

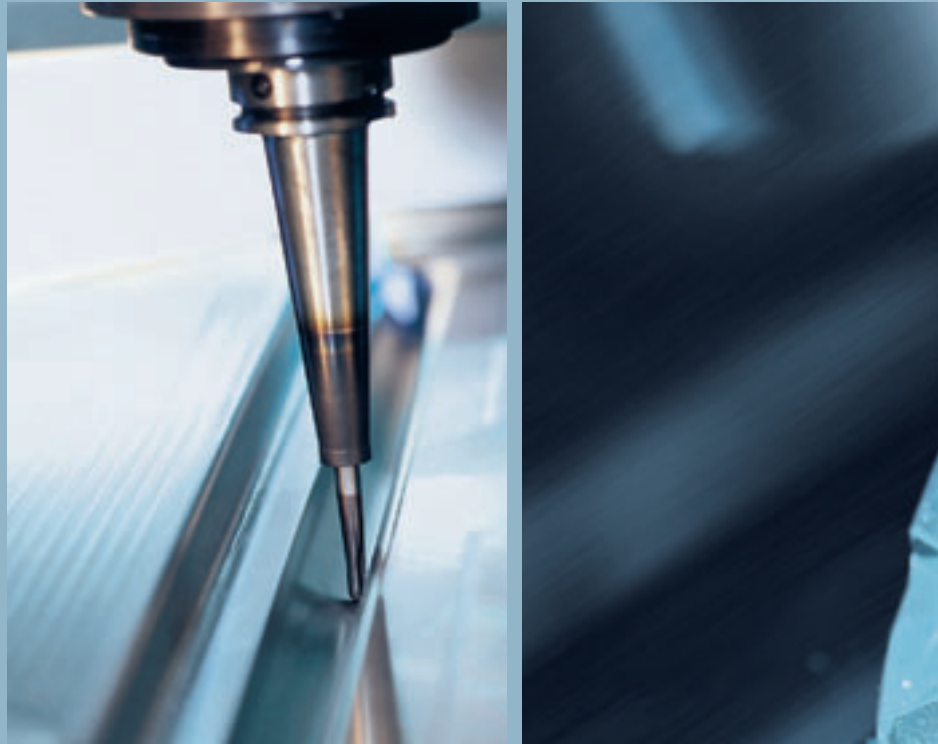
# From the idea to the precise form

Perfect CNC solutions for  
tool- and moldmaking

sinumerik

[www.siemens.com/sinumerik](http://www.siemens.com/sinumerik)

**SIEMENS**



## Productivity in motion

Whether in the automotive industry, consumer goods or power engineering, the rising demand for different molds and more varied design has led to enormous growth in tool- and moldmaking. For users, it is increasingly important to get from the initial idea to the finished mold quickly and precisely if they want to stay ahead of the competition. High-speed machining with SINUMERIK® reduces the demands to a common denominator. You'll realize a time saving of up to 20% with the SINUMERIK control solution for five-axis machining as compared to other solutions and with outstanding precision and surface quality.



### **Success factors for tool- and moldmaking**

Crucial to success is the consistent use of CAD/CAM systems and an optimized CAM-PP-NC interface for activating the SINUMERIK control functions for high-speed machining. This makes SINUMERIK CNC the perfect automation solution for tool- and moldmaking.

### **Functions for more performance**

SINUMERIK computer numerical controls possess a range of outstanding functions that simplify day-to-day tasks such as the production of models, metalforming tools or injection molding tools many times over. To this end, we work closely with our customers and partners in tool- and moldmaking – from development, through model-making, moldmaking, sub-contracting and prototype building.





## SINUMERIK – the automation platform for metal cutting

With SINUMERIK, Siemens offers an integrated automation system with products, solutions, and services for equipping machine tools – for use in the workshop or in industry. The SINUMERIK platform offers user-friendly components, from the CNC control, through the drive and the motor, to the communication infrastructure. User interfaces, safety solutions and services are precisely coordinated with the product range.

