GET), i.e. to replace the axis/spindle. The relevant axes/spindles are determined via machine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

Benefits

- Implementation of special machine tools

© Siemens 2021

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 2021

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 2021

Online ISO dialect interpreter

Function

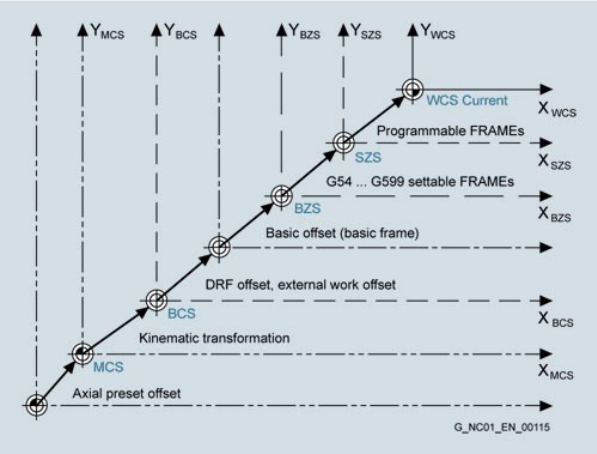
In general, part programs for SINUMERIK CNCs are programmed according to DIN 66025 and relevant expansions. Part programs created according to the ISO standard (e.g., G codes from other manufacturers) can be read in, edited and executed on SINUMERIK CNCs using the online ISO dialect interpreter.

Benefits

- Part programs from other manufacturers can be used

© Siemens 2021

Work offsets



Function

Coordinate transformation during which the coordinate origin is moved.

According to DIN 66217, clockwise, rectangular (Cartesian) coordinate systems are used in machine tools. The following coordinate systems are defined:

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

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

The following work offsets are possible:

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

© Siemens 2021

Work offsets	
	Benefits • Easier programming of workpieces → *Frame concept*

© Siemens 2021

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 2021

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 2021

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 2021

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 2021

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 2021

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 2021

Execution from external storage EES	In the SINUMERIK MC ECO package

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.

→ *SINUMERIK MC ECO package*

© Siemens 2021

DRF offset (differential resolver function)

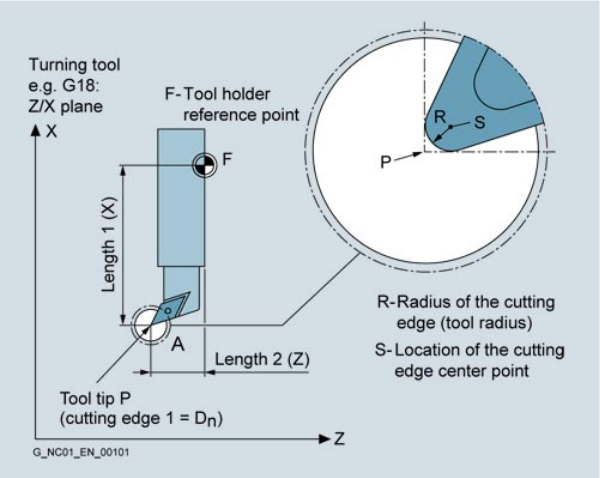
Function
The DRF offset generates an additional incremental work offset in the AUTO control mode via the electronic handwheel.

Benefits
- Compensate for tool wear within a programmed block

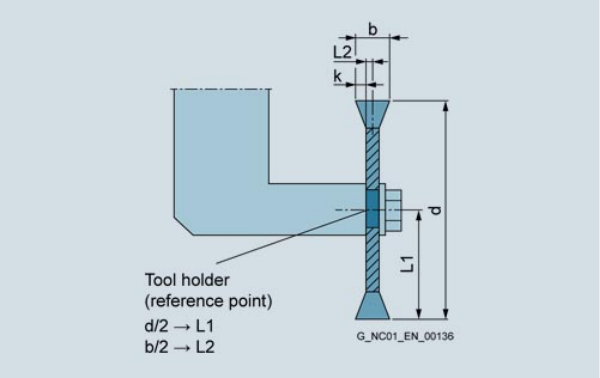
→ *Handwheel override*

© Siemens 2021

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.

