SINUMERIK MDynamics
Milling expertise in a package for perfect workpiece surfaces
Contents

SINUMERIK MDynamics — milling expertise in a package 04
Advanced Surface — intelligent path control 08
SINUMERIK Operate — the new user interface 10
Extended setup functions 11
Programming to suit your style 12
Program handling and offsets with new features 14
Innovative technology cycles and functions 15
Innovative functions for five-axis machining with SINUMERIK 840D sl 17
Measuring cycles for process measuring 18
Workpiece simulation 19
Multitasking machines: cost-effective machining of complex workpieces 20
CAD/CAM/CNC process chain — well thought out from A to Z 22
Virtual NC kernel — element of the process chain 24
The SINUMERIK CNC family 25
More information about milling 26
Precise machining, perfect surfaces and high process reliability are the demands placed on the milling process chain. Particularly in tool- and mold-making, the demands relating to machining quality, cost efficiency and speed are very high.

But also in other industries, the manufacturing process must be perfectly coordinated — from the blueprint to the finished part.

Always at your side during the entire process
From simple workpieces that are programmed directly based upon a drawing — through to complex workpieces that are created in a CAD/CAM system — SINUMERIK® supports the entire machining process — from production planning to setup on the CNC, right through to production. Siemens provides a uniform and very highly-integrated system. It extends from programming directly at the CNC or from the creation of CAD/CAM data in NX CAM or other systems through to processing data in the SINUMERIK CNC system. And with SINUMERIK MDynamics, our new expertise in milling, SINUMERIK is now even more technology-oriented.

Which manufacturing strategy is used to get to the finished workpiece?
Different shapes and designs require different machining operations and the decision often has to be made as to which manufacturing strategy should be used: three axes, three plus two axes or five axes. Convex or concave free-form surfaces with uniform curvature are usually machined with three controlled axes. However, five axes are required for deep cavities or frequent changes of curvature. No matter whether three- or five axes — SINUMERIK supports every machining strategy. The ideal CNC is always available no matter the requirement.
SINUMERIK MDynamics — milling expertise in a technology package

SINUMERIK MDynamics combines milling expertise, powerful SINUMERIK CNC hardware, intelligent CNC functions and our unique CAD/CAM/CNC process chain to create technology packages for three-and five-axis milling.

Powerful functions for every industry

SINUMERIK MDynamics guarantees perfect surfaces through Advanced Surface and thus innovative motion control and optimized NC data compression. With SINUMERIK Operate, SINUMERIK provides perfect usability and quick adaptation to workpiece, tool and program handling, optimum machining thanks to flexible programming and extremely short programming times. This ensures excellent technological know-how and simple operability in all sectors: automotive and aerospace industry, power generation, medical field, as well as in the job shop or tool- and moldmaking environment. To achieve the best milling results with perfect workpiece surfaces, quality, precision and speed are required, and all of this with simple operability and a uniform process chain.
## Sectors/areas where there are milling applications

<table>
<thead>
<tr>
<th>Renewable energy</th>
<th>Tool and mold making</th>
<th>Job shop</th>
<th>Medical</th>
<th>Aerospace</th>
<th>Automotive</th>
</tr>
</thead>
<tbody>
<tr>
<td>Productivity</td>
<td>Surface quality</td>
<td>Flexibility</td>
<td>Surface quality</td>
<td>Velocity</td>
<td>Powerful CNC platform</td>
</tr>
<tr>
<td>Surface quality</td>
<td>Machining quality</td>
<td>Process reliability</td>
<td>Process reliability</td>
<td>Productivity</td>
<td>Velocity</td>
</tr>
<tr>
<td>Functionality</td>
<td>Velocity</td>
<td>Usability</td>
<td>Process reliability</td>
<td>Stock removal</td>
<td>Productivity</td>
</tr>
<tr>
<td>Process reliability</td>
<td>Usability</td>
<td>Productivity</td>
<td>Technological expertise</td>
<td>Process reliability</td>
<td>Cycle time</td>
</tr>
</tbody>
</table>

