NX CAM Foundation

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NX CAM product configuration from Siemens Digital Industries Software

Benefits
- Master model capability ensures that NC programming stays associative to the source model
- Synchronous modeling makes it easy to adjust model for optimal NC programming
- Integrated postprocessor provides production-ready output
- Supporting functions are provided

Features
- Synchronous technology
- Translators
- Post Configurator
- Shop documentation

Summary
NX™ CAM Foundation software serves as the cornerstone of every NX CAM product configuration. All of the supporting functions for numerical control (NC) programming are provided, from translators and toolpath visualization to postprocessing.

Capabilities
Standard supporting functions include:

Translators: NX CAM Foundation reads data from practically any computer-aided design (CAD) system with a set of translators for the Parasolid® software, Initial Graphics Exchange Specification (IGES), Drawing Exchange Format (DXF), Standard for the Exchange of Product Data (STEP) formats and the JT™ data format. You can write data in all of these 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: NX CAM Foundation instills confidence by providing toolpath visualization and material removal verification tools. Integrated collision checking identifies any possible problem.

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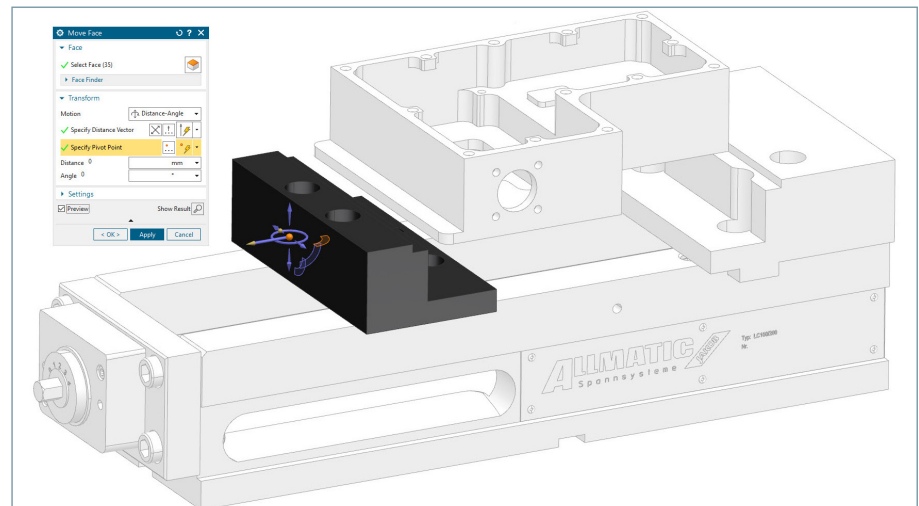
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CAM templates: You can utilize the powerful template mechanism to create semi-automated processes customized to your environment. Highly familiar operation dialogs help you streamline common tasks with minimum user input.

Postprocessing: NX CAM Foundation 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 set up 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 design 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 often 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 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 try to dig into the feature. You can extend faces to help guide tooltips 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, the trim, sew and join functions enable surfaces 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 and provides intelligent collision avoidance. Our master model concept keeps the assemblies and toolpaths 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.
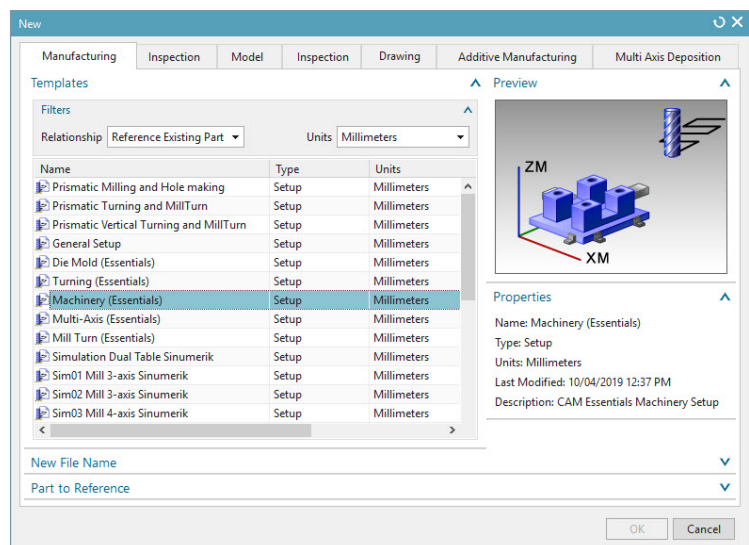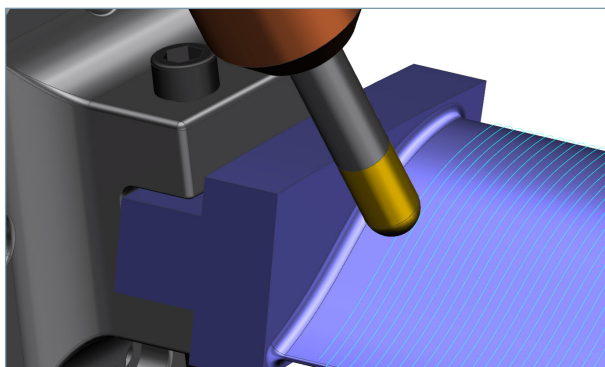
Available add-ons

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

- Turning

- Simulation

- Robotics

- Wire electrical discharge machining (EDM)

- 2.5-axis

- 3-axis

- 5-axis

- 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.