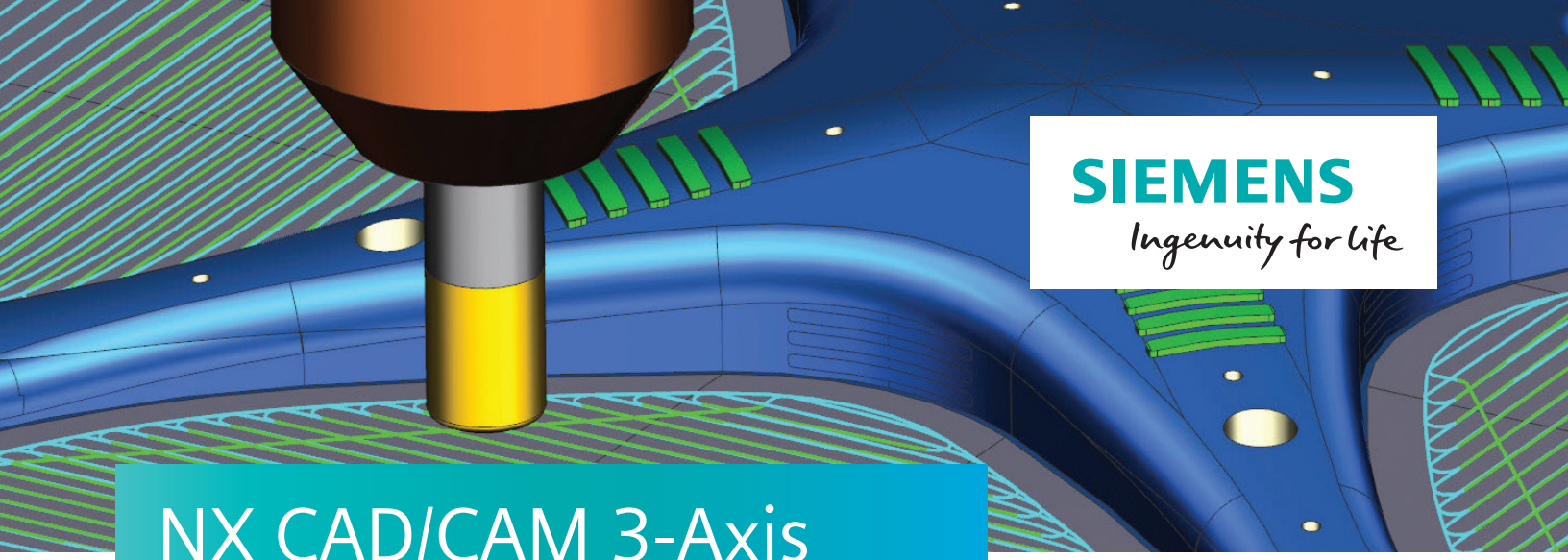


SIEMENS
Ingenuity for life

NX CAD/CAM 3-Axis Milling Foundation

Benefits

- High-speed machining maximizes CNC machine investments
- Adaptive milling enables high material removal rates
- Smooth cutting pattern for high quality surfaces
- Feature-based machining automatically creates complete machining processes
- Master model capability ensures that NC programs stay associative to the source model
- Integrated postprocessor provides production-ready output

Summary

NX™ CAD/CAM 3-Axis Milling Foundation software provides an entry-level package focused on general 3-axis milling with integrated computer-aided design (CAD). All of the supporting functions for numerical control (NC) programming are provided, from translators to toolpath visualization to postprocessing.

3-axis milling

3-axis milling provides extensive fixed-axis cutting capabilities for complex surfaces such as those found in mold and die applications. This functionality enables you to achieve the smoothest toolpath results in steep areas by specifying a projection vector different from the tool axis.

Independent projection vector

NX CAM achieves the smoothest toolpath results in steep areas by enabling you to specify a projection vector different from the tool axis.

Smooth raster cutting

To maintain high feed rates and fine finishes, you can output the smoothest raster, or zig-zag, cuts with special stepover moves and optimized point distribution to maintain high feed rates and fine finishes.

Smooth offset cutting

NX facilitates smooth offset pattern cuts with special stepover moves and corner treatments to maintain high feed rates and fine finishes. You can cut inside-out or outside-in.