## Solution

- Powerful CNC platforms for best usability and time-optimized process sequences
- Competence in milling with SINUMERIK MDynamics
- Surface quality not requiring refinishing through intelligent path control with Advanced Surface
- New user interface SINUMERIK Operate combines simple usability and functionality
- Special high-speed cutting (HSC) functions and milling cycles
- Innovative services for networking machines, program and tool data management or condition monitoring
- Getting quickly from the blueprint to the workpiece through the integrated CAD/CAM/CNC process chain
SINUMERIK MDynamics — made for perfect milling

With SINUMERIK MDynamics, technology packages comprised of CNC hardware, intelligent CNC functions and CAD/CAM solutions are now available for SINUMERIK CNCs in the compact and premium classes.

**SINUMERIK MDynamics — SINUMERIK 828D compact class CNC for 3-axis/3+2-axis machining**

The SINUMERIK 828D integrates a technology package for 3-axis milling machines and masters all the possible drilling and milling operations. Of course, also in any swiveled workpiece planes and on cylindrical workpieces.

**SINUMERIK MDynamics — SINUMERIK 840D sl premium class CNC for 3-axis/3+2-axis and 5-axis machining**

For SINUMERIK 840D sl, you can choose between two SINUMERIK MDynamics technology packages — for three axes and for five axes. They can be expanded with additional functions to meet your needs.

More information about SINUMERIK CNCs is provided on Page 25 or at www.siemens.com/sinumerik
<table>
<thead>
<tr>
<th>Functions in detail</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>SINUMERIK 828D</strong></td>
</tr>
<tr>
<td>3-axis/3+2-axis machining</td>
</tr>
<tr>
<td>- Advanced Surface</td>
</tr>
<tr>
<td>- User memory on Compact Flash (CF) card</td>
</tr>
<tr>
<td><strong>SINUMERIK 840D sl</strong></td>
</tr>
<tr>
<td>3-axis/3+2-axis machining</td>
</tr>
<tr>
<td>- Advanced Surface</td>
</tr>
<tr>
<td>- User memory on Compact Flash (CF) card</td>
</tr>
<tr>
<td>- Spline interpolation</td>
</tr>
<tr>
<td>- Transmit and peripheral surface transformation</td>
</tr>
<tr>
<td>- Automatic measuring cycles</td>
</tr>
<tr>
<td>- 3D simulation — simultaneous recording</td>
</tr>
<tr>
<td>- ShopMill/ShopTurn-machining step programming</td>
</tr>
<tr>
<td>- Residual material detection</td>
</tr>
</tbody>
</table>

| **SINUMERIK 828D**  |
| 5-axis machining |
| - Kinematic measurement |
| - Volumetric compensation system (VCS) |
| **SINUMERIK 840D sl** |
| 5-axis machining |
| - Advanced Surface |
| - User memory on Compact Flash (CF) card |
| - Spline interpolation |
| - Transmit and peripheral surface transformation |
| - Automatic measuring cycles |
| - 3D simulation-simultaneous recording |
| - ShopMill/ShopTurn-machining step programming |
| - Residual material detection |
| - 5-axis machining package |
| - 3-D tool radius compensation |
| - Kinematic measurement |
| - Volumetric compensation system (VCS) |

- CAD/CAM/post-processor connection
- Technological/automatic measuring cycles
- High-Speed Cutting (HSC) functions
- Advanced Surface
- HMI functions
- SINUMERIK MDynamics
When using the new path control, an optimized “look ahead” function helps achieve perfect surface quality through reproducible results in adjacent milling paths, superior accuracy and increased speed. The new, optimized compressor ensures high contour precision and very high machining speeds. Intelligent jerk limiting puts less stress on the mechanical system of the machine. It enables smooth acceleration and deceleration during all dynamic responses and thus extends the service life of the machine.

A major advance is the automatic harmonization of the velocity profiles on adjacent milling paths by the CNC. It is also effective during the forward/backward line-by-line milling of contours and free-form surfaces, and results in a higher surface quality — to be more precise: in perfect workpiece surfaces. Fluctuations of the orientation are smoothed across several blocks using the “Smoothing of the orientation characteristic (ORISON)”. As a consequence, a smooth path, orientation and contour are achieved, resulting in a more harmonized traversing motion of the axes.