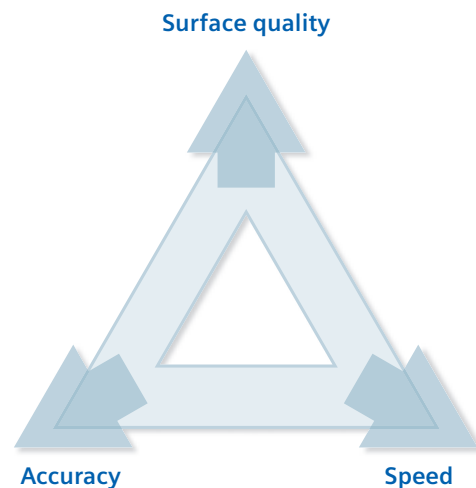


### Made-to-measure for tool- and moldmaking

We offer you exactly the right solution for your production tasks in tool- and moldmaking based on the SINUMERIK platform. The requirements of many sectors and technologies have already been integrated into the equipment at the development stage for maximum efficiency and performance. SINUMERIK enables a simple, modular system configuration, flexible operating concepts with a uniform structure for operation, programming and visualization, as well as fast startup.

### Error compensation for precise machining

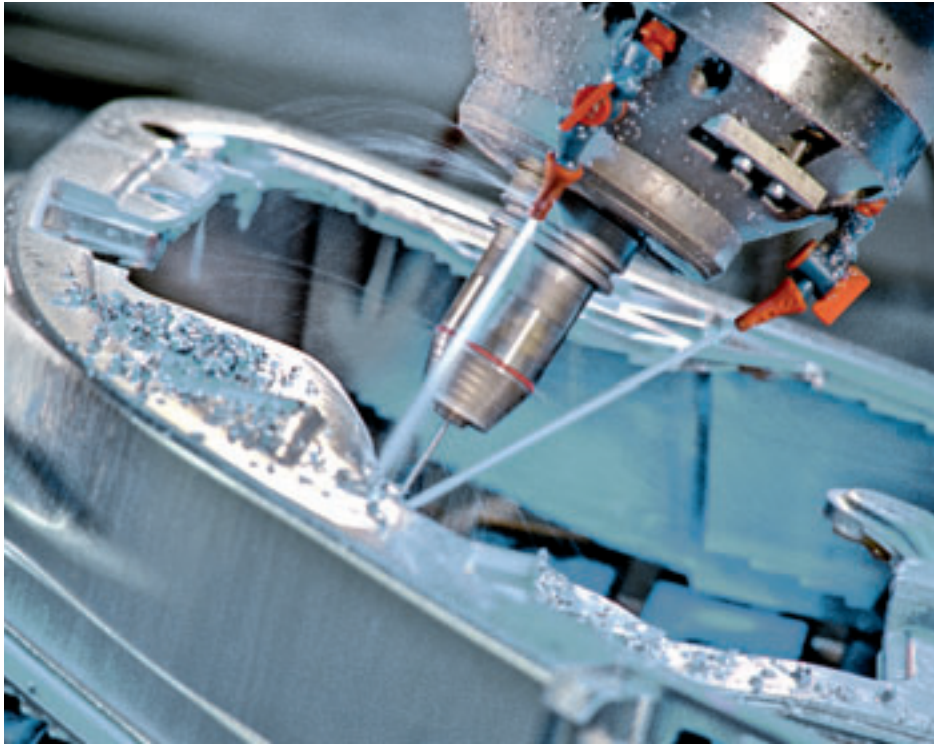
Whether you're talking Advanced Position Control (APC), quadrant error compensation, multi-dimensional pass compensation or temperature compensation, we have decisively improved SINUMERIK to compensate for errors and provide even more precise machining. The result is even higher surface quality and increased productivity.





## Finding the right production strategy for the tool and mold component

Different molds require different machining operations. This often presents you with the decision of which production strategy to use: three axes, three plus two axes or five axes? Evenly convex, curved free-form surfaces are usually produced with three controlled axes. However, in the case of deep cavities or frequent curve changes, five controlled axes are required. Whether it's three or five axes, SINUMERIK 840D supports all your machining strategies.



Left to right:  
Cutting with three axes,  
cutting with three + two axes,  
cutting with five axes

### Three axes with constant orientation

In the case of controlled path axes X, Y, Z, the orientation of the cutter does not change along the entire cutting path. Cutting conditions at the tool tip are never optimal.

