SIEMENS
Ingenuity for life

NX CAD/CAM 2.5-Axis Milling Foundation

Multiple part programming
NX CAM 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.

Hole making
Extensive hole making functions are provided for spot drill, drill, ream bore and tap. You can use deep drill, peck drill and special boring cycles. User-defined cycles provide any needed hole making capability. Efficient traverses minimize lifts while accounting for all part and fixture geometry. Special multi-axis filtering automatically switches MCS for non-parallel hole selections.

Feature-based automation
You can take advantage of automatic feature processing that comes standard with NX. Feature recognition, process application and tool selection functions automate NC programming of many features, including pockets, slots and holes. Custom process development is separately available.
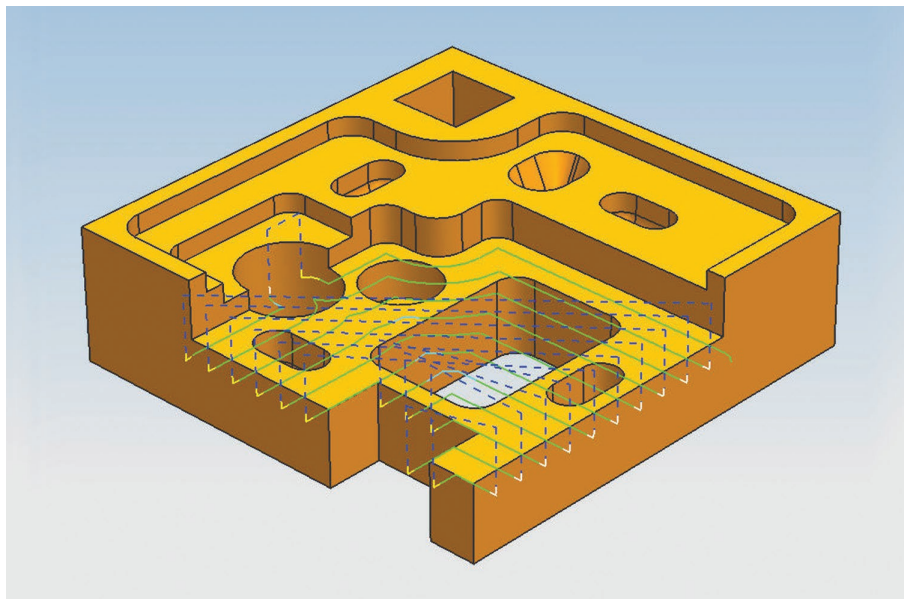
Face milling
Solids-based face milling automatically respects part and fixture boundaries while efficiently clearing faces.

NX CAM benefits
- Volume-based milling automates programming of prismatic parts

- Automated hole making capability speeds common processes

- Solids-based cutting machines complex shapes intelligently

- HSM support maximizes machine tool investments

- Easily program multiple parts and multiple stage fixtures

- Intelligent rest milling options facilitate associative multi-stage roughing techniques

- Master model capability ensures that NC development stays associative to the source model

- Model editing capabilities make it easy to adjust model for optimal NC programming

- Integrated postprocessor and Post Builder provide production-ready output

- All supporting functions are provided

Summary
NX™ CAD/CAM Milling 2.5-Axis Foundation software provides an entry-level package focused on general prismatic milling with integrated CAD. All of the supporting functions for NC programming are provided, from trans-lators to tool path visualization to postprocessing.

2.5-Axis Milling and Drilling
2.5-Axis Milling and Drilling operations cover solids and boundary-based z-level machining through a mix of manual and automated programming methods.

Flexible machining coordinate system (MCS)
With NX CAM, you can make holes and planar cuts anywhere on the part, using any valid tool axis. These 2.5-axis cuts can be performed by 3+2 positional machines in any orientation.

Multi-stage machining and in-process workpieces
Efficient production of prismatic com-ponents requires the ability to machine parts 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.

NX CAD/CAM 2.5-Axis Milling Foundation

NX CAM features
- Hole making
- Face milling
- Boundary cutting
- Generalized roughing
- Z-level finishing
- Synchronous technology
- Wizard builder
- Postprocessing and Post Builder
- Shop documentation
- Raster and offset patterns
- Interpolated patterns

Volume based 2.5 D milling

Cutting volumes are automatically identified based on floor and wall selections. These floors and walls in combination with the in-process work piece result in volumetric cut regions.