With its new Advanced Surface intelligent path control, SINUMERIK provides an optimal workpiece surface at a very high machining speed. The highest marks are achieved for surface quality, precision and machining speed. This enables the complete machining to be performed in only one machining operation — for a new level of productivity for HSC milling of complex parts and free-form surfaces.

Advanced Surface — the intelligent path control
Conventional CNC

Inhomogeneous surface

with Advanced Surface

Homogeneous surface

Perfect surface quality through reproducible results in adjacent milling paths

Very high machining speed through an improved velocity profile

Compressed rapid traverse motion G0 with smoothing through optimized compressor and rapid traverse movement factor

Smoothing of the orientation characteristic through orientation smoothing in 5-axis applications with ORISON
The SINUMERIK Operate user interface

**SINUMERIK Operate**, the new graphical user and programming interface, is clear and intuitive. It combines simple usability, functionality and technology for efficient operator control and programming. The result — a uniform user interface for SINUMERIK CNC.

**Designed for easy operation**

The SINUMERIK Operate graphical user interface has a clear layout, can be operated intuitively and is equipped with a large number of new, powerful functions. This combines machining step programming and high-level language programming on one system user interface — and therefore ensures quick, efficient and intuitive NC programming and production planning.

**Whether turning or milling — the look-and-feel is the same**

Intelligent SINUMERIK Operate functions provide simple support for the operator’s everyday work — for example, screenshots with the key combination CTRL+P. The optimum Windows style display is also user-oriented. The new user interface has also set a new standard in usability by means of graphical support. Animated Elements are a helpful function as they provide a graphically-animated simulation of each machining step in advance. Practical tooltips facilitate the operation.

**New — G0 Rapid traverse limiting with SINUMERIK Operate**

The “G0 Rapid traverse limiting” function integrated into SINUMERIK Operate now increases safety and reliability even more by reducing the feedrate for rapid traverse motion through program control.
Extended setup functions

A machine is setup with the aid of clear and understandable graphical support. Through the use of Animated Elements, SINUMERIK CNC sets new standards in ease-of-operation and programming in addition to technology cycles.

Extended setup functions

A large number of measuring functions make it easy to setup the machine and workpiece. This includes self-explanatory symbols on every softkey, which can be pre-configured as favorites. There are also intelligent functions available in the JOG mode — for tool and workpiece measurement and for touch trigger and non-switching probes.

Measuring in JOG mode — workpiece setup for three- and five-axis machines
Edge alignment using two holes.

Measuring in JOG mode — workpiece setup for five-axis machines
Plane alignment using three measuring points with switching probes and two rotary axes.

Swiveling in JOG mode — setup
Complex workpieces can be machined with simple and fast handling in one clamping operation. Different kinematic types can easily be setup. Swiveling can be performed axis-by-axis and directly including optional coordinate rotation.
Programming to suit your style

SINUMERIK Operate offers unique programming methods for every application. This includes the ShopMill/ShopTurn workstep programming, as well as the SINUMERIK high-level language with programGUIDE. ISO-code programming is also supported.

ShopMill workstep programming

The ShopMill and ShopTurn workstep programming is the tailor-made programming solution to machine single parts and small batches. In addition to programGUIDE, ShopMill provides unique workstep programming for extremely short programming times —

- Clear display of machining steps without G-code knowledge
- Simple linking of technology functions with geometric elements
- Dynamic display of the workpiece during programming

Functions:

- User-friendly feature with graphical workstep editor
- Perfect input help via Animated Elements
- Dynamic broken-line graphics for complete workpiece and cycle screens
- Large selection of standard drilling and milling cycles
- Geometry computer facilitates the input of workpiece contours
- Cycles for contour milling with residual material detection
- Powerful 3D CNC simulation

In addition to programGUIDE, ShopMill offers unique workstep programming, for example — centering, deep-hole drilling and tapping on a position pattern.
programGUIDE, the SINUMERIK high-level language, has been developed for DIN/ISO G-Code programming with advanced cycle support and ensures maximum flexibility and short machining times. Ideal for use with medium to large batch sizes, programGUIDE guarantees maximum productivity and flexibility in programming combined with innovative technology and machining cycles —

