SIEMENS

GEOMETRIC SOLUTIONS

NX CAM for machinery

Optimized NC programming for machinery and heavy equipment

Benefits

- Effectively program any type of machinery part
- Program faster
- Reduce air cutting
- Automate programming of common tasks
- Work on solid, surface or wireframe models
- Maintain safety around fixtures and assemblies
- Handle changes quickly with associative tool paths
- Easily edit models as needed for optimal NC programming
- Simulate programs in the programming system

Summary

NX™ CAM software redefines manufacturing productivity with a full range of NC programming capabilities optimized for machinery parts that can be configured by purchasing selected modules or complete bundles of applications depending on your requirements. This fast and highly accurate machining solution offers automated, context-specific functions that simplify the generation of smart tool paths for prismatic parts. Advanced programming capabilities that are necessary for machinery parts are available with milling, drilling, turning, mill-turn and wire EDM applications.

Challenges in the production of machinery components

Machinery components can require multiple setups, complicated fixturing and are often machined in small to medium batch sizes. Effective cutting makes the difference between profit and loss. The machining process must eliminate any unneeded motion and account for:

- Chip control
- Coolant access
- Traverses and engages
- Tool life
- Multiple orientations on production fixtures
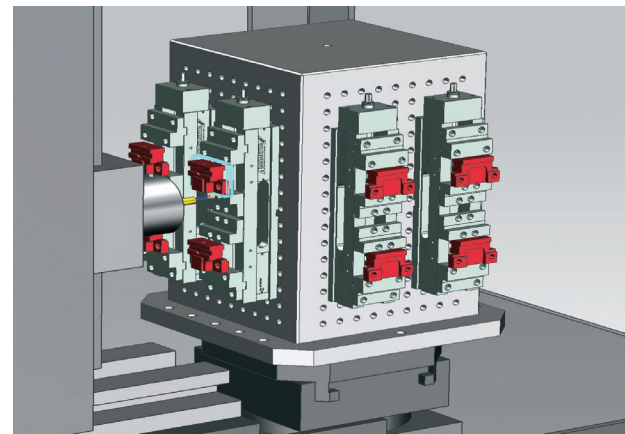
NX CAM provides advanced programming capabilities optimized for machinery parts.

www.siemens.com/nx

# NX CAM for machinery

Features

- 2½ D volume-based milling

- Multi-stage in-process workpiece

- Full range of milling, turning, holemaking and wire EDM machining

- Synchronous technology

- Feature-based machining (FBM)

- Integrated machine tool simulation

- Postprocessing and Post Builder

- Shop documentation

NX CAM addresses the need for careful control over tool motion with several levels of user control. At the same time, powerful automation tools provide quick programming for standard features, part families and even multiple parts arranged in fixtures.

Program prismatic parts faster

Volume-based 2.5D milling
You can program faster by working in the context of machinery parts with the new volume-based programming framework.
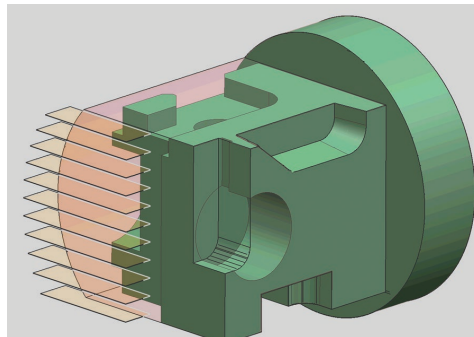

Instant visual preview drives the volume-based 2.5D milling process.

Quickly program prismatic parts (the faces to be machined are perpendicular or parallel to the tool axis) by simply specifying the volumes to be machined. The new volume-based 2.5D milling processor lets you select floors and walls, and then NX CAM calculates the machining volume so you can easily specify the optimum machining process. The instant visual preview of the machined area helps you quickly validate your programming decisions. The system continuously keeps track of uncut material and minimizes air-cutting, reducing machining time up to 10 percent.