### The formula: three + two axes

In the case of three controlled path axes X, Y, Z plus two fixed rotary axes, e.g. A and C of a table, the orientation of the tool or the table position can be changed, for example, by retooling. Cutting conditions deteriorate the further the tool tip moves up or to the side on the workpiece. To achieve optimal cutting conditions, the table is swiveled. To machine a free-form surface completely, it is often necessary to swivel the table several times in different directions.

### Optimal cutting conditions with five axes

In the case of three controlled path axes (X, Y, Z) and two controlled rotary axes (e.g. A and B), the clear benefit is that the orientation of the tool over the entire path can be optimally adjusted synchronously with the linear movement. Therefore, optimal conditions apply on the entire path.



Five-axis machining of complex milled part

# Setting up and programming tool and mold components

SINUMERIK supports you in the setup of the machine, that is, the workpiece and the tools. The functions in manual mode enable you to set up the machine quickly and correctly. Particularly, these functions calculate the workpiece position in the machine along with maintenance and measurement of the tools used. For programming with SINUMERIK, the DIN/ISO editor is available for the creation of DIN/ISO programs on the machine, as well as for the correction or modification of externally created DIN/ISO-programs.

## Calibrating the tool

In setup mode, you can calculate the correction values for the tools directly on the machine. SINUMERIK supports the following variants:

### Variant 1: Automatic measurement with probe ball (table contact system) manually or switching

1. Calibration of the probe ball. This calculates the position of the probe ball with reference to the machine zero point.
2. Following calibration, manual traverse of the tool to the area of the probe.
3. Start of the measurement operation for the length and diameter of the tool. SINUMERIK probes the tool automatically to the dynamometer and calculates the tool correction data from the position of the workpiece carrier reference point and the probe ball.

### Variant 2: Manual measurement with scratching of the tool to known workpiece geometry

1. Manual traverse of the tool to a known reference point in X, Y or Z direction to calculate the tool dimensions.

2. SINUMERIK automatically calculates the tool correction data from the position of the workpiece carrier reference point and the reference point.

### Variant 3: Setup/measurement with SINUMERIK functions

#### Three-point measurement

With the help of three-point measurement, you can swivel to the machining level of the workpiece. For this purpose, three-point measurement captures the exact angle of an inclined surface in space. The reference point measurement of the X/Y axis and the Z axis enable the coordinates to be adjusted to the workpiece for further machining.

#### Measuring cycle function for simple measuring

CYCLE996, the new SINUMERIK measuring cycle, is suitable for simple measurement or calibration of new or existing multi-axis kinematics. The operator only has to traverse to the start coordinates for measurement on the calibration ball and then call the measurement cycle. The function also handles the rest and it automatically optionally writes back the values direct to the control.



**Flexible programming:  
FRAME concept**

With frames, one Cartesian coordinate system can be transferred to another. This functionality of the SINUMERIK 840D is also used to move, rotate, scale or mirror coordinate systems. We distinguish here between settable and programmable frames. Their interaction in a frame chain allows extremely flexible workpiece programming.

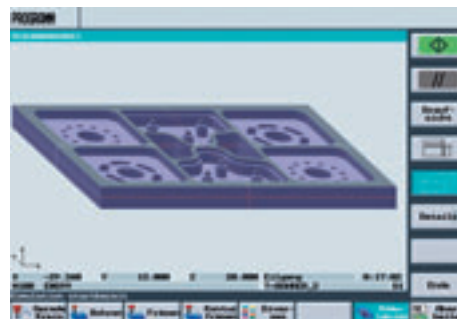
**Defined retraction: TOROT**

An absolute highlight of the frame concept is TOROT. The command refers to an internal system FRAME, whose Z axis always agrees with the current tool orientation. If a tool breaks while in use, for example, it can be recovered in manual mode simply in the tool direction. This is also supported by the handwheel function.

**Simple and fast programming with  
ShopMill**

Graphical programming makes it possible to program individual parts and small batches quickly. Without knowledge of

DIN/ISO, the production steps can be quickly converted into machining plans that clearly represent all the technological information of the individual work steps. Dynamic help screens enable input without documentation. As an alternative to graphical programming, ShopMill also allows DIN/ISO programming. The simulation facility ensures maximum process safety – based on actual tool data. The workpiece is optimally represented here in high-resolution 3D graphics.



# Everything you need for the perfect mold

Proven functions support the setup and programming of tool- and moldmaking. They contribute to fast and exact test and production sequences.