- CNC language with high-level programming commands
- programGUIDE with graphical cycle support screens including tooltip (context-related brief information)
- Online ISO dialect interpreter is available

**Functions —**

- Fully flexible ASCII editor for CNC high-level language commands
- Perfect input help via Animated Elements
- Dynamic broken-line graphics for cycle screens
- Large selection of standard milling, drilling and turning cycles
- Contour calculator facilitates the input of workpiece contours
- Cycles for contour milling and turning with identification of residual material
- Powerful 3D CNC simulation
Program handling and offsets with new features

Tool and workpiece offsets, program management and file handling have been optimized. The tool list has a configurable display. This can be operated intuitively thanks to context-dependent functions and self-explanatory symbols. The work offset overview clearly displays all the relevant data so that everything can be seen at a glance.

Program management

- Sub-directories for workpieces on local disk drives
- Direct access to external storage media including network drives
- Part programs with plain text and a maximum of 24 characters
- Time savings through easy data transfer and simple program handling
- Quick program check through program preview
- Large programs can also be easily edited in the NC memory or on external storage media, including program preview
- Graphical files such as *.jpg, *.png, *.bmp and PDF are displayed in the Program Manager
- “Multiple clamping” function to generate a complete program from ShopMill programs for individual machining operations

Tool management

- Tool types shown as symbols
- Tooltips available (context-related brief information)
- Tool and magazine data on one screen including details
- Tool name as plain text or number with a maximum 24 characters
- Efficient tool data management including all details and replacement tool handling

Work offset

- All active, settable and programmable work offsets listed on one screen
- Offset rotation of the active swivel data set
- Offset display of the active swivel data set
Innovative technology cycles

SINUMERIK cycles help the user program workpieces even quicker and more easily. Even in complex machining tasks, comprehensive machining steps can be solved faster and more easily by using innovative cycles — such as trochoidal milling and plunging for cutting with low cutting pressure and short delay. A selection of possible technology cycles is provided in the following examples —

Machining grooves

Trochoidal milling
- Preferred high-speed cutting (HSC) strategy for roughing materials that are difficult to cut and for machining grooves
- Simple parameterization via dialog boxes — roughing, pre-finishing, finishing, base and edge finishing

Plunging
- Preferred strategy for roughing and opening deep grooves or cavities with long tools
- Simple parameterization using dialog boxes — roughing, pre-finishing, finishing, base and edge finishing

Engraving cycle

- The SINUMERIK engraving cycle supports the input and programming of engraving directly on the machine
- Shorter times for prototype production and small series
- Graphical support when engraving any type of text, date and time or quantity (serial number)

Multi-edge cycle

SINUMERIK cycles support the simple programming of standard geometries —
- Standard cycle for the milling of a polygon with any number of edges and width across flats
- Graphical support for standard parts
Innovative technology cycles and functions

Whether in the 2-D area with the contour calculator, during high-speed cutting or in 5-axis applications – SINUMERIK cycles help you to get from the blueprint to the finished part even more quickly.

High-speed setting — CYCLE832
The SINUMERIK machining cycle for roughing/pre-finishing and finishing as simplified high-speed setting cycle for all milling applications, particularly in tool- and mold-making, as well as in the aerospace industry.

Swivel cycle — CYCLE800
Complex workpieces can be easily programmed, which leads to increased productivity. Simple parameterization using dialog boxes: swivel modes — direct, axis-by-axis, angle of projection, solid angle.

Contour calculator — user-defined contours and geometry programming
The SINUMERIK contour calculator supports the input and programming of any contour directly at the machine —
- Up to 256 geometry elements
- Undefined elements can be calculated automatically
- Display can be scaled automatically
- Available in programGUIDE and ShopMill
- CAD reader — offline DXF converter available for the PC
Innovative functions for five-axis machining with SINUMERIK 840D sl

SINUMERIK has powerful functions that significantly simplify the entire sequence of multi-axis programming and machining. These include, for example, tool center point programming — TRAORI or orientation smoothing when milling with ORISON. Whether handling or machining complex workpieces, SINUMERIK CNC supports all kinematics.

