© Siemens AG 2018

3D simulation, finished part

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

Required for enabling 3D view in part program simulation. 3D simulation can also be displayed in real-time simulation. Without this option, only 2D simulation can be displayed with various views.

If the simultaneous recording option is already being used in the system, it is also possible to display the 3D view during real-time simulation.

© Siemens AG 2018

Axis/spindle, each additional

Option
Article No.: 6FC5800-0AC20-0YB0
(For V 2.x: 6FC5800-0AA00-0YB0)

This option must be selected if the required number of interpolating axes/spindles is greater than the basic quantity of axes/spindles. The basic quantity of axes/spindles that are available with the SINUMERIK 828D family can be seen in the table below:

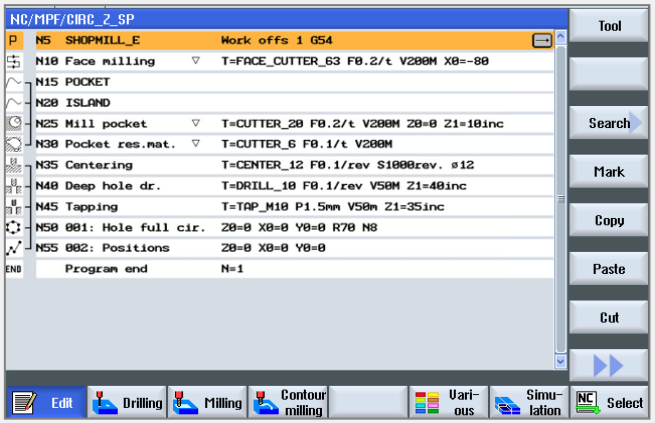
Axes/spindles	SINUMERIK 828D BASIC	SINUMERIK 828D		
	SW 24x	SW 26x	SW 28x	SW 28xA
Basic quantity				
• Turning	3	3	3	3
• Milling	4	4	4	4
• G-Tech Cylindrical/G-Tech Surface	3	3	–	3
Maximum configuration				
• Turning	5	6	8	10
• Milling	5	6	6	8
• G-Tech Cylindrical/G-Tech Surface	5	6	–	10

→ Positioning axis/auxiliary spindle, each additional

© Siemens AG 2018

ShopTurn/ShopMill machining step programming

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



Processes such as drilling, centering, plunging or pocket milling are represented as machining steps in a simple and clear manner. In this way, part programs – even for complex machining operations – are very compact and easy to read. Associated machining steps are automatically interlinked and can be assigned any position patterns.

This unique programming convenience allows you to achieve the shortest programming times, even for highly demanding machining tasks.

© Siemens AG 2018

Evaluation of internal drive variables

Option
Order code: S53
Article No.: 6FC5800-0AS53-0YB0
(For V 2.x: 6FC5800-0AM41-0YB0)

The following real-time drive variables can be called/evaluated in the part program:
$AA_LOAD drive capacity utilization in percent (%)
$AA_POWER drive active power in watts (W)
$AA_TORQUE drive torque setpoint in Newton meters (Nm)
$AA_CURR actual axis/spindle current in amperes (A)
These variables can be used in conjunction with synchronized options.
They can also be read out via the PLC interface, NCvariables DB1200.DBxxxxx. Regarding the PLC, the evaluation of internal drive variables is standard.

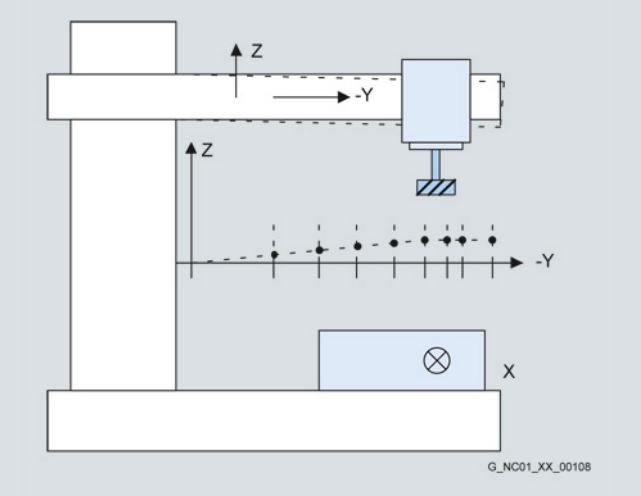
Application examples:
Evaluation of these drive variables also permits machines and tools to be protected from overload as well as shorter machining times and improved surface quality of workpieces. Evaluation of internal drive variables is a prerequisite for implementing adaptive control (AC).

Adaptive control can be parameterized within the part program as follows:
- Additive influence: The programmed feed value is corrected additively.
- Multiplicative influence: The feed value is multiplied by a factor (override).

© Siemens AG 2018

Sag compensation, multi-dimensional

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



Sag compensation

Multidimensional compensation is also possible for the effects of physical influences and manufacturing tolerances such as sag or leadscrew pitch errors. The compensation tables can be switched over from the PLC.