Steep/nonsteep area recognition

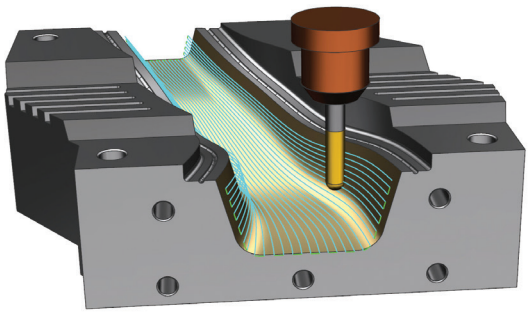
You can limit raster cuts to nonsteep areas for better cut depth control, as well as provide consistent semifinish cuts with combination z-level and raster fill-in patterns.

Interpolated passes

NX provides toolpaths that flow organically with your part faces. With these smooth patterns, you can interpolate guide curves for the smoothest finishes. You can even cut true 3D spiral patterns with no stepover marks.

Automatic valley rest milling

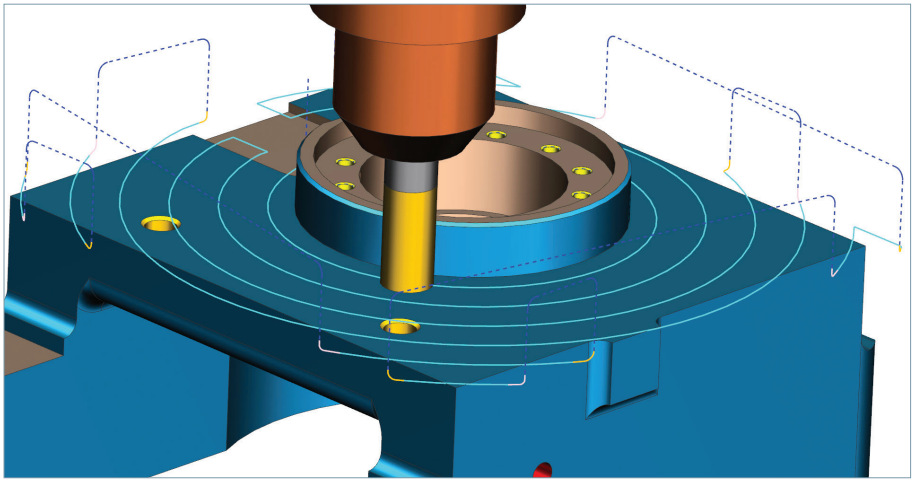
NX discovers uncut valley material remaining from larger diameter tools, including bullnose tools. You can cut the remaining tools along or across valleys.



NX CAD/CAM 3-Axis Milling Foundation

Features

- Adaptive roughing
- Raster and offset patterns
- Interpolated patterns
- Automatic valley cutting
- Flexible cut region management for pattern optimization
- Post Configurator



NURBS output
You can output non-uniform rational basis spline (NURBS) records for the smoothest machine motion and take advantage of high-level controller capabilities.

Facet machining
With NX CAM, you can use scanned data or other faceted representations, such as the stereolithography (STL) and the JT™ data formats for NC programming.

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. It enables 2.5-axis operations to be performed on 3+2 positional machines in any orientation.

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 enable machining of any hole. Special multi-axis filtering automatically switches MCS for nonparallel 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 machining of many features, including holes, pockets and slots. Custom process development is separately available.

Face milling
Solids-based face milling automatically categorizes the part and fixture, enabling easy programming and safe machining.

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 multilevel volume removal patterns and automatically cuts levels corresponding to horizontal faces. Area-clearing patterns include zig, zig-zag, part or blank offsets.

High-speed machining (HSM)
NX provides an adaptive high-speed cutting strategy that helps you achieve a new level of machining efficiency. The intelligent roughing method enables deep cuts and constant tool load that increases material removal rate. Adaptive milling is ideal for cutting hard material that enables a significantly shorter machining cycle while extending tool life.

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 stepovers.

Rest machining
For re-roughing, NX CAM cuts only the areas untouched 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 computer-aided design (CAD) system with a set of translators for the Parasolid® software, the Initial Graphics Exchange Drawing Exchange Specification (IGES), Drawing Exchange Format (DXF), Standard for the Exchange of Product Data (STEP) formats and the JT data formats, plus STL.

Operation navigator
From the computer-aided manufacturing (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.

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

Toolpath editing
You can make fast changes to toolpath results with graphical editing tools and protect your edits by locking toolpaths from future re-calculations.