Innovative compressor functionality
Through homogeneous transitions at the block boundaries, the innovative online compressor provides better results for three- and five-axis simultaneous milling.

Orientation smoothing for five-axis milling with ORISON
Fluctuations of the orientation are smoothed across several blocks using the “Smoothing of the orientation characteristic (ORISON)”. As a consequence, a smooth path, orientation and contour are achieved, resulting in a more harmonized traversing motion of the axes.

3D tool radius compensation CUT3DC — CUT3DF
SINUMERIK supports the handling and machining of complex workpieces, for example, during circumferential milling and face milling with tool radius compensation in five-axis applications.

Spline interpolation
ASPLINE/BSPLINE/CSPLINE with integrated NC functionality for high-level CAD/CAM programming commands
Measuring cycles for process measurements — a new feature in the SINUMERIK Operate style with Animated Elements

SINUMERIK supports you when measuring with a comprehensive range of practical cycles. With these measuring cycles, you can measure workpieces, tools and the kinematics in the process with graphical support. SINUMERIK measuring cycles are now also available in the SINUMERIK Operate style with Animated Elements in the various programming types — such as programGUIDE or ShopMill.

Innovative measuring cycles for workpiece and tool measurement during automatic operation

SINUMERIK measuring cycles in the new SINUMERIK Operate style — with Animated Elements — guarantee the quality of the produced parts by automatically making measurements at the machine:

- Powerful cycles for automatic workpiece and tool measurement
- Automatic measuring cycles available in programGUIDE and ShopMill
- Input screens with graphic support and automatic measurement result logging

Innovative measuring cycles for measuring kinematics — CYCLE996 (for SINUMERIK 840D sl)

The “Measure kinematics” cycle (CYCLE996) has been developed to measure multi-axis kinematics. It is easy-to-use and does not require expensive measuring equipment. The calibration ball is measured with a 3D probe and the “Measure kinematics” cycle in three ball positions for each rotary axis. The cycle can be called directly from the NC program. Compared to conventional measuring, this can save a lot of time with the highest level of measuring accuracy.
Workpiece simulation and simultaneous recording

Workpiece simulation with SINUMERIK CNC provides more safety and checking options during workpiece programming. Simultaneous workpiece recording in the automatic mode also increases safety.

CNC simulation — workpiece simulation for multside- and five-axis machining

Increased safety through 3D simulation with 3-plane view and volume model of the finished part — including simultaneous recording in the automatic mode. Blanks can be input, such as rectangle, tube, cylindrical spigot, polygon.

Simulation speed can be controlled using softkeys — single block mode / start / stop / zoom. The number of checking options increases as the machining time is automatically calculated.

Also for five-axis workpieces, SINUMERIK CNC offers optimum support, reliability and safety during programming and workpiece simulation. With SINUMERIK 840D sl, also during actual machining.

Simulation offers a variety of different workpiece views. In this way, programming can be checked immediately. During simulation, you can zoom into and view individual sections — with SINUMERIK 840D sl also on swiveled planes and for five-axis simultaneous machining.

Program visualization — Quick Viewer / faster moldmaking view with 3D preview — even for large NC programs

The new "Quick Viewer fast moldmaking view” function allows a significantly faster overview of a workpiece and the corresponding NC program to be achieved. Even the largest NC programs and part programs are converted in a relatively short period of time into a 3D preview of the workpiece.

Simultaneous recording in the automatic mode

Simultaneous recording of the machining process can be activated and displayed.
Cost-effective machining of complex workpieces on multi-tasking machines

Complex workpieces require cost-effective machining techniques and innovative CNC solutions. SINUMERIK CNC supports multi-tasking machines when machining workpieces in one operation. SINUMERIK provides all of the necessary functions for machining components on one machining center — even when changing between different technologies. The SINUMERIK Operate graphical user interface provides integrated turning functions for milling, milling functions for turning and supports B-axis kinematics — including multi-tools, thus further improving the cost-efficiency of the manufacturing process.