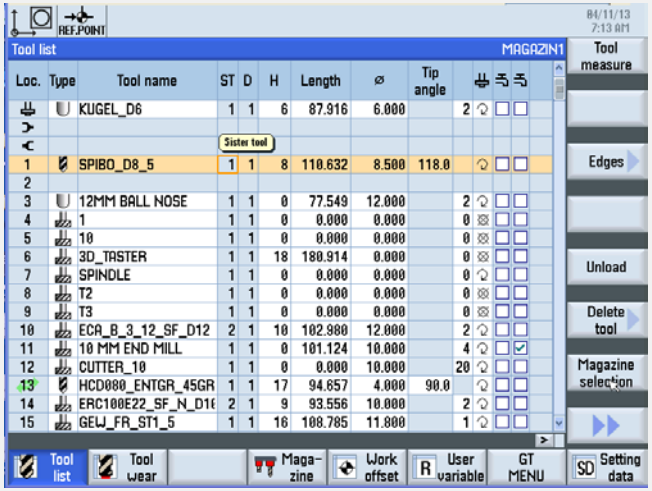
When the reference axis and the compensating axis are identical, leadscrew pitch errors can be compensated. By transferring weighting factors (PLC interface), stored compensating characteristics can be adapted to different conditions (e.g. tools).

The most important features of interpolation and compensation using tables are as follows:
- Independent error characteristics can be defined; a total of 2x the maximum number of axes
- Freely selectable compensating positions; number configurable, depending on the configuration of the CNC user memory
- Interpolating inclusion of the compensation values
- Weighting factor for compensation of tool weights
- Reference axis and compensating axis are selectable

© Siemens AG 2018

Replacement tools for tool management

Option
Order code: M78
Article No.: 6FC5800-0AM78-0YB0



Option for defining replacement tools. Tool life is monitored on the basis of machining time or number of components.

Once the tool lifetime has been reached, or the tool has machined the specified number of components, then the defined replacement tool is automatically loaded.

© Siemens AG 2018

Extended operator functions	Option Order code: P16 Article No.: 6FC5800-0AP16-0YB0
	• Number of levels for skip blocks 10 (default value 2) • Teach-in function • Backup of workpiece setup data • Display of active synchronized actions • DRF offset • Overstoring • Extended block search (program/search pointer, level up/down, interrupt point) • Manual workpiece measurement: advanced strategies for part probing • Additional measuring versions beyond standard scope (standard function scope, workpiece zero: set edge, align edge, right-angled corner, 1 drilled hole, and 1 circular spigot. Expansion of the measurement window via combo box) • Load/save MDI program

© Siemens AG 2018

Advanced technology functions	Option Order code: P58 Article No.: 6FC5800-0AP58-0YB0

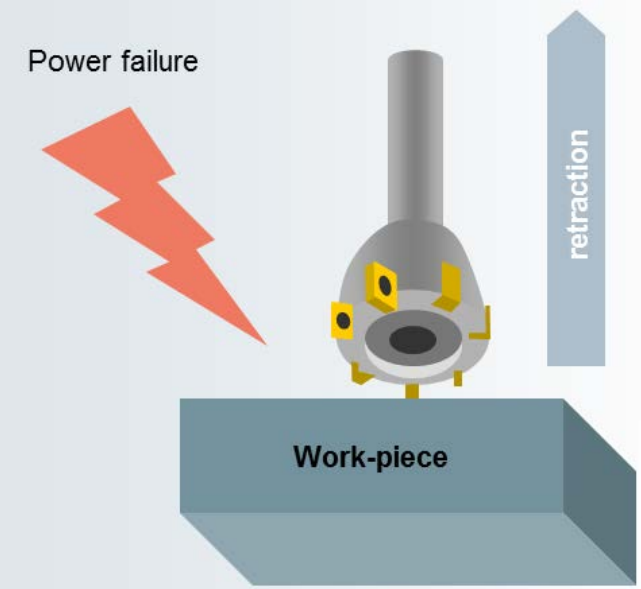
The following extended technologies are only available in SINUMERIK 828D BASIC:

- Asymmetric grooves (turning only)
- Drill and thread milling
- Thread milling
- Multi-edge milling
- Engraving
- Extended stock removal along contour with segmentation of blank (turning only)
- Contour grooving and plunge turning (turning only)
- Milling of contour pockets and spigots (with up to 12 islands)
- Position pattern – hide position
- Asymmetrically turn a shoulder
- DIN thread undercut

© Siemens AG 2018

Stop and retract ESR, drive-autonomous

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



A safe position is assumed from the machining level without any collision between tool and workpiece.

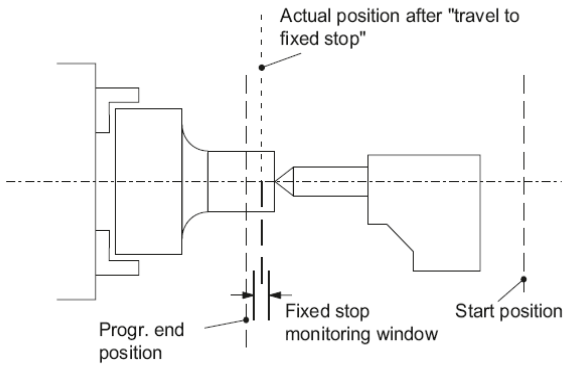
In addition to the drive-autonomous stop and retract function, the CNC-controlled stop and retract functionality is also provided. To permit gentle interpolated retraction on the path or contour, the path interpolation can be processed further for a definable period following the triggering event.