Tool types, parameterizable

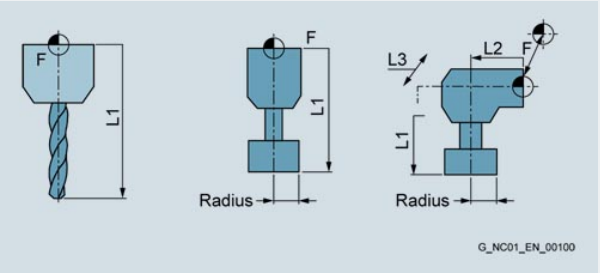
Group 1xy	Milling tools	From spherical head cutters to bevel cutters
Group 2xy	Drills	From twist drills to reamers
Group 4xy	Grinding tools	From surface grinding wheels to dressers
Group 5xy	Turning tools	From roughing tools to threading tools
Group 700	Slotting saws	Wood technology
Group 900	Special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

Benefits

The tool types define how the geometry specifications are computed.

© Siemens 2021

Tool offsets



Function

Taking into account the dimensions of the tool when machining the workpiece.

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.

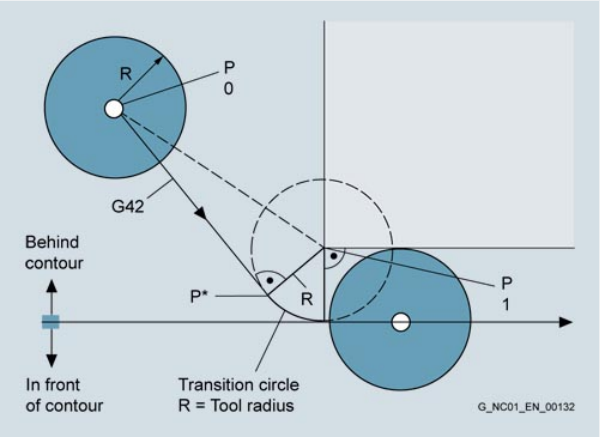
Tool offset D always has a reference to tool number T when Siemens tool management is active, e.g. with monitoring functions and management of sister tools.

Benefits

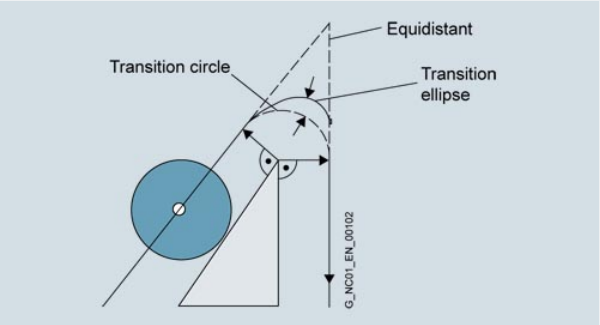
- Easier programming, as the tools are only taken into consideration upon execution

© Siemens 2021

Tool radius compensation



CONT for selection behind the contour



Traveling around the outside corners with transition circle/transition ellipse

Function

Tool offset that takes into account the radius of the tool.

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 2021

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 2021

3D tool radius compensation	In the SINUMERIK MC TECHNOLOGY package

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

→ *SINUMERIK MC TECHNOLOGY package*

© Siemens 2021

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 2021

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.

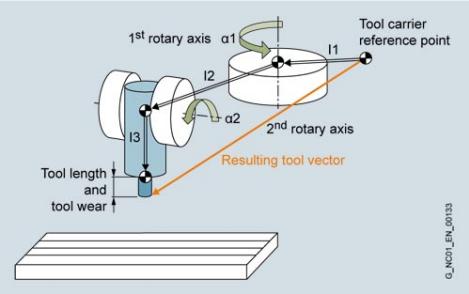
In turret magazines on turning machines, the T command carries out the entire tool change, that is, locates and inserts the tool. The tool change mode can be set using machine data.