Boundary cutting

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

Generic motion control

You can build step-by-step tool motion with interactive drag handles and chain together cut traces with the most efficient transitions.

Probing cycles

NX CAM performs on-machine probing with included Renishaw probing cycles for single-tip probes. You can measure faces, holes and bosses.

Generalized roughing

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 patterns.

High-speed machining (HSM)

NX CAM provides specialized HSM patterns for either boundary- or solids-based roughing. You can keep your tool path smooth with corner treatments, step-overs, engages and retracts to facilitate the highest possible feed rates. Using the included cutting parameter library, you can optimize spindle speeds, feed rates and engagement depths to maximize safe material removal. Trochoidal loops protect tools from excessive engagement. When complete corners are required, feed rates automatically reduce to account for heavier local tool loads.

Z-level finishing

You can create z-level, or waterline, finishing passes on complex 3D geometry. Suitable for relatively steep areas, these cuts provide zig and zig-zag options with smooth engages, retracts and step-overs.

Rest machining

For re-roughing, NX CAM cuts only the areas uncut by previous roughing operations. You use smaller tools only as needed for corner cleanout and use longer tools only as needed for deep reaches.

Standard supporting functions

Translators

You can read data from practically any CAD system with a set of translators for Parasolid® software formats and the IGES, DXF, STEP and JT™ formats. In addition, you can write data in all of these formats, plus stl.

Operation navigator

From the CAM application's information center, you can create, view, modify, verify, inspect and organize your NC programs. You can manage tools, methods, geometry and output from an information-rich program display.

Tool path visualization

You can stay confident with tool path visualization and material removal verification tools. Integrated collision checking identifies any possible problems.

Milling Foundation benefits
- Full design capability with integrated CAD/CAM seat

Milling Foundation features
- 2D drafting
- Solid modeling
- Basic surface modeling
- Sheet metal modeling

Tool path editing
You can make fast changes to tool path results with graphical editing tools and protect your edits by locking tool paths from future recalculations.

NC wizards
With the NC Wizard Builder, you can create semi-automated processes customized to your environment. These highly familiar wizard presentations enable you to streamline common tasks with minimum user input.

Postprocessing
The Milling Foundation outputs production-ready g-code with integrated postprocessing. You can edit and fine tune existing postprocessors with our included interactive Post Builder application or even build a postprocessor from scratch using Post Builder templates. In addition, you can use included sample posts or download a post from our support library.

Shop documentation
You can produce html setup documents with the included shop doc capability.

Feed rate optimization
NX CAM helps you to easily maximize material removal rates and minimize tool wear. The ability to adjust feed rates to achieve a specified chip volume eliminates overloading as well as under-utilization of the cutting tool.

Assembly modeling
NX CAM includes important fixture components in the programming session while facilitating intelligent collision avoidance. Our master model concept keeps assemblies and tool paths associative to your source model. NX assembly design supports "top-down" and "bottom-up" approaches while facilitating rapid navigation of the assembly hierarchy through direct access to the design model of any component or subassembly.
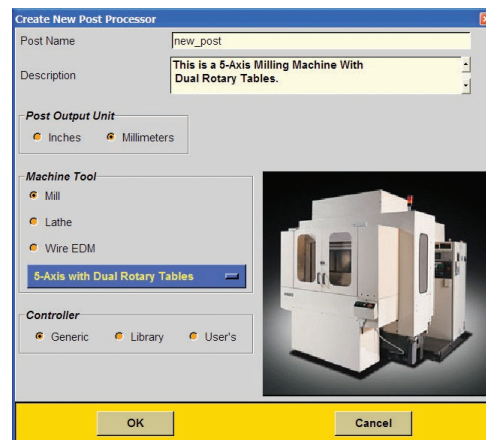
Standard model editing functions
NX CAM provides geometry tools to support common model editing functions that make NC programming more efficient. The standard range of functions provided are effective for making changes to existing models that are sometimes needed in the NC programming context, such as altering a basic stock material shape.

Using the standard model editing functions, you can perform important model cleanup tasks with key editing capabilities for:

Associative geometry
You can create associate copies of master geometry, enabling model editing without affecting the original design model.