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

© Siemens AG 2018

Travel to fixed stop with Force Control

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



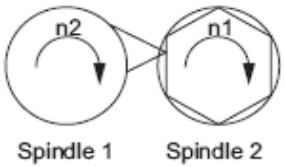
The extended travel to fixed stop function can be used to adapt torque or force on a modal or non-modal basis, to initiate travel with limited torque/force (force control, FOC) or to program traversing functions at any time using synchronized actions.

Application example:
Servo tailstock function with force control requirement. This function can also be combined with synchronized actions.

© Siemens AG 2018

Generic coupling Basic: CP-Basic

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



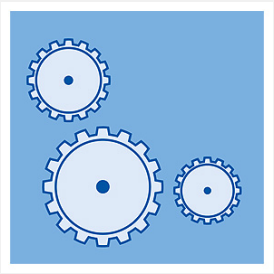
- Up to 4 x simple coupled motion
- Up to 1x synchronous spindles/multi-edge turning or master value coupling/curve table interpolation or axial coupling in the machine coordinate system

Application example multi-edge machining (polygon turning):
The synchronous spindle provides the basis for multi-edge machining by specifying an integer ratio between the leading spindle and the following spindle. The diagram shows spindle 2 with the workpiece and spindle 1 with the cutting tool. Both spindles are synchronized and operate with the same ratio (e.g. 1:6) in order to achieve the polygon shaping on the workpiece.

© Siemens AG 2018

Generic coupling Comfort: CP-Comfort

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



- Up to 4 x simple coupled motion
- Up to 4 x synchronous spindle/multi-edge turning and/or master value coupling/curve table interpolation and/or axial coupling in the machine coordinate system

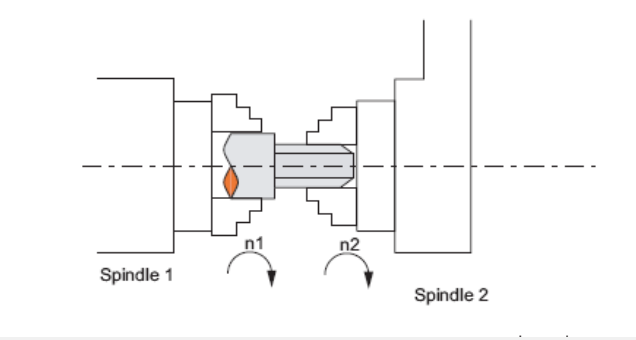
Or:
- 1 x electronic gear for up to 3 leading axes is possible (without curve table interpolation and without cascading)

Application example:
This option is suitable for hobbing. Gear hobbing machines have a rotating, multi-tooth cutting tool for machining teeth on helical gears, worm gears, cyclic gears etc.
Also suitable when two axes (e.g. tool axes) with different ratios are to be coupled to third axes (e.g. blank axes).

© Siemens AG 2018

Generic coupling Static: CP-Static

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



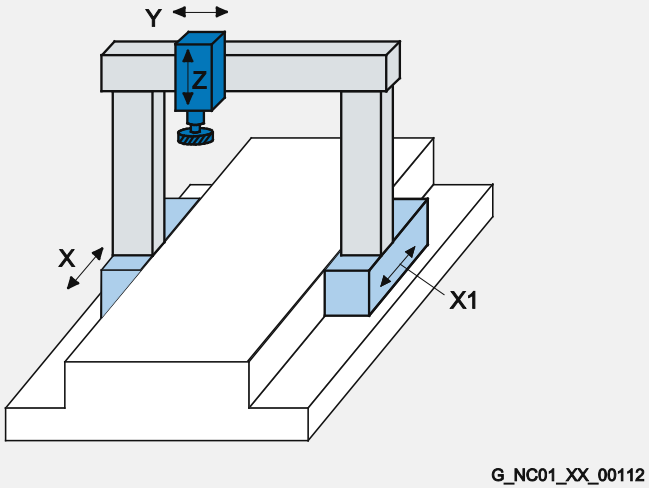
Simple synchronous spindle with coupling ratio 1:1 – no multi-edge machining

Application example:
Reverse-side machining on a double-spindle lathe on-the-fly workpiece transfer from the position-synchronous leading spindle to the following spindle (or vice versa), without having to brake to standstill.