Benefits

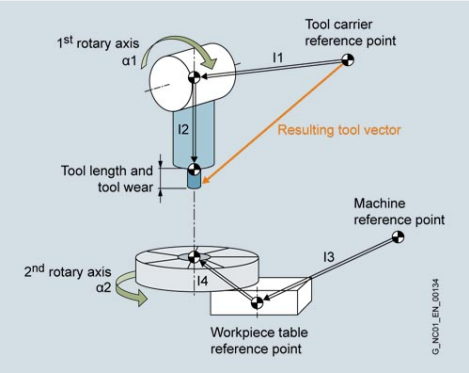
- Adapting the tool change to the machine

© Siemens 2021

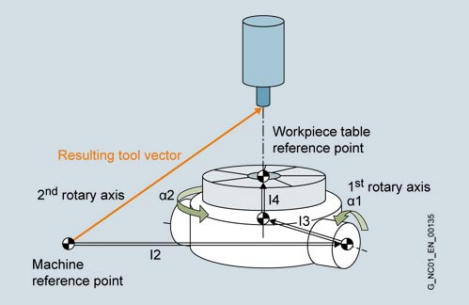
Tool carrier with orientation capability



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:

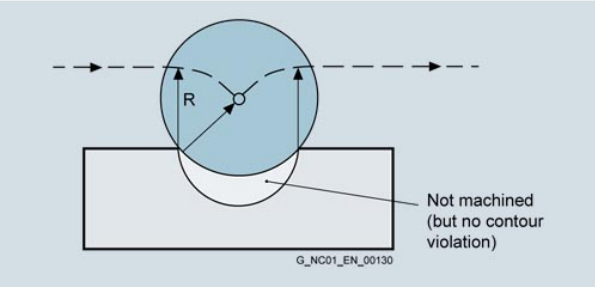
- Rotatable tool type T (tool)
- Rotatable tool/rotatable workpiece table type M (mixed)
- Rotatable workpiece table type P (part)

Benefits

- Any position of the tool/TCP can be reached in the machining space.

© Siemens 2021

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 2021

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 2021

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)*

© Siemens 2021

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 2021

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 IO*
→ *www.siemens.com/profinet*

© Siemens 2021

PROFINET IO

Function

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

A PROFINET IO system is assembled from the following devices:

- An IO controller is a controller, typically a PLC, CNC, robot control or motion controller, that controls the automation task. An IO controller is a master as compared to PROFIBUS.
- An IO device is a distributed field device which is linked via PROFINET IO. It is controlled by an IO controller. An IO device can consist of several modules and submodules. All data to be exchanged are assigned slots and subslots for the purpose of addressing. These are defined in the General Station Description (GSD) file. 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.

© Siemens 2021

PROFINET IO	
	Benefits • Openness offers flexibility for future innovations • Greater freedom to implement tailored machine and plant concepts

© Siemens 2021

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 2021

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 2021

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 2021

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.

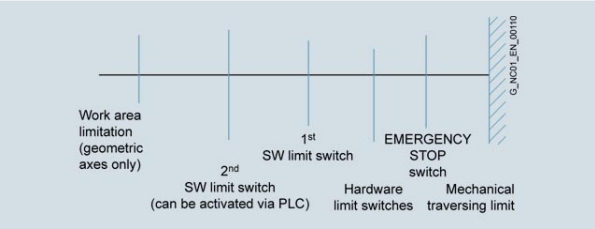
Benefits

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 2021

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 2021

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 2021

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 2021

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.

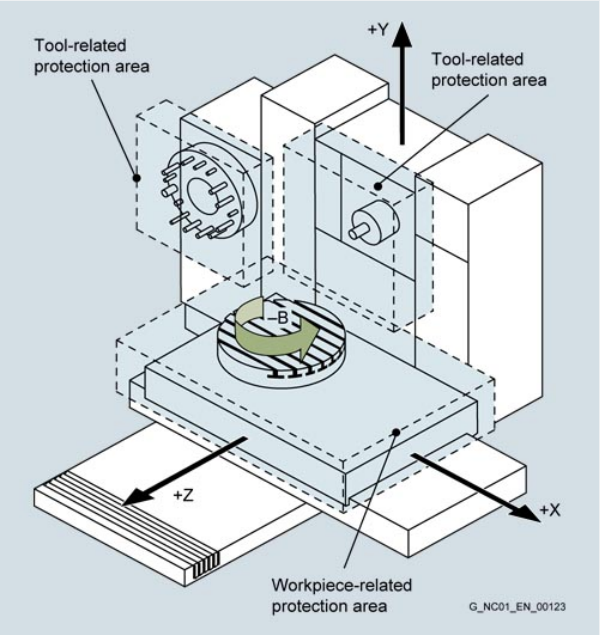
Benefits

- Monitors adherence to the parameterized clamping tolerance

→ *Position monitoring*
→ *Standstill monitoring*

© Siemens 2021

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 2021

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 2021

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 2021

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 2021

Collision avoidance Protect MyMachine /3D Primitives

Option
Article No.: 6FC5800-0CS03-0YB0

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:

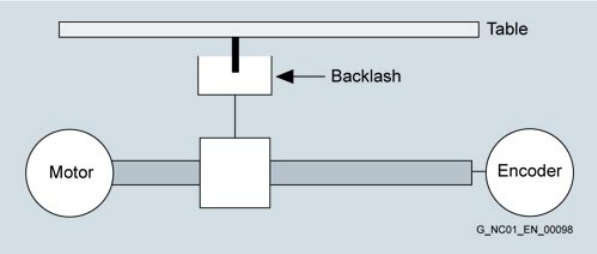
- Protection area elements include simple bodies (cubes, cylinders, spheres, ...)
- Protection elements ≤ 34
- Protection areas ≤ 17
- Collision pairs ≤ 10
- 1-channel
- Graphical visualization (requirement: simultaneous recording)
- Transparent model data
- Simple configuration with copy & paste
- Tool 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

© Siemens 2021

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 2021

Leadscrew/measuring system error compensation

Function

Compensation of thread pitch errors of the ball screw or errors of the measuring system, e.g. due to inaccuracies of the code disk/code ruler.

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

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

The principle of "indirect measuring" on CNC-controlled machines is based on the assumption that the leadscrew pitch is constant at every point within the traversing range. This means that the actual position of the axis can be derived from the position of the drive spindle (ideal situation).

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

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

Benefits

- More precise workpiece machining

© Siemens 2021

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.

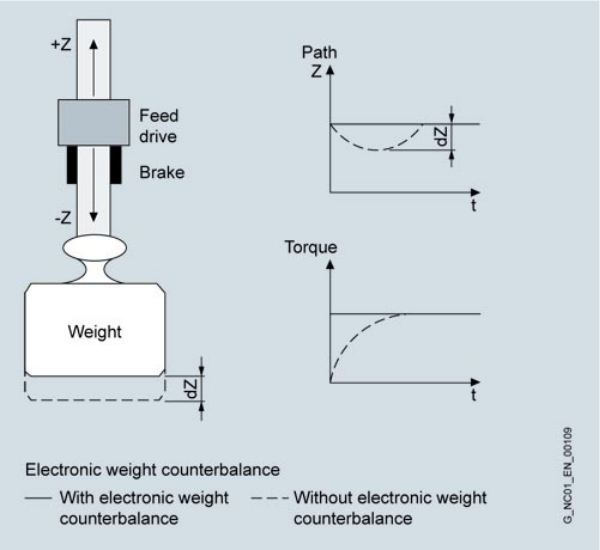
Benefits

- 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 2021

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 2021

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.

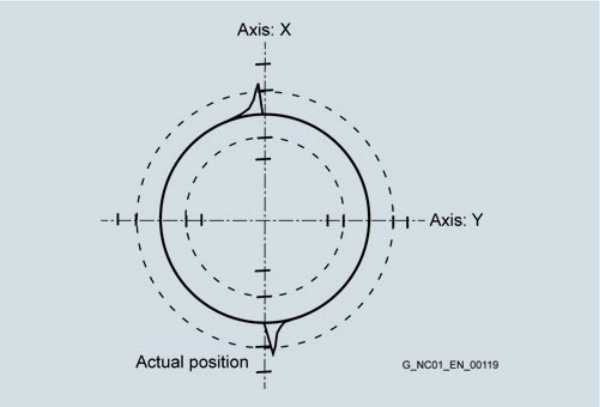
Abrupt changes in these parameters are automatically smoothed by the SINUMERIK CNC system in order to prevent machine overload and avoid inadvertent triggering of monitoring functions.

