

NX CAM – Siemens Sinumerik Post Builder

Enable content-rich data exchange between NX CAM and the Sinumerik controller to improve machining efficiency

fact sheet

Siemens PLM Software

www.siemens.com/plm

► Summary

Achieve new levels of efficiency in mold and die, complex machining and machinery manufacture by activating the advanced features of your Sinumerik 840D controller. The new NX CAM Sinumerik Post Builder makes best use of the Sinumerik controller's powerful functions to simplify and speed day-to-day tasks in part manufacturing. Post processors built for machine tools with the Sinumerik 840D will generate NC programs with content-rich controller instructions like 3+2 axis and full 5-axis transformation, high speed cutting (HSC) settings and drilling, tapping and threading cycles.

Benefits

Faster, easier implementation and setup for new parts

Reduced errors in the manufacturing process

Increased flexibility to react to change

Advantages

Employ advanced machine tool capabilities with the Sinumerik controller

Rich data communication between CAM system and NC controller

Features

3+2 axis frame concept for shifting, rotating, scaling and mirroring of coordinate system (CYCLE800)

5-axis programming with orientation transformation (TRAORI)

High speed cutting (HSC) settings (CYCLE832)

Drilling (CYCLE81 – CYCLE83), boring (CYCLE85 – CYCLE89), rigid tapping (CYCLE84) and tapping with chuck compensation (CYCLE840)

Thread cutting (CYCLE97)

HMI instructions for Sinumerik controller to display cycles on the control so that the parameters can be verified/modified during setup

Generate better NC programs

NC programmers and manufacturing planners can communicate more effectively with shop floor machine operators by referring to specific cycle names rather than G-codes. For example, NX CAM drilling operations are mapped to equivalent Sinumerik drilling cycles instead of G-codes. As a result, machine operators can more easily identify the drilling cycle NC program statements, with the support of the Sinumerik HMI, and if necessary modify the parameters and communicate the changes back to the programmer.



It is also possible to automatically specify shop floor machining process parameters by using information from NX CAM. The NX CAM post processor uses the machining operation (roughing, semi-finishing and finishing) to determine the appropriate machining parameter settings for the machine tool. High speed cutting parameters like tolerances, velocity control settings and jerk limitations are set based on the specified balance of machining requirements pertaining to accuracy, speed and finishing quality.

The Sinumerik post builder also incorporates the unique 3+2 axis and 5-axis machining capabilities of the Sinumerik 840D. User friendly commands like CYCLE800 for swiveling the machining coordinate system or TRAORI for 5-axis machining are Sinumerik NC statements that the post processor now generates for complex machining operations.

The value of enhanced post processing

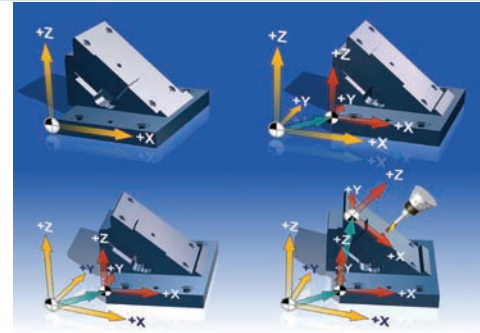
NC programmers have become accustomed to the user-friendly modeling and programming tools provided by NX CAD and NX CAM. Now, with a Sinumerik post processor, NC programmers and machine operators can also leverage the user-friendly programming functions of the Sinumerik controller to easily drive machine tools.

Unlike conventional post processors, which only support ISO6893/DIN66025 code, a Sinumerik post processor maintains the high-level information about machining processes, making the NC program easy to understand.

A Sinumerik post processor uses information about the machining process to generate a user-friendly NC program. The following functions are supported by the Sinumerik controller and post processor:

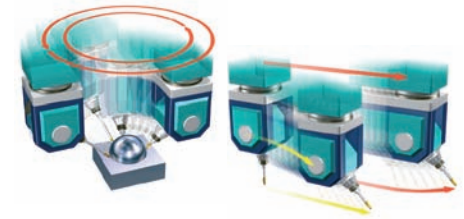
Frame concept – CYCLE800

The Sinumerik frame concept enables shifting, rotating, scaling and mirroring of coordinate systems. When used in conjunction with swiveling tools, this capability makes it easier to program complex work pieces.



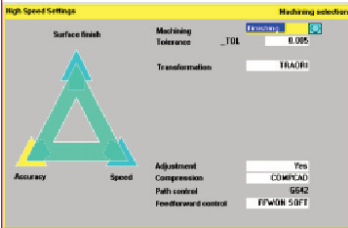
5-axis transformation – TRAORI

The Sinumerik 840D has unique programming functions that reduce the complexity of 5-axis programming. The orientation transformations provided by TRAORI enable simple 5-axis tool path programming independent of the machine tool kinematics.



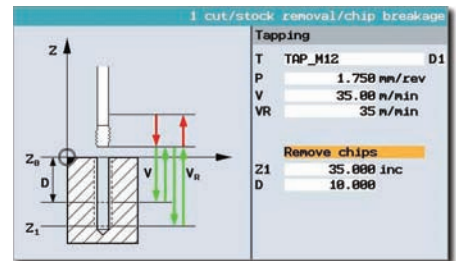
High speed cutting settings – CYCLE832

The Sinumerik 840D offers convenient high speed cutting settings for achieving the optimum balance between machining speed, accuracy and surface quality. Depending on the machining operation, roughing, semi-finishing or finishing, a Sinumerik post processor can automatically activate the appropriate parameter settings.



Drilling, boring, tapping and threading cycles

A Sinumerik post processor maps each drilling, boring, tapping and threading operations to its respective Sinumerik cycle. These cycles provide NC programmers and machine operators with a high-level NC programming language that is easily understood.



Moreover, this enables machine operators to easily select, display and modify cycle parameters at the machine tool using the Sinumerik program editor.

Technical solution example

During NC program generation the Sinumerik post processor uses its machining process knowledge to generate optimal Sinumerik NC program statements. For example, when the post recognizes planar milling operations on different planes with fixed rotary axes on a 5-axis machine, it will generate a CYCLE800 command. However, if the rotary axes are variable, then it will generate a TRAORI command to enable full 5-axis machining. Drilling and threading cycles are derived from machining features and the high speed cutting (HSC) settings can be generated depending on the type of machining operation specified.

Sinumerik post processors generate the optimal Sinumerik command for each machining operation.