Multi-stage machining and in-process workpieces
Efficient production of machinery components requires the ability to machine components in several stations, tracking the in-process workpiece (IPW) all along the way so that motion is efficient and air cuts are eliminated. Blank models follow the part from one station to the next,

representing the uncut material for the most efficient cuts.
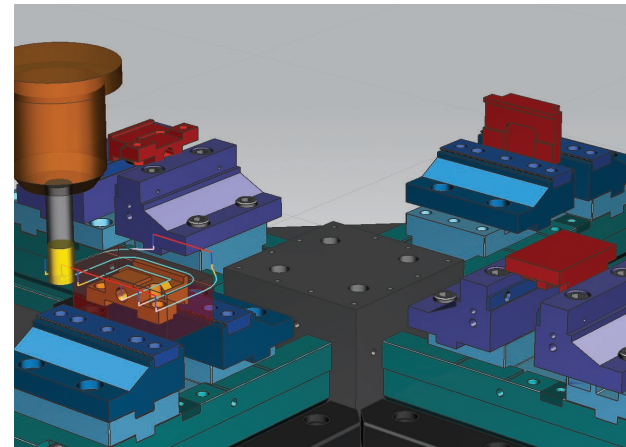
Multi-stage machining tracks in-process material from station to station.

Multiple part programming
NX CAM 9 delivers powerful, streamlined multi-part machining. It speeds the development of various tombstone and fixture configurations, distributing tool paths to as many components as required. Complete machining sequences and tool paths developed for one workpiece can be immediately distributed to the other workpieces in other positions and orientations.
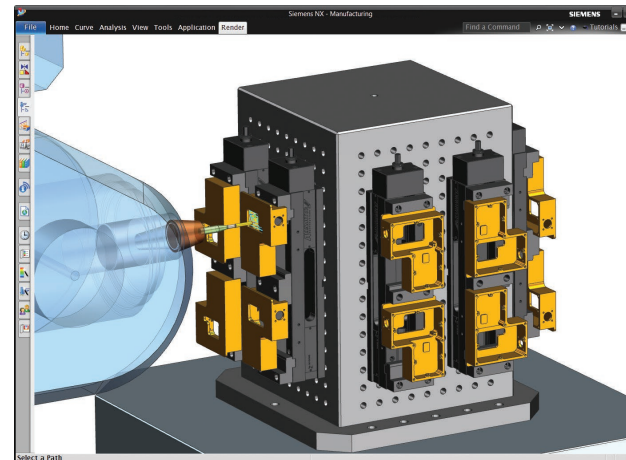
Machine multiple parts easily by distributing tool paths to your different workpieces

Full range of NC programming capabilities – all in one system

Face milling

Solids-based face milling automatically respects part and fixture boundaries while efficiently clearing faces. Multiple depths are provided as necessary in heavier stock.

Pocketing

NX CAM roughs any generalized 3D shape with intelligent multi-level volume removal patterns and automatically cuts levels corresponding to horizontal faces. Area clearing patterns include zig, zig-zag, part offsets, blank offsets and trochoidal for high speed machining (HSM).

Profiling

Solids-based profiling automatically understands material side, providing the appropriate engages and retracts with cutter compensation. Multiple passes and multiple depths are stock sensitive.

Boundary cutting

You can trace boundaries the traditional way with either edges or wireframe elements. You can specify offsets, compensation and other factors for boundary elements. These boundaries serve as the basis for either single-trace cuts or area clearing patterns.

Hole making

NX CAM supports hole cycles directly and even provides user-defined cycles for your more demanding hole-making needs.
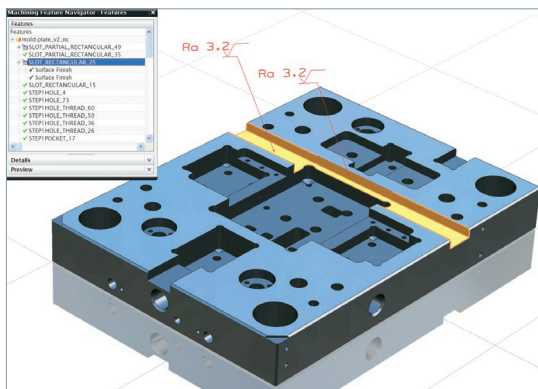
Feature-based automation speeds programming and supports best practices.