© Siemens AG 2018

Pair of synchronized axes (gantry axes), Basic

Option
Order code: S51
Article No.: 6FC5800-0AS51-0YB0
(For V 2.x: Option
Order code: M02
Article No.: 6FC5800-0AM02-0YB0)



Gantry axes (pair of synchronized axes X/X1)

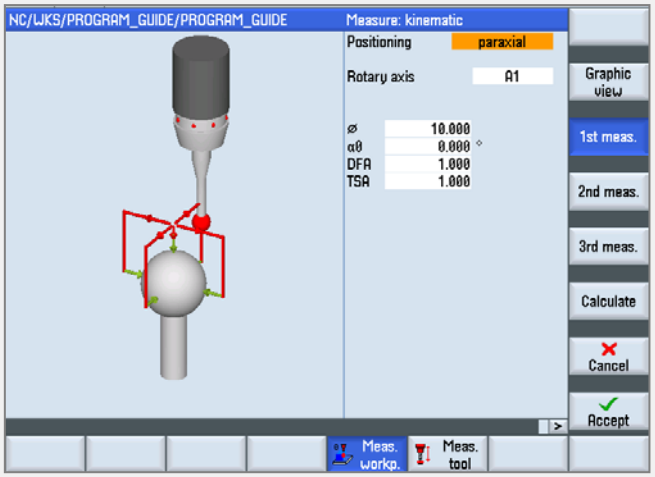
The gantry axes function can be used to traverse mechanically-coupled axes simultaneously without mechanical offset. The actual values are continuously compared and even the smallest deviations corrected.
During both operation and programming, the axes defined in a gantry grouping are treated like a single machine axis.

A maximum of one gantry pair is possible with this option.

© Siemens AG 2018

Measure kinematics

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



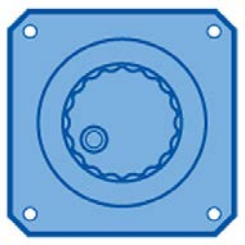
This option is for determining the parameters of kinematic transformations of the digitally or manually alignable rotary axes quickly and automatically. The function is ideal for initial startup, because a dimension drawing of the machine is not required.

It can also be used for regular checking of the production process when high precision is required.

© Siemens AG 2018

Contour handwheel

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



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

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

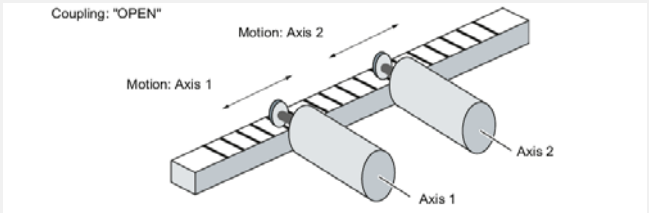
The handwheel's direction of rotation determines the direction of travel:

- 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.

© Siemens AG 2018

Master/slave for drives, Basic

Option
Order code: S52
Article No.: 6FC5800-0AS52-0YB0
(For V 2.x: 6FC5800-0AM03-0YB0)



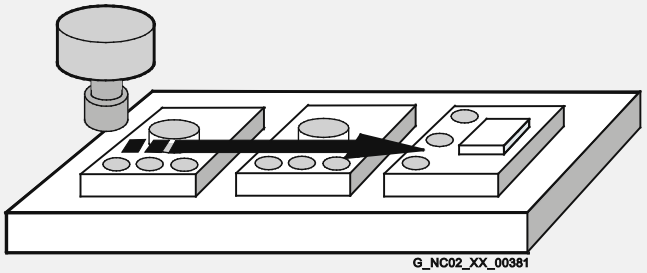
A master-slave coupling is a speed setpoint coupling between a master and slave axis – performed at the position controller level – with and without torque equalization control. The coupling can be permanently closed.

Applications for a master-slave coupling include:
- Increasing the power of mechanically-coupled drives
- Compensating the edge backlash of gearing and gears by entering a pretensioning torque

© Siemens AG 2018

Multiple clamping of various workpieces
(included in ShopMill/ShopTurn)

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



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

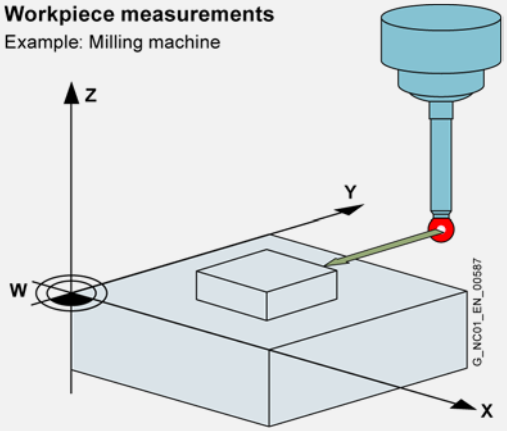
This function also allows different workpieces to be finished on multiple vises or gripping yokes in a time-saving process.

Note:
From software release 4.5, multiple clamping (6FC5800-0AP14-0YB0) is included in ShopMill/ShopTurn (6FC5800-0AP17-0YB0).