Benefits

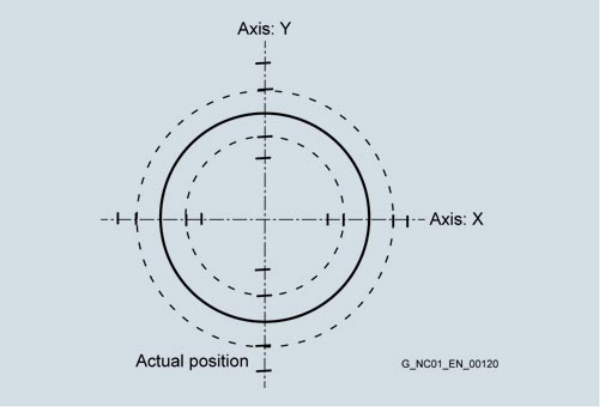
- Compensation of thermal expansion of machine parts
- Consistent manufacturing accuracy of machined workpieces

© Siemens 2021

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*

© Siemens 2021

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.

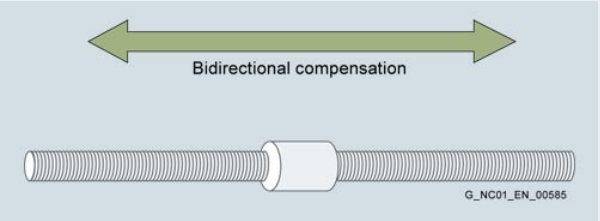
Benefits

- Creation and parameterization of a characteristic for quadrant error compensation.

© Siemens 2021

Leadscrew error compensation, bidirectional

In the SINUMERIK MC PERFORMANCE package



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

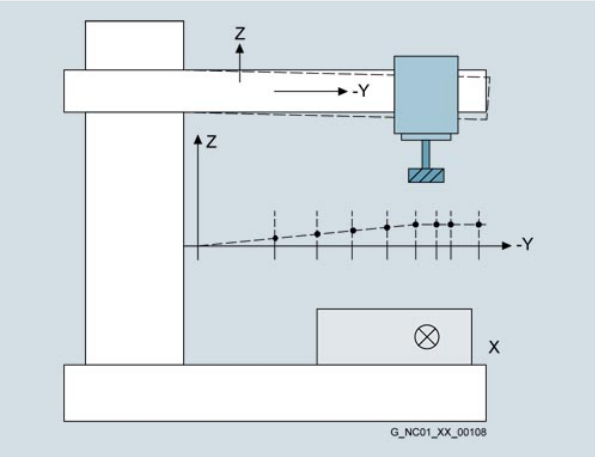
Restricted functionality for export versions:
The correctable tolerance band is restricted to 1 mm (0.039 in).

→ *SINUMERIK MC PERFORMANCE package*

© Siemens 2021

Sag compensation, multi-dimensional

In the SINUMERIK MC ADVANCED package



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).

→ *SINUMERIK MC ADVANCED package*

© Siemens 2021

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 2021

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 2021

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

© Siemens 2021

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

→ *Safety Integrated – F-PLC*
→ *Axis/spindle*

© Siemens 2021

Drive-based safety functions and integrated F-PLC

Benefits

- Highly efficient personnel and machine protection

→ *Safety Integrated*

© Siemens 2021

Safety Integrated F-PLC	Option Order code: S60 Article No.: 6FC5800-0CS60-0YB0
	Function This option activates the F-PLC integrated in SINUMERIK for failsafe programming.

© Siemens 2021

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 2021

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

© Siemens 2021

Commissioning series machines

Function

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

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

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

© Siemens 2021

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 2021

Use HMI applications Run MyHMI /3GL	In the SINUMERIK MC ECO package

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*
→ *SINUMERIK MC ECO package*

© Siemens 2021

Program screens, operating areas and user interfaces Create MyHMI

Function

Create MyHMI offers scalable HMI openness through high-level language programming. Create MyHMI is available for the different programming languages currently in use (C++, .net).

Benefits

Creation of 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)*
→ *SINUMERIK MC ECO package*

© Siemens 2021

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 their own screen form layouts.

This function is realized in a high-level language development environment based on .net or C++ . The screen forms are created in the development environment Visual Studio.

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)
- Enable data exchange between screen forms

Notes

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

Benefits

Configuration of machine manufacturer or end user-specific user interfaces to stand out from the competition.

→ *Use HMI applications Run MyHMI /3GL (option P60)*
→ *SINUMERIK MC ECO package*

© Siemens 2021

Program screens, operating areas and user interfaces Run MyHMI /WinCC	Option Article No.: 6FC5800-0CP61-0YB0
	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