## **A quick look into the NC program: Visualization with Quick View**

The visualization system gets an improved inspection process underway. The Quick View function supports the machine operator with a fast and clear preview of the visual assessment and optimization of parts quality, thus minimizing downtimes.

### **The functions of Quick View**

Quick View allows the visualization of all moldmaking part programs containing G01 blocks. The following functions are also available:

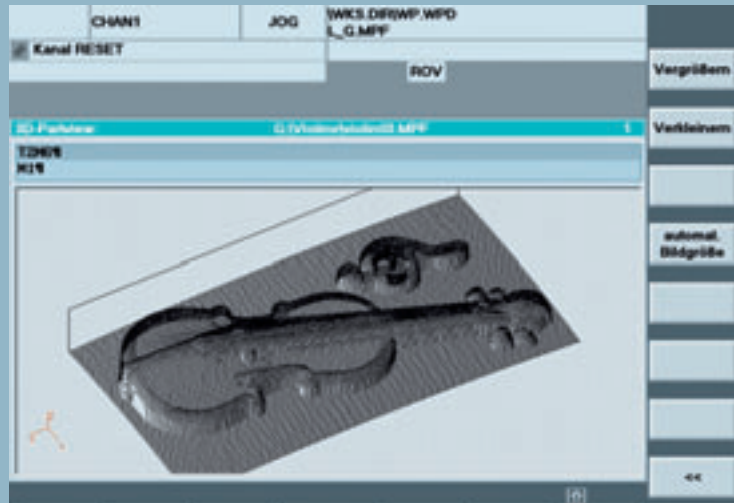
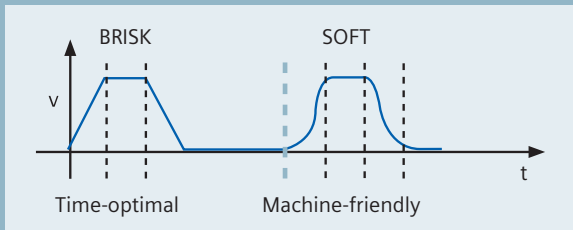
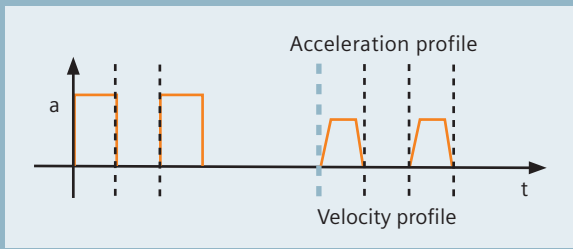
- Find a specific block
- "Zoom in/out" on the screen section
- Shifting and turning
- Distance measurement between two points
- Editing of the displayed NC part program

## **Full speed ahead – high-speed machining with SINUMERIK**

The main demands that high-speed machining places on a CNC control result from the three-dimensional surfaces that are complex to machine, high cutting speeds, and constantly rising demands in quality. The innovative CNC platform SINUMERIK is crucial to continuous progress in high-speed machining.

### **Continuous-path mode: speed profile in accordance with acceleration ability**

To avoid contour violations, SINUMERIK 840D ensures that the speed profile is always oriented around the acceleration ability of the drives – with Look Ahead G642. Depending on an axis-specific tolerance window, this function can smooth discontinuities at block transitions, thus avoiding mechanical oscillation of the machine and improving the surface quality of the finished workpieces.



Quick View: Complete and detailed view of the surface area (test sample)

**Acceleration software:** BRISK – without jerk limitation: Stepped acceleration of the path axes  
SOFT – with jerk limitation: jerk-limited acceleration of the path axes