Measuring cycles for drilling, milling, and turning

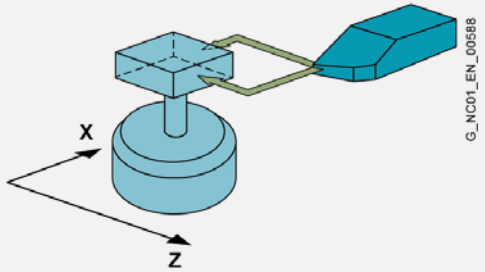
Option
Article No.: 6FC5800-0AP28-0YB0

Workpiece measurements
Example: Milling machine



Workpiece measurement

Tool measurements
Measure turning tool length 1, length 2



Tool measurement

Measuring cycles are subprograms for implementing specific measuring tasks. They can be adapted to specific requirements through the appropriate parameterization.

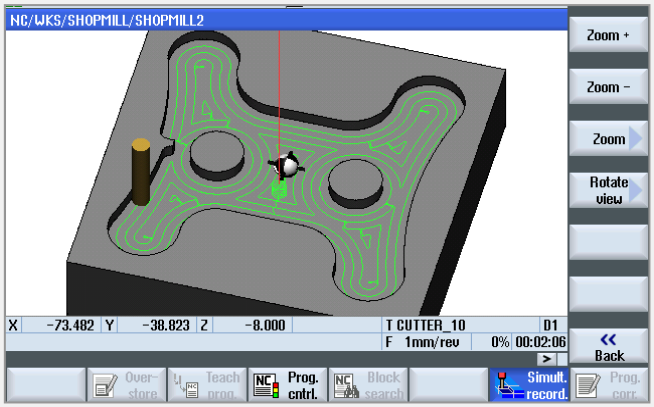
A probe is moved up to the clamped workpiece in the same way as a tool and the measured values are acquired. The flexible selection of measuring cycles means that you can perform practically every measuring task required on lathes and milling machines. An automatic tool offset or work offset can be applied based on the results of the workpiece measurement.

The selected tool is moved to the probe and the measured values are acquired. The probe is either in a fixed position or is swung into the working area using an appropriate mechanism. The tool geometry measured is entered in the appropriate tool offset data set.

© Siemens AG 2018

Simultaneous recording

Option
Order code: P22
Article No.: 6FC5800-0AP22-0YB0



If the program is running in the automatic mode, the simulation can be displayed on the machining screen of the CNC using real-time simulation. Tool motion on the simulation screen depends on the actual tool motion (axis motion) and precisely displays the machining operation and its progress.

Note:
To avoid an incomplete display, you must activate the simultaneous recording prior to starting the CNC.

© Siemens AG 2018

Positioning axis/auxiliary spindle, each additional

Option
Article No.: 6FC5800-0AC30-0YB0
(For V 2.x: 6FC5800-0AB00-0YB0)

This is an option for a positioning axis that is required in addition to the basic quantity of axes/spindles. The total number of axes/spindles is limited by the maximum number of axes available in the SINUMERIK 828D family.

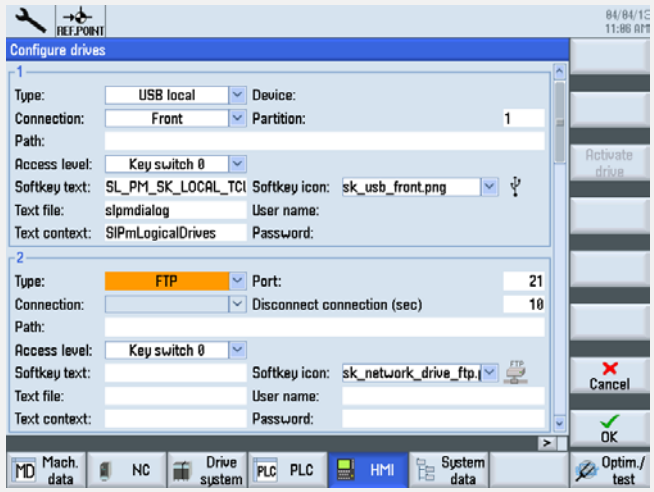
Application examples:
- ATC tool magazine
- Palette, auxiliary spindle (without tapping/thread cutting)
- Any positioning axis.

→ Axis/spindle, each additional

© Siemens AG 2018

Program management via a network drive

Option
Article No.: 6FC5800-0AP01-0YB0

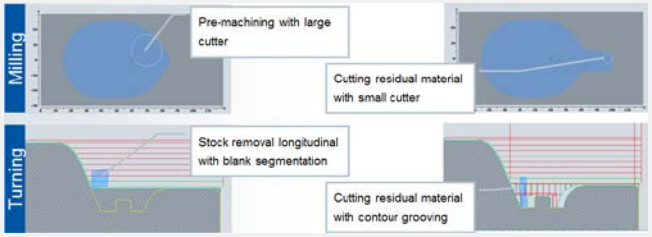


- Allows part programs to be executed from an external server with Windows Share/per FTP client.
- A maximum of 4 additional drives can be defined via the Ethernet.
- Part programs can be executed from each of these network drives.
- Program/workpiece data can also be managed.