In addition, NX CAM provides several special milling process for milling of holes and bosses. With these cutting options you can mill holes into solid material or clean up holes and bosses in near net shape components. Threads can be milled internally or externally.

Turning

NX CAM provides a complete solution for turning, including multi-turret and multi-spindle turning centers. It performs OD and ID programming for turning centers and vertical turret lathes. Face, turn, bore and groove operations are supported for roughing and finishing cuts.

You can cut efficiently with turning operations that always track cut and uncut material. Teach mode provides step-by-step motion instructions with a few mouse clicks. You can configure left or right handed tools and specify the holder and insert as part of a tool assembly.
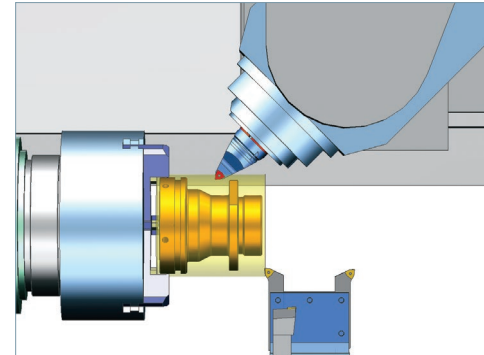
Mill-turning

NX CAM makes it easy to program mill-turns, with a full spectrum of both milling and turning capability. Multiple spindles and multiple turrets are supported along with the availability of live tooling on the turrets. Even multi-axis milling techniques are supported, since any milling operation can be applied to a mill-turn application if the machine will support it.

Multi-channel synchronization helps shrink cycle times for multi-channel machines (multiple milling or drilling heads on the same machine) by optimizing synch codes. The visual system displays timelines and simulates multi-channel motion to ensure that minimum cycles are achieved without interference.

Wire EDM

The Wire EDM capability in NX CAM cuts either 2 axis or 4 axis geometry configurations. You can rough out volumes with the no-core area option or cut profiles. Flexible geometry input means that no matter what geometry you start with, you are ready to create Wire EDM tool paths. You can select


Turned and mill-turned components are examples of machinery components.

curves, faces or solids. And NX Wire EDM offers the productivity benefits of feature-based automation. Wire EDM features are recognized and intelligent processes are applied to the recognized features.

Graphical-driven programming
NX puts you in the context of a specific work task by displaying previews of results and allowing you to directly interact with on-screen elements. With interactive grab handles you can quickly establish preferred positions, tool axes and approach directions.
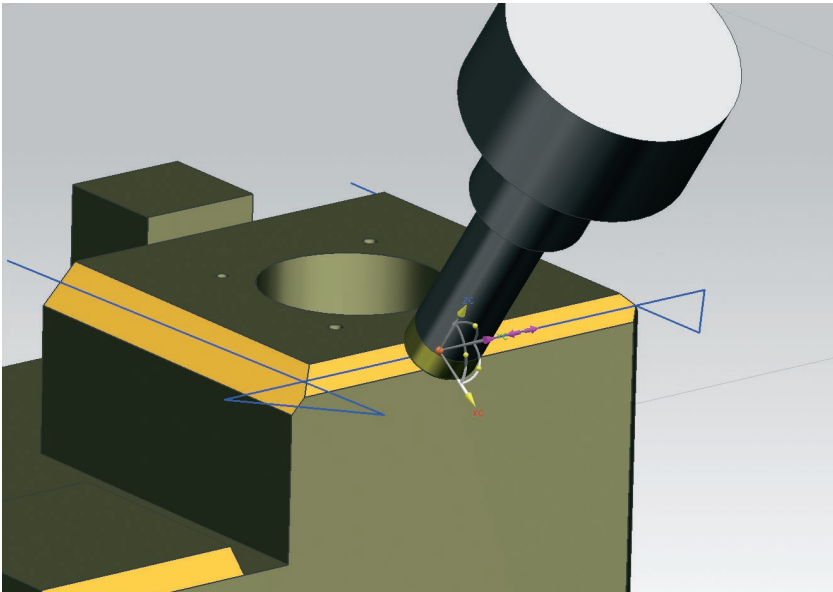
Quickly establish positions and tool axes with interactive grab handles.