Basic solid shapes
Solid blocks and cylinders can be created for use as blank models or fixture components.

Surface extensions and patches
You can patch over features that don't need to be machined. Tool paths will move smoothly across the patch, rather than trying to dig into the feature. You can extend faces to help guide tool tips around 3D edges if necessary.
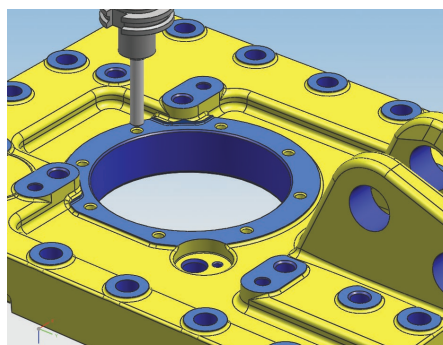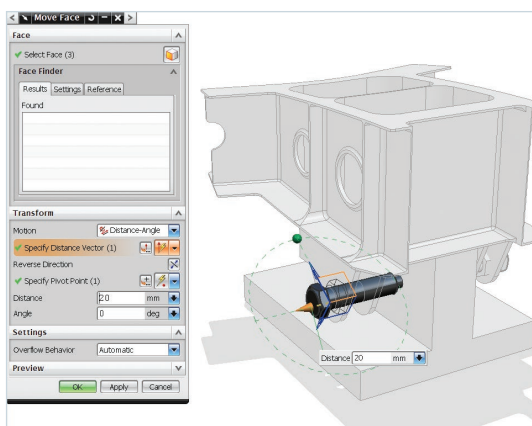
Surface and edge extraction

Surfaces and edges can be extracted, divided, projected and used in various ways to more precisely support the needs of NC operations. For example, a divided surface provides an additional edge that can be used to drive or define tool paths. Likewise, trim, sew and join functions enable surface and edges to be modified to support NC operations.

Synchronous modeling

Synchronous modeling, a powerful method for directly editing solid faces, is also available in the NC environment. Developing as-cast models or other derivatives from the existing parts is easy since the programmer simply needs to push and pull on the existing faces.

Extended CAD content (included with the CAD/CAM package)

The standard CAM version of the CAD/CAM package provides geometry tools to support common model editing functions. A set of more extensive CAD functions (described below) is available in the CAD/CAM version of this package that enables you to create and edit complex 3D part models and produce drawings. The combination of CAD and CAM in one system provides one of the most fully integrated solid modeling systems available today.

Drafting

You can create engineering drawings that remain associative to your part models, including automatic view creation from solids, hidden line processing and automated parts lists. You can use templates to automatically create drawings for borders and view layouts.

Solid modeling

Using the CAD/CAM Milling Foundation, you can develop 3D models with a full range of synchronous modeling and parametric modeling tools, including wireframe, primitives, swept and revolved bodies; Boolean operations and associative editing are supported. You can provide details such as blends, tapers and thin-wall features. NX modeling is powered by synchronous technology, Siemens PLM Software's unique approach to 3D design that combines the power of parametric feature-based modeling with the freedom of history-free modeling.

User-defined features

The CAD/CAM Milling Foundation lets you capture and store families of features for easy retrieval and editing, starting with existing associative solid models. You can establish relationships between parameters, define feature variables, set default values and decide the general form the feature will take when started.

Basic surfacing

You can create complex surfaces and solid models using loft and sweep techniques.

Sheet metal

With the CAD/CAM Milling Foundation, you can create sheet metal component models using feature-based design tools for tabs, flanges and other typical features. You can use forming tables and bend sequence tables to account for bend deformation and generate accurate flat pattern data for manufacturing.

Associativity

The CAD/CAM Milling Foundation allows you to quickly respond to design changes through a fully integrated CAD/CAM system that keeps machining operations associative to the part model.

Available add-ons

You can configure the perfect CAM seat by adding the tool path creation options that your programs require. You can choose from:

- Turning
- Wire EDM
- 5-Axis Milling
- Turbomachinery Milling
- Machine Simulation
- Feature-Based Machining Author

Available Teamcenter integration

You can integrate NX CAM with Teamcenter® software by including the NX CAM Teamcenter Client add-on. This integration provides a managed environment for your NX data. Additional Teamcenter functionality is available separately.