

NX CAM Foundation

Benefits

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

Features

- Synchronous technology
- Wizard builder
- Postprocessing and Post Builder
- Shop documentation

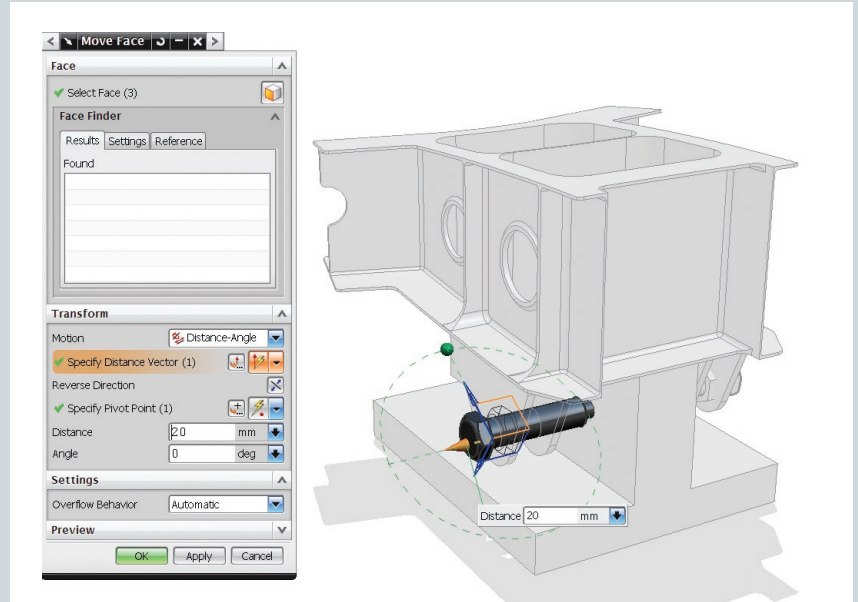
Summary

The NX™ CAM Foundation serves as the cornerstone of every NX CAM product configuration. All of the supporting functions for NC programming are provided, from translators and tool path visualization to postprocessing.

Standard supporting functions

Translators NX CAM Foundation reads data from practically any CAD system with a set of translators for the Parasolid® software, IGES, DXF, STEP and JT™ formats. You can write data in all of these formats, plus STL.

Assembly modeling NX CAM includes important fixture components in the programming session while providing intelligent collision avoidance. Our master model concept keeps the assemblies and tool paths associative to your source model. NX assembly design supports “top-down” and “bottom-up” approaches; it also provides rapid navigation of the assembly hierarchy with direct access to the design model of any component or subassembly.



NX

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NX CAM Foundation

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 NX CAM Foundation instills confidence by providing tool path visualization and material removal verification tools. Integrated collision checking identifies any possible problem.

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 re-calculations.

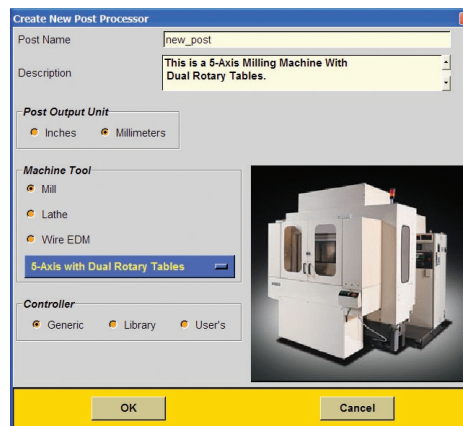
NC wizards You can use the NC Wizard Builder to create semi-automated processes customized to your environment. Highly familiar wizard presentations help you to streamline common tasks with the minimum user input.

Postprocessing NX Foundation outputs production-ready g-code via its integrated postprocessing. You can edit and fine tune existing postprocessors with the included interactive Post Builder application, or

even build a postprocessor from scratch using Post Builder 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 Foundation provides integrated feed rate tables that help 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 geometry tools to support common model editing functions that make NC programming more efficient. The standard range of functions provided are effective for making the kinds of changes to existing models that are sometimes needed in the NC programming context, such as altering a basic stock material shape. A set of more extensive CAD functions are available in the CAD/CAM version of this package to enable you to create/edit complex 3D part models and produce drawings.

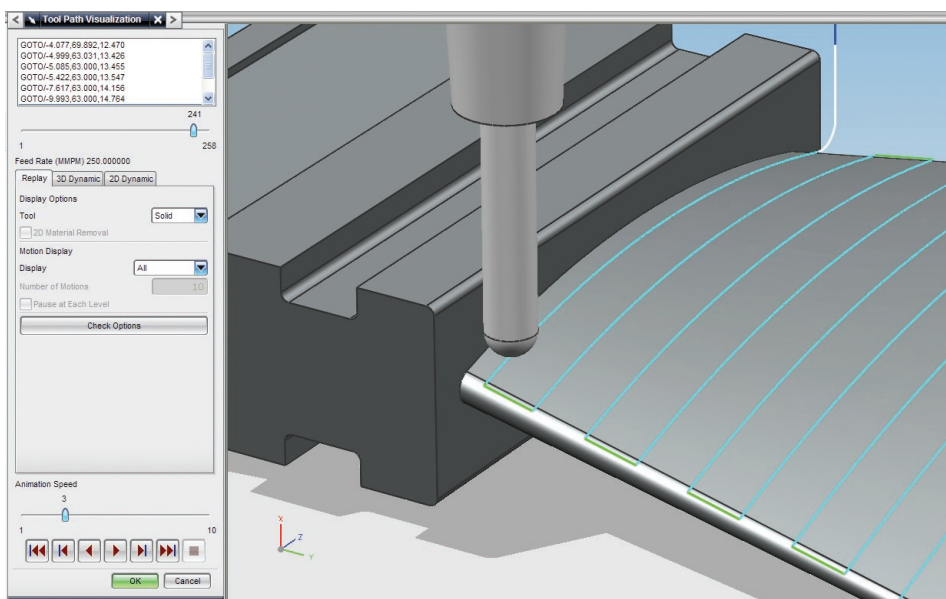
Using the standard model editing functions, you can perform important model cleanup tasks with key editing capabilities for:

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 tool path 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. Tool paths will move smoothly across the patch, rather than try 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.

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
- 2.5 Axis Milling
- 3 Axis Milling
- 5 Axis Milling
- Turbomachinery Milling
- Machine Simulation
- Feature-Based Machining Author

Available Teamcenter integration

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

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