CAM templates
You can utilize the powerful template mechanism to create semi-automated processes customized to your environment. Highly familiar operation dialogs help you to streamline common tasks with minimum user input.

Postprocessing
NX outputs production-ready g-code via its integrated postprocessing. You can edit and fine-tune existing postprocessors with the included interactive Post Configurator application, or even build a postprocessor from scratch using Post Configurator templates. You can also 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.

Standard model editing functions
NX CAM provides design 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 often needed in the NC programming context, such as altering a basic stock material shape.

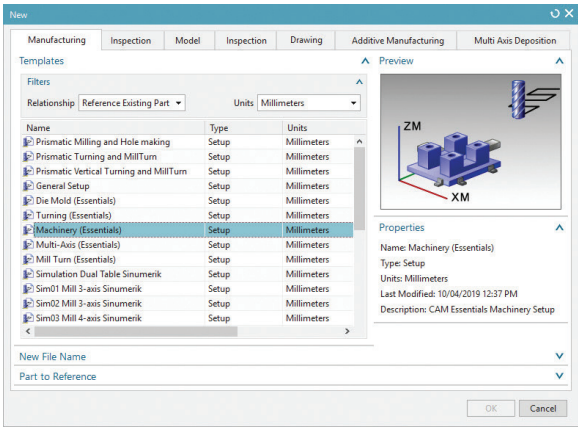
Using the standard model editing functions, you can perform important model cleanup tasks with key editing capabilities for:

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

3D wireframe construction for boundaries
Wireframe elements can be constructed from scratch or derived from part edges. Boundaries can be used directly for certain 2D operation types or used indirectly as containment boundaries to control toolpath coverage.

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. Toolpaths 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 toolpaths. 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.

Assembly modeling
NX CAM includes important fixture components in the programming session while facilitating intelligent collision avoidance. Our master model concept keeps assemblies and toolpaths 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.

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 NX 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 Digital Industries 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 NX 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, including multirail methods. You can also fine-tune surfaces with surface extension and surface normal control.

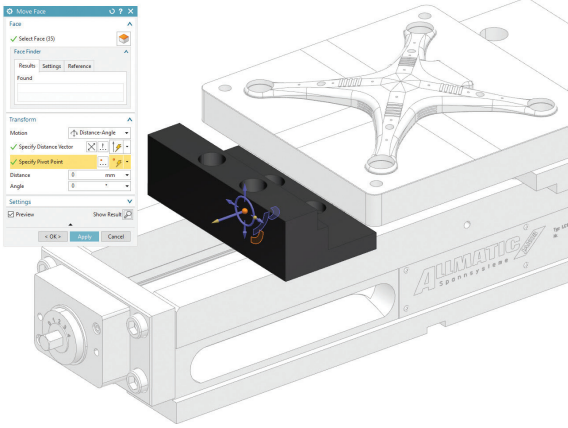
Sheet metal
With NX CAD/CAM Milling Foundation, you can create sheet metal component models using feature-based design tools for tabs, flanges and other typical features. You can use forming tables and bend sequence tables to account for bend deformation and generate accurate flat pattern data for manufacturing.

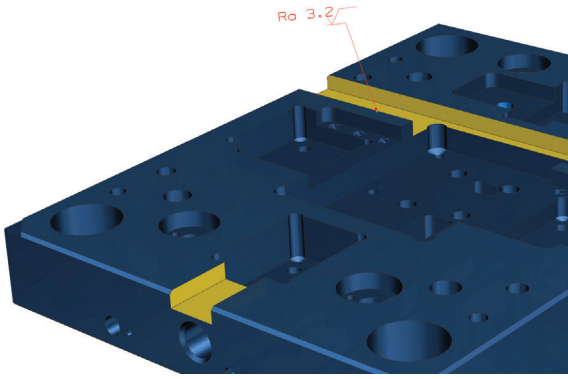
Associativity
This CAD/CAM package allows you to quickly respond to design changes through a fully integrated CAD/CAM system that keeps machining operations associative to the part model.

Available add-ons
You can configure the perfect CAM seat by adding the toolpath creation options that your programs require. You can choose from:

- Turning
- Wire electrical discharge machining (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.





Siemens Digital Industries Software
siemens.com/software

Americas +1 314 264 8499
Europe +44 (0) 1276 413200
Asia-Pacific +852 2230 3333

© 2020 Siemens. A list of relevant Siemens trademarks can be found here. Other trademarks belong to their respective owners.
23331-C12 5/20 C