### Acceleration software with and without jerk limitation

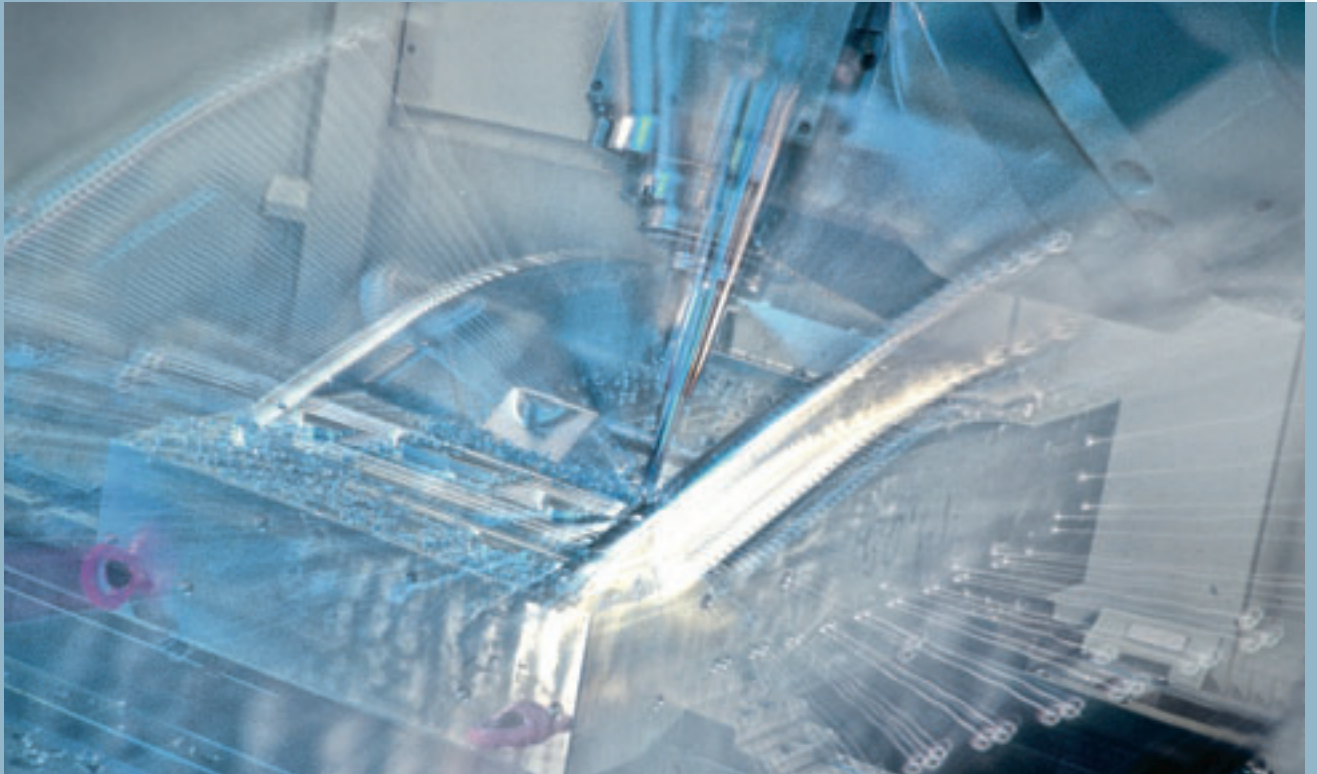
To make acceleration as machine-friendly as possible, the acceleration profile of the axes can be influenced by the commands SOFT and BRISK. If SOFT is activated, the acceleration behavior does not change in steps, but is instead increased by means of a linear characteristic. This is easy on the machine and has a positive effect on workpiece surface quality since less resonance is created in the machine. BRISK, on the other hand, enables stepped working in the course of acceleration. The axes slide traverse at maximum acceleration until they reach feedrate.

### Makes difficult decisions really easy: high-speed setting cycle

Functions like spline compressor, jerk limitation and Look Ahead G642 significantly improve the characteristics of the machine. The parameters of the relevant functions – whether speed, precision or surface quality – are adapted in accordance with the relevant optimization criteria. This is an often difficult adjustment that the high-speed setting cycle significantly simplifies.