© Siemens AG 2018

Residual material detection

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



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

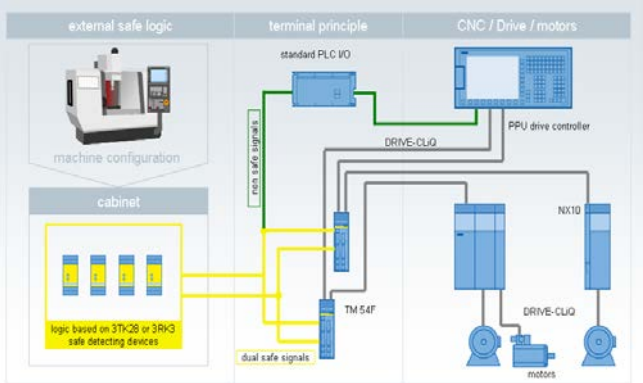
Contour turning offers:
- Contour/axis-parallel cutting with residual material detection
- Contour cutting with residual material detection
- Plunge-turning with residual material detection

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

© Siemens AG 2018

Safety Integrated, extended functions

Option
Order code: C50
Article No.: 6FC5800-0AC50-0YB0
(For V 2.x: 6SL3074-0AA10-0AA0)



- Safe Torque Off (STO)
  The drive pulses are cancelled and therefore the energy feed is safely and electronically disconnected
- Safe Stop 1 (SS1) time-controlled or speed-controlled
  Safe stopping of a drive, with subsequent transition into the STO state
- Safe Stop 2 (SS2)
  Safe stopping of the drive with subsequent monitoring for standstill (SOS)
- Safe Operating Stop (SOS)
  Monitors drives for standstill – the drives remain fully functional in the position control
- Safely-Limited Speed (SLS)
  Monitoring of 4 configurable velocity limit values, e.g. during setup
- Safe Acceleration Monitor (SAM) / Safe Brake Ramp (SBR)
  Safe monitoring of the braking operation
- Safe Speed Monitor (SSM)
  Safe checkback signal when a value falls below a settable speed limit, e.g. for enabling a protective door
- Safely-Limited Position (SLP)
  Switchable traversing range limitation (2 areas)
- Safe Direction (SDI)
  Safe monitoring of the direction of motion
- Safe Brake Management (SBM)
  - Safe Brake Control (SBC)
    2-channel braking signal – integrated in the SINAMICS S120 Motor Module
  - Safe Brake Test (SBT) → Diagnostics function
    Cyclic brake test

The extended Safety Integrated functions require a software license in the form of a CNC option per axis/spindle with Safety functions. A SINAMICS TM54F Terminal Module is required for controlling the extended Safety Integrated functions.

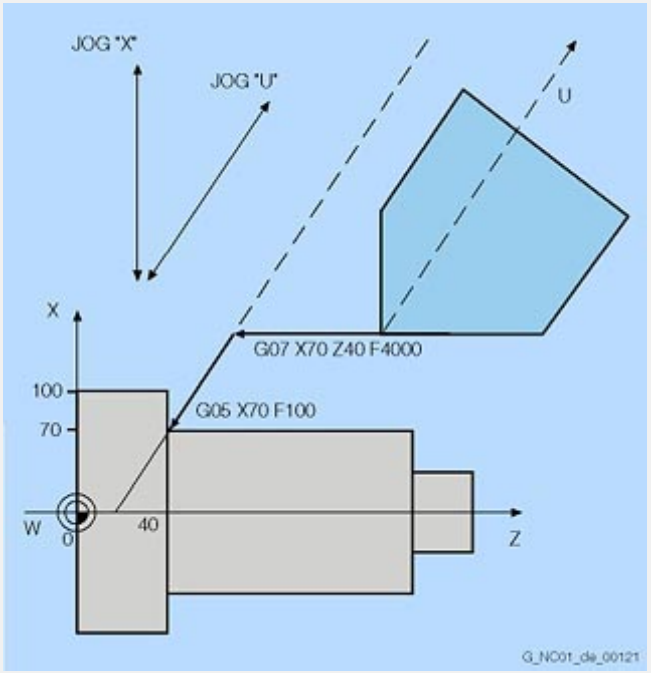
Commissioning support:

- Graphical commissioning with SINUMERIK Operate
  For each Safety function there is a clear, graphical display available for fast commissioning
- Integrated acceptance test with SINUMERIK Operate
  Partially automated acceptance test for all safety-related functions. Simple operation of the test process, automatic configuration of trace functions and automatic generation of an acceptance record.

© Siemens AG 2018

Inclined axis

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



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

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

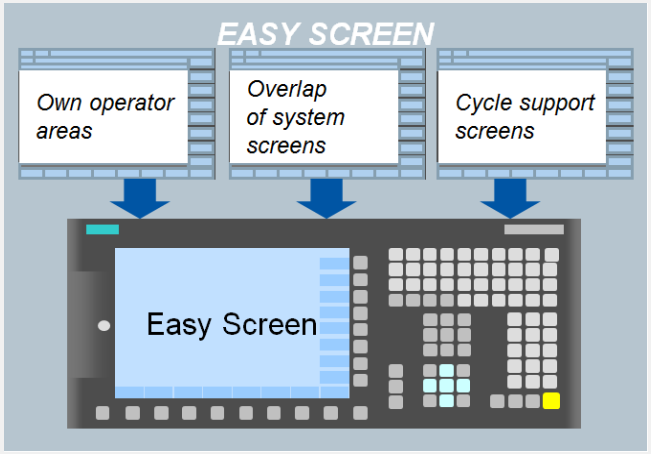
Tool offsets and work offsets are also entered in the Cartesian system and converted over to the real machine axes.