Simple operating control of multi-tasking machines

As a standard, SINUMERIK CNC has innovative functions for the operation of multi-tasking machines, such as mill-turning and turn-milling machines. These make machining simple and cost-effective in one machining operation. The SINUMERIK Operate graphical user interface offers integrated turning functions for milling and integrated milling functions for turning, supplemented by innovative measuring cycles in the Animated Elements design. The usability and look-and-feel are completely identical — also when switching between different machining technologies. The tool manager facilitates simple handling of turning and milling tools as well as multi-tools in a standard user interface. Programming in programGUIDE has been supplemented by functions such as the alignment and positioning of turning and milling tools including turning and milling cycles — as well as contour programming. Multi-tasking programs can be simulated, therefore offering even higher machine efficiency.

Operator-friendly solution for multi-channel machines

The multi-channel display is one of the many functions provided for easy operation. During milling, the second channel can be used to control and display handling modules or the tool change. This increases the flexibility and cost-efficiency during production.

With the programSYNC option, multichannel machining can be synchronized simply and efficiently. Multichannel machining can also be visualized through the use of the simulation with the SINUMERIK 840D sl. This means, multichannel machining operations can be programmed even more efficiently:

- Creating the structure of the part programs
- Filling individual machining steps (blocks)
- Simulating part programs
- Running-in part programs (channel-by-channel or spindle-by-spindle)
- New block forming function — programSYNC including the block forming option
**Efficient tool management**

To ensure efficient tool data management — including all of the details, SINUMERIK Operate is equipped with extended tool management for milling, drilling and turning tools. All tool and magazine information, including the associated details, are displayed on one screen. In addition, all tool parameters are provided for turning. All tool types including turning tools are transparently displayed as symbols. SINUMERIK also supports the use of complex tools such as multi-tools, which eliminates the need to change tools and also increases the productivity when using multi-tasking machines.

**Maximum programming productivity and flexibility**

With programGUIDE and ShopMill/ShopTurn, SINUMERIK Operate provides innovative functions for complex workpiece programming and ensures maximum productivity and flexibility thanks to innovative technology cycles, including turning.

**Innovative functions for programming complex workpieces with programGUIDE**

**Kinematic cycle**

Aligning turning and tools and swiveling the coordinate system for turning or milling.
In the production of milled parts with free-form surfaces, high quality demands are placed on precise machining, perfect surfaces and high process reliability in high-speed cutting. Siemens PL provides comprehensive solutions for the complex tasks associated with the product lifecycle management of machine tools. SINUMERIK also perfectly supports other CAM systems.

Well thought out from A to Z —
the CAD/CAM/CNC process chain

Efficient processes with SINUMERIK CNC

During workpiece production, special focus is placed on the workpiece-related process chain between the initial blueprint and the finished part.

This process chain encompasses —
■ The computer-supported product development at the CAD/CAM level
■ The part program generation
■ The optimization of the post-processor, the NC program and the CNC parameters
■ Online and offline simulation for monitoring and optimizing the manufacturing process on the PC
■ The optimized, highly efficient workpiece machining on the machine tool

With NX, the CAD/CAM system for the entire design and manufacturing process, Siemens provides an open and flexible 3-D system. This involves the development, design, generating drawings, simulation and manufacturing. NX enables 3-D and 2-D design in full, partial and non-parameterized form. The considerations that have to be made during the CAD design process concerning the manufacturing properties — for example, locally-required manufacturing precision, fits etc. are recorded directly and assigned to the contours or components. This means that all of the essential information is available in the CAM phase for production planning and can be used, controlled by the software, when generating the part program.
Optimized CAM output for SINUMERIK

A post-processor generator is also available in NX CAM, which allows a post-processor, tailored precisely to the target machine tool, to be generated. Not only the type and version of the SINUMERIK CNC are taken into account, but also the installed software version together with the options provided on the CNC and machine. The part program output by NX CAM via such a post-processor uses all of the productivity-enhancing functions, cycles and options of the machine.

Another significant benefit provided includes the functions, cycles and options available on the CNC. These are already parameterized at the CAM level for the respective machining step (e.g. roughing, prefinishing or finishing).
Virtual NC Kernel — the element of the process chain

Programming errors are costly. Therefore, the objective must be to detect errors as early as possible and reliably correct them before real production starts.