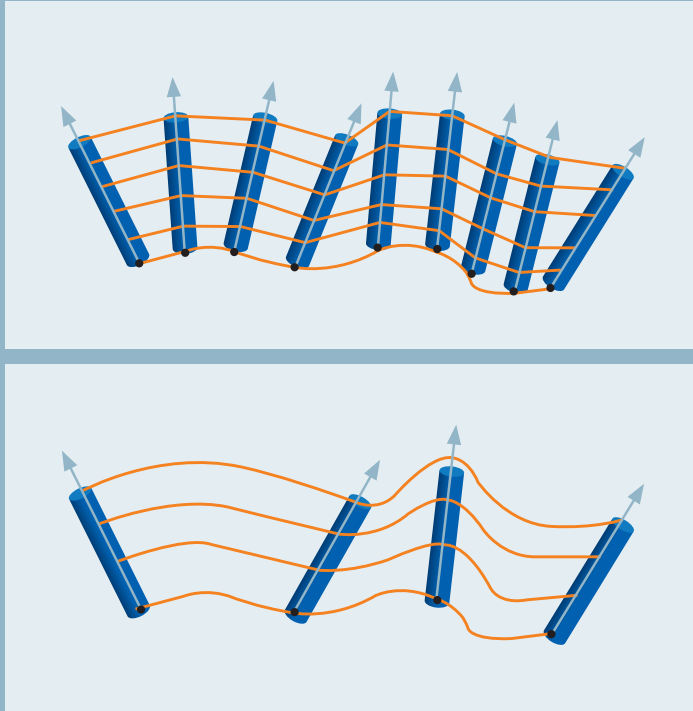
### For higher path velocities: Spline interpolation

In contrast to linear interpolation, spline interpolation ensures block transitions with constant curvature. This avoids velocity jumps in the relevant machine axes and also undesired resonance in the machine elements. Improvements in traverse rate and surface quality are thus enabled.



## Maximum functionality for five-axis machining

Five-axis machining enables complete machining in one clamping. This saves time, increases accuracy and achieves high surface quality.



Compressor technology  
Optimizes traverse rate and surface quality

### For all machine kinematics: Integral five-axis transformation

One outstanding feature of SINUMERIK 840D is the integral five-axis transformation. With this dynamic function, programming takes place in the workpiece coordinate system or with reference to the tool center point – regardless of whether this is a rotary table/swivel head, rotary table/inclinable table or fork head. In other words, the identical parts program can be executed on different machine kinematics.

### Benefits for users: dynamic transformation

Another highlight, in conjunction with the integral five-axis transformation, is the high-level language command TRAORI (transformation orientation). The benefits: you can switch online between different machine kinematics and you can change tool lengths without modifying the NC program. Any desired change to the tool orientation in the workpiece coordinate system will also be carried out automatically.

### Made-to-measure – orientation interpolation

One version of orientation interpolation is large-radius interpolation. The SINUMERIK 840D supports this with automatic switching from large-radius interpolation to linear interpolation within the polygon. The SINUMERIK 840D solves transitions from tool positions (three-axis) to a new path orientation (five-axis) through predictive orientation of the rotary axes. The result is significant optimization of machining speed and surface quality.

### No costly postprocessor runs: 3D tool radius correction

Along with tool length correction, SINUMERIK 840D also includes three-dimensional tool radius correction. This dispenses with costly postprocessor runs and the insertion depth of the tool can also be defined with reference to the programmed contour using the ISD parameter. This way, you can re-use expensive tools.

### Simply user-friendly: RTCP tool orientation in manual mode

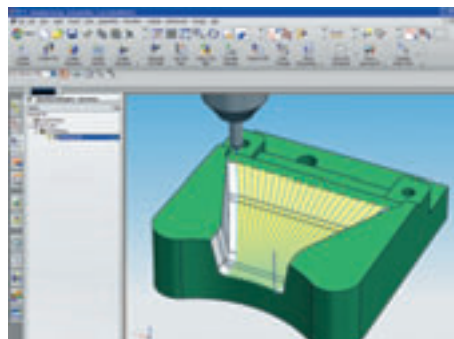
Rotate Tool Center Point (RTCP) is a component part of the dynamic five-axis transformation. With RTCP, users can change the tool orientation – while the tool point remains at one position. This makes operation simple and user-friendly.

### Online conversion in Splines: compressor technology

An outstanding feature of SINUMERIK for optimizing traverse rate and surface quality is the integral compressor. Based on linear blocks, it can be used to convert programs online into splines. This functionality can be activated in a user-friendly way in the parts program. It has been expanded so that the spline compressor also smoothes the orientation interpolation of a five-axis machine.

# Well thought-out from A to Z: CAD/CAM integrated system

The consistent use of CAD/CAM technology results in time savings in the process chain – from data acceptance to production. The integrated CAD/CAM/CNC chain pushed by us is now firmly established everywhere: the CAD/CAM systems of almost all well-known manufacturers can output the Siemens interpolation format. First-class solutions for high-speed milling are created in conjunction with specially designed, turnkey postprocessors from leading vendors.