For oblique plunge-cutting with G05, it is necessary to program the start position with G07.

© Siemens AG 2018

SINUMERIK Run MyScreens

Option
Order code: P64
Article No.: 6FC5800-0AP64-0YB0



The Easy Screen functionality allows SINUMERIK users to design their own user interfaces in order to display machine OEM or end user-specific function expansions, or to configure their own screen form layout.

User interfaces configured by Siemens or other machine manufacturers can be modified or replaced. This function is implemented using an integrated interpreter and configuring files containing the description of the user interface.

The screen forms can be designed directly on the control itself. A graphic tool is required to create graphics and pictures. Part programs can be processed with newly created user interfaces.
Configuring examples for new screen forms, which can also be used as the basis for the user's own screen forms, are contained in the toolbox provided.

You can implement the following functions using Easy Screen:
- Display screen forms and provide softkeys, variables, tables, texts, help texts, graphics, and help screens
- Start actions when screen forms are called and exited, when softkeys are pressed, and values (variables) are entered
- Dynamic restructuring of screen forms, including changing softkeys, designing display fields and displaying, replacing as well as deleting display texts and graphics
- Read and write variables, combine with mathematical, comparative or logical operators
- Execute subprograms, file functions, program instance services (PI services) or external functions (HMI Advanced)
- Enable data exchange between screen forms
- Easy Screen is configured using ASCII files that can be stored on the PCU. Files that contain ASCII descriptions for the layout of interactive screen forms, softkey functions and display texts and graphics are interpreted. These configuring files are created with the ASCII editor, taking into account certain special syntax rules.

The user interface can be expanded even in the basic version by up to 5 screen forms using predefined softkeys with the integrated editor.
More than 5 screen forms are possible with the Operate Runtime license OA Easy Screen (option P64).

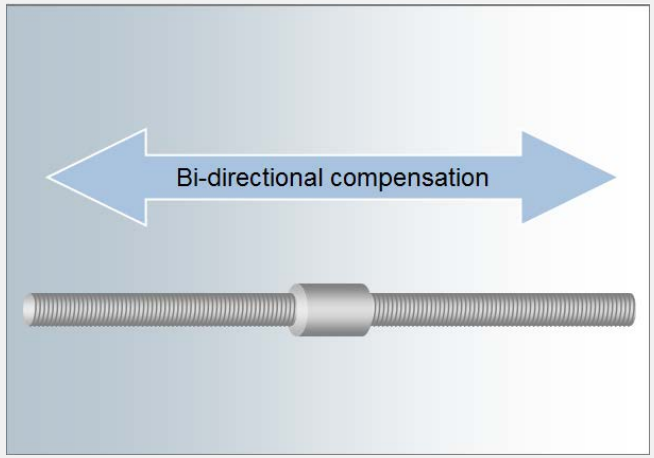
© Siemens AG 2018

SINUMERIK Operate Runtime license OA Easy Screen	
	→ SINUMERIK Run MyScreens

© Siemens AG 2018

Leadscrew error compensation, bidirectional

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



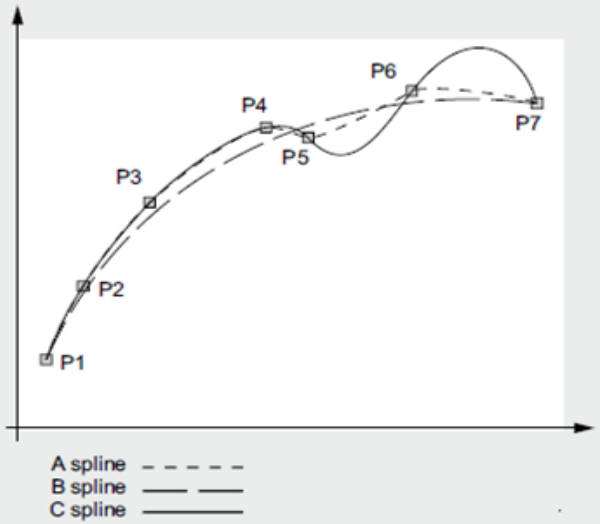
Bidirectional compensation is an expansion of the leadscrew error compensation function (LEC) or the measuring system error compensation function (MSEC). In contrast to LEC and MSEC, bidirectional compensation works in both directions.

This option supports price-sensitive machining applications for face and peripheral surfaces on lathes without a mechanical Y axis.