Siemens provides the “Virtual Machine” software package for machines equipped with SINUMERIK CNC systems. This includes the virtual NC kernel (VNCK) for a control-related image of the real control, the graphic simulation system (RealNC or NX CAM) for the stock removal and collision calculation as well as the original SINUMERIK Operate to operate the virtual controller and the machine.

Simulation with “Virtual Machine” enables numerically controlled machining processes to be very realistically simulated on a PC. The use of the simulation reliably excludes programming errors and collisions. This significantly reduces setup times.

The “Virtual Machine” provides a new perspective for process optimization in the production environment. Thanks to numerous benefits, such as efficient programming, shorter setup times and a minimized error risk, “Virtual Machine” opens up completely new dimensions for the simulation of production processes with regard to productivity. More information can be found online at www.siemens.com/plm
The SINUMERIK system platform consists of various products that meet the different demands placed on machine tools — the SINUMERIK 828D for compact machines and the high-performance SINUMERIK 840D sl for sophisticated solutions.

### The SINUMERIK CNC family

#### Two powerful systems for milling

<table>
<thead>
<tr>
<th>Compact class CNC</th>
<th>Premium class CNC</th>
</tr>
</thead>
<tbody>
<tr>
<td>SINUMERIK 828D</td>
<td>SINUMERIK 840D sl</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Machine applications</th>
<th>Compact, powerful and simple CNC system</th>
<th>Open, flexible and powerful CNC system</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design</td>
<td>Compact panel CNC in horizontal and vertical operator panel layout</td>
<td>Modular, scalable CNC universal control</td>
</tr>
<tr>
<td>Degree of expansion</td>
<td>Up to 6 axes/spindles in one machining channel</td>
<td>Up to 31 axes/spindles in 10 machining channels</td>
</tr>
<tr>
<td>Field of application</td>
<td>3 axes and 3+2 axes</td>
<td>From 3 axes to 5 axes simultaneously</td>
</tr>
</tbody>
</table>

### SINUMERIK — your benefits at a glance

- Advancing quickly from the CAD/CAM system to the workpiece through integrated solutions
- Cost-effective manufacturing with high-speed CNCs
- Powerful CNC system with dynamic drives and motors
- SINUMERIK MDynamics technology packages for best milling results
- Quick and easy to operate thanks to the new, intuitive user interface SINUMERIK Operate
- Higher cost-efficiency through special high-speed cutting (HSC)
- Simple and intelligent setup functions
- Easy-to-use, graphic programming and simulation
Siemens is the expert when it comes to every technological aspect of milling. Visit us at one of our technology and application centers or attend one of our many CNC training classes.

Global technology competence — Technology and Application Centers

Our Technology and Application Centers, also known as TACs, are important information centers and meeting places for machine tool builders and end users. They provide the opportunity of demonstrating our systems, solutions and services to customers and partners within the framework of training courses and events. More information on the technologies and events presented at our TAC can be found online at www.siemens.com/cnc4you

SinuTrain — practical CNC training

With SinuTrain, we have developed a realistic training system for the SINUMERIK CNC system. In addition to the traditional range of training courses and workshops, we also offer services such as online learning modules, learning software and technical books. Practical and perfectly matched to the respective training level, we offer basic courses, programming and operator training through to professional training — everything that you require to achieve your learning goals. More information is provided online at www.siemens.com/sinutrain

Documentation made easy

With DOConWEB, we have provided a unique opportunity to search our documentation online, either with a plain text search, index search or via the table of contents. Individual documentation can also be compiled quickly, easily and for a specific requirement.

Visit www.siemens.com/automation/doconweb for more information.
The information provided in this brochure contains merely general descriptions or characteristics of performance which in actual case of use do not always apply as described or which may change as a result of further development of the products. An obligation to provide the respective characteristics shall only exist if expressly agreed in the terms of contract.

All product designations may be trademarks or product names of Siemens AG or supplier companies whose use by third parties for their own purposes could violate the rights of the owners.