CAD system

CAM system

Post-processor



### Modern NC programming software for modern machine tools

An integrated CAD/CAM/CNC process chain that is supported by Siemens and is becoming more widely used exercises great influence on machining quality and throughput. Modern CAD/CAM systems are used here. With solutions from product planning and development, to design and production planning, we enable interoperability and global cooperation through the use of the most up-to-date software technology and industrial standards. Together with Siemens A&D PL (formerly UGS), Siemens offers customized system development, system integration and system and process management.

Modern software systems for NC programming of CNC machines demand not only powerful functions for fast creation of NC programs, but also realistic virtual machine simulation. This can drastically reduce setup times and achieve significant value added from machine investment at a fraction of the costs of machine and tool procurement.

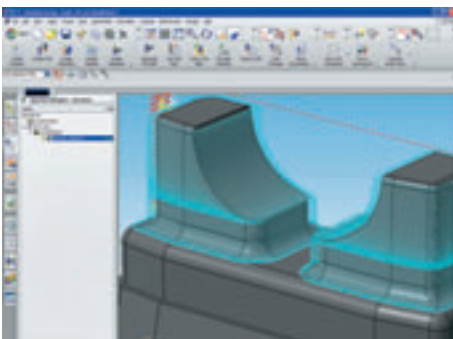
### NX CAM: universally applicable

The CAM modules for NC programming in NX address the most diverse machining methods from three-axis cutting to five-axis cutting. All tasks such as process planning, NC code generation and machining simulations or optimization always refer directly to the digital models of the end products thanks to the integral CAD/CAM/CAE system architecture of NX. Interface errors are removed, the association between product model and finished workpiece is guaranteed and the costs for model changes are minimized. In addition,

knowledge-based engineering (KBE) functions of NX can be used for planning processes of the production system to automate frequently recurring tasks.

### Realistic machining simulation

The simulation feature integrated in NX CAM makes external software superfluous for this task. It is based on the actual NC code so that the sequence of the actual NC program is simulated. The NX controller-driven machining simulator thus combines the NX CAM software with the original software model of the SINUMERIK 840D. This combination offers a new level of accuracy in the digital simulation of machine operation.



# Virtual NC Kernel (VNCK) – element of the integrated system

In tool- and moldmaking, programming errors are an expensive matter. The goal is to detect these as early as possible and, ideally, to reliably eliminate them before the start of actual production.

## Simulation with VNCK

Powerful simulation systems not only test the parts programs for collision and surface errors, but they can also calculate the machining time – an important parameter in planning and optimizing production. It is a general rule here that the more precisely the simulation system represents the control behavior, the more meaningful the results achieved with it will be. This is why we provide the possibility, with SINUMERIK, of integrating VNCK into reputable simulation software.

## Benefits of virtual simulation based on VNCK

Control by VNCK – a complete image of the CNC software and adapted to the actual configuration of the machine tool – achieves a previously unknown level of agreement between the virtual simulation and the real world. This results in a general improvement in the quality of the NC programs and it significantly reduces

the setup time and test time on the machine. Another benefit is that, with this approach, machining time can be calculated with extreme precision in advance.

## Perfect communication: Fast-Ethernet network

SINUMERIK 840D is ideally equipped for high-speed data exchange between the control and the host computer. The Ethernet TCP/IP interface required for this is onboard as standard.

## Ideal for large volumes of data: processing directly from the SINUMERIK 840D hard disk

It makes more sense to process large NC programs directly from the hard disk. That's why the Ethernet TCP/IP interface of the SINUMERIK 840D is especially valuable for data transfers to the hard disk.



Siemens AG  
Automation and Drives  
Motion Control Systems  
P.O. Box 31 80  
91050 ERLANGEN  
GERMANY

[www.siemens.com/sinumerik](http://www.siemens.com/sinumerik)

Subject to change without prior notice 11/07  
Order No.: E20001-A740-P610-X-7600  
Dispo 06311  
21/8332 MK.MC.10.STAN.52.8.01 WS 11075.  
Printed in Germany  
© Siemens AG 2007

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.