© Siemens AG 2018

Spline interpolation (A, B and C splines)

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



Using spline interpolation, it is possible to obtain a very smooth curve from just a few defined interpolation points along a set contour. The interpolation points are connected by polynomials. The compressor converts linear blocks (e.g. from CAD) at block transitions to splines with constant velocity (COMPON) or to splines with constant acceleration (COMPCURV). Soft transitions are created, thereby reducing the wear on the mechanical parts of machine tools. However, if the interpolation points are placed close together, sharp edges can also be programmed. Spline interpolation also considerably reduces the number of program blocks required.

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

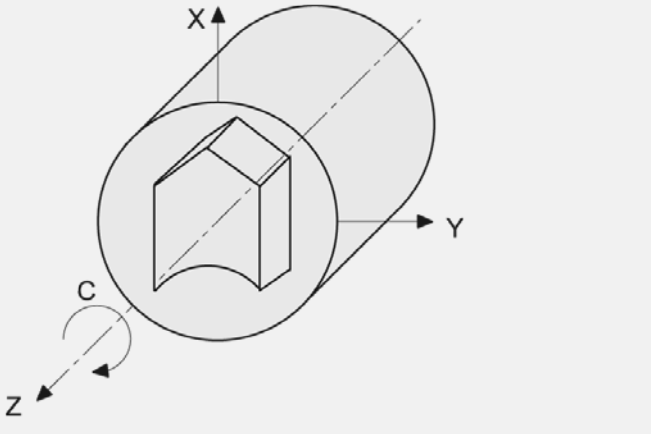
Every polynomial can represent a spline. Only the algorithm determines the type of spline.
- A-spline is only true to the tangents.
- B-spline is true to the tangents and the curvature, but does not run through the nodes (interpolation points).
- C-spline is continuous along tangents and curves and runs through the nodes.

With the COMPCAD compressor, smooth curves (i.e. curves without any steps) of this kind can be generated within the boundaries of compressor tolerance (parallel tool paths) so that surfaces of a high optical quality are obtained, even in the case of increased tolerances.
Spline interpolation for 3-axis machining is suitable for more simple applications and in the JobShop domain.

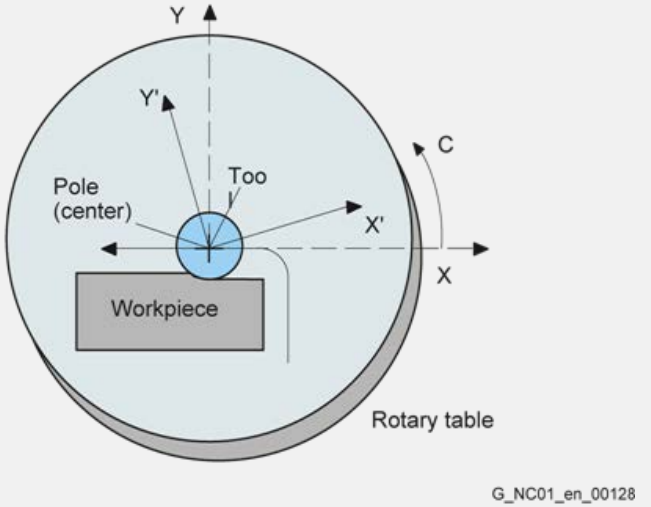
© Siemens AG 2018

TRANSMIT/cylinder surface transformation

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



Face machining with TRANSMIT



Tool-center-point path through the pole

TRANSMIT is used for milling outside contours on turned parts, e.g. square parts (linear axis with rotary axis).
As a result, 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 two linear axes and one rotary axis is possible. The two linear axes are mutually perpendicular and the rotary axis lies at right angles to one of the linear axes.
TRANSMIT can be called up in different channels simultaneously. The function can be selected and deselected with a path condition (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.

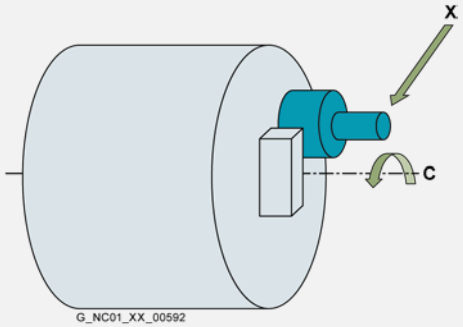
TRANSMIT through the pole is implemented in different ways:
- When traveling through the pole, the rotary axis is turned automatically through 180° when the turning center is reached and the remaining block is then executed.
- When traveling close by the pole, the control automatically reduces the feedrate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

Peripheral surface transformation is used on turning machines and milling machines. It enables peripheral surface transformation, e.g. for turned parts.

Cylinder surface transformation TRACYL can be used to manufacture grooves of any shape on the surface of cylindrical bodies, with or without groove/slot side offset. The groove shape is programmed with reference to the unwrapped, level surface of the cylinder.

TRANSMIT/cylinder surface transformation without Y axis

Option
Order code: S50
Article No.: 6FC5800-0AS50-0YB0



This option – similar to TRANSMIT and the cylindrical surface transformation of a cylindrical component – facilitates face surface machining using the Cartesian coordinate system and machining the peripheral cylindrical surface using TRACYL, if no physical Y axis is required.