Tools preview right in the setup, so that you can drag them around and check for fit and clearance. Tool path patterns preview interactively for control over cut and stepover directions. Generic motion operations let you drag and drop the motion itself, snapping it to part geometry as needed.

NC programming automation

Feature-based machining
You can automate programming for standard features, part families, and even multiple parts arranged in fixtures. NX is able to recognize features by topology, by product manufacturing information (PMI), and even by color. Features can then be machined by applying rule-based processes to them. Rules can even reflect the tolerances or surface finish stored on the part as PMI or embedded GD&T.

When features and their associated rules are customized to your shop's environment and best practices, there are remarkable benefits available for productivity, repeatability and reliability.

With the Teach Features and Teach Operations functions, you can easily teach NX CAM your company's standard machining processes. Then feature recognition can find your features on parts and automatically apply your specific machining operations – for fast error-free programming.

Integrated machine tool simulation
With NX CAM, you can perform machine simulation right from the programming environment so that information is at hand when you need it, avoiding the requirement for a separate machining simulation package for the NC programmer.

The NC Simulation capability provides a 3D model-based visualization of the planned machining operations. NX CAM facilitates multiple levels of simulation, ranging from basic tool path verification to full machining simulation that includes a complete machine tool model. Collision checking and multi-channel synchronization are simulated at the g-code level for the most accurate results.

CAD for the NC programmer

With NX CAM you have access to the industry leading CAD modeling and assembly functions inside NX – all within the same system. This powerful system lets you design fixtures, position component parts for fixtures and clamps, and create drawings for setups without ever leaving to work in a different system.
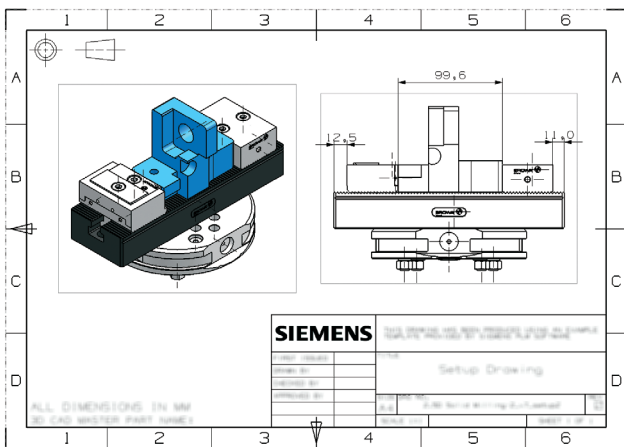
Design fixtures, create drawings, prepare models, and repair defects using available CAD, including synchronous technology.

The CAD capabilities in NX allow you to import and prepare models for NC programming. You can even repair poor quality geometry and adjust models to tolerance conditions by using Synchronous Technology. As a leading CAD solution, NX has virtually every tool to support your NC programming tasks.

And NC programs are able to update to design changes with ease through a fully integrated CAD/CAM system that keeps the NC program connected to the design model.

Work in the context of a complete solution

Resources

A big part of programming in the machinery environment is the effective use of the cutting tools, holders, fixtures and other setup components that are important to your production process. NX has an extensive tool library that you can update and optimize. Cutting parameters are likewise provided in an extensible, customizable database that puts the best practices of your shop into the center of your programming process.

Fixtures and related setup components can be developed in as much detail as necessary for the fast creation of safe and accurate NC programs. By working in an assembly environment NX CAM maintains specified clearances from clamps and fixtures. Plus NX CAM lets you share the setup assemblies as drawings for shop floor use.

Shop documentation

The manufacturing of machinery components is process focused. Many revisions can occur over the long life spans of machinery products, and special fixturing is often developed to streamline operations or minimize tool changes.

NX makes sure the setup configurations can be shared properly with the shop floor, either via conventional drawings or via 3D viewer technology. Tool lists and operation lists can be easily produced and shared via web pages or paper documents. NX CAM can automatically produce shop documentation that includes both tooling info and operation sheets based on an easily editable excel spreadsheet template.

844-GEO-SUPT
support@geoplm.com
geoplm.com