



DIGITAL INDUSTRIES SOFTWARE

NX X Manufacturing Solutions

Benefits

- Boosts part manufacturing efficiency
- Master model capability ensures that NC programming stays associative to the source model
- Improves collaboration
- Integrated postprocessor provides production-ready output

Features

- End-to-end part manufacturing solutions with a rich set of supporting tools
- Modeling for Manufacturing, translators and comprehensive NC programming
- Gain the flexibility, security and scalability of cloud technologies

Summary

NX™ X Manufacturing software products are prepackaged Software as a Service (SaaS) solutions powered by cloud technologies delivering the high-performance computer-aided manufacturing (CAM) capabilities of NX software with all the associated flexibility, security and scalability benefits of SaaS. Based on the same NX architecture and leveraging the same data model, Siemens provides one NX—with one common user experience—with different deployment options available to meet your business needs. They offer competitively priced solutions tailored to specific product development roles, practices and processes in three performance tiers. Value-added upgrades are available to move customers from one tier to the next. Each package delivers complete capabilities for NC programming work.

The [NX X Manufacturing](#) solutions are delivered via cloud SaaS named-user licensing. Token licensing exclusively extends and enhances these solutions. Traditional on-premises software is also available through NX Manufacturing solutions.

NX X Manufacturing [Standard](#), [Advanced](#) and [Premium](#) solutions include base-level data management and everything you need to integrate with Teamcenter® software to provide optional extended engineering process management tools through scalable collaboration and advanced management. NX and Teamcenter are part of the Siemens Xcelerator business platform of software, hardware and services.

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Features continued

- Comprehensive, preconfigured SaaS solutions tailored for the full range of part manufacturing applications
- Power and flexibility that support virtually any design methodology, whether top-down or bottom-up
- Foundation for product lifecycle management
- Extendable with a selection of add-on application modules

All the NX X Manufacturing solutions share a common denominator of integrated cad tools, post configurator, post hub, and toolpath animation, enabling NC programmers to effectively build confidence in their program. Each level of the solutions builds upon the other, providing increasingly sophisticated and advanced machining capabilities.

NX X Manufacturing Standard

The NX X Manufacturing Standard is an entry-level software solution to automate prismatic part programming using:

- 2.5 axis milling
- Turning
- Holmaking

- Integrated CAD tools – modeling, editing and assembly capabilities, which speed up CNC programming.
- Post Hub – modern, browser-based online library of more than 1,200 postprocessors and machine kits

NX X Manufacturing Advanced

The NX X Manufacturing Advanced software solution includes all the capabilities of NX X Manufacturing Standard and suited for programming freeform parts, providing enhanced product design capabilities, including:

- 3-axis milling
- Advanced freeform modeling

NX X Manufacturing Premium

The NX X Manufacturing Premium software solution includes all the capabilities of NX X Manufacturing Standard and Advanced but also includes a high-performance machine tool simulation, synchronization manager, rotary machining capabilities, and 5-axis milling.

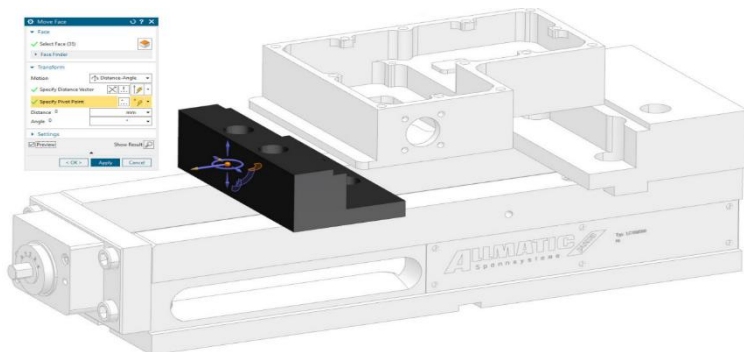
Enhance NC programming through production

Leveraging NX X Manufacturing solutions enables you to deliver comprehensive and scalable digital part manufacturing development that addresses all stages of the NC program process, from

programming to simulation through production.

Using the NX X Manufacturing packages transforms the part manufacturing process and supports dynamic change within organizations by:

- Increasing innovation throughout the part manufacturing process
- Eliminating waste by better utilizing time, material and intellectual resources
- Improving quality from the beginning



	NX X Manufacturing Standard	NX X Manufacturing Advanced	NX X Manufacturing Premium
Data Management			
Teamcenter integration for NX X	•	•	•
Base level data management	2503	2503	2503
Machining Capabilities			
2.5 axis milling	•	•	•
Feature-based machining	•	•	•
Turning	•	•	•
3-axis milling		•	•
3+2 positioning machining	•	•	•
Rotary machining			•
5-axis milling			•
Toolpath simulation	•	•	•
Postprocessing	•	•	•
Shop documentation	•	•	•
Machine code simulation			•
Synchronization manager			•
Modeling for Manufacturing			
Synchronous Modeling	•	•	•
Feature-based solid modeling	•	•	•
Basic freeform modeling	•	•	•
Advanced freeform modeling		•	•
Core convergent modeling	•	•	•
Assembly tools	•	•	•
CAD translators	•	•	•
Productivity			
Teamcenter Share	•	•	•
Post Hub	•	•	•
Data exchange			
DXF/DWG, IGES, STEP 203/214, JT, Solid Edge™ software open, SolidWorks open	•	•	•

With NX X Manufacturing solutions, you can gain key advantages such as:

- A unified solution – seamless application integration rapidly propagates changes to design and associated process information
- Teamcenter integration – all tools necessary to integrate with Teamcenter
- Integrated simulation and validation – comprehensive simulation and validation tools check NC program performance and manufacturability throughout every step of product development

NX X Manufacturing capabilities

Teamcenter Integration for NX X

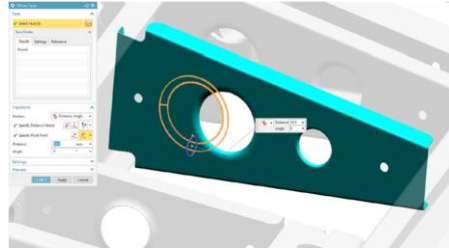
NX X offers full integration with Teamcenter software. The powerful data management system includes tools for vaulting, check-in/check-out, revision management, attribute synchronization and searching. It also includes data conversion capabilities for generating visualization files.

Modeling for Manufacturing



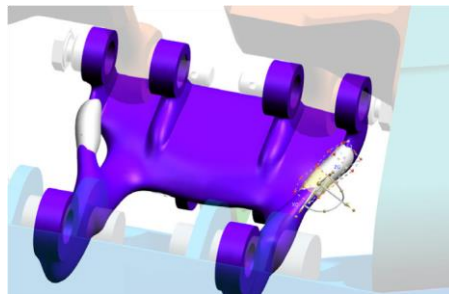
Feature-based solid modeling The core modeling capability in NX combines wireframe, surface, solid, parametric and direct modeling in a single environment that enables designers to choose the most appropriate tool for the task at hand.

Using NX delivers full feature based parametric solid modeling and advanced features, including blends, thin-wall, draft, mirrored features, open profile features and patterns. Additionally, it can support creating 2D/3D wireframe models, swept and revolved solids, Boolean operations and basic parametric editing. This includes variational sketching tools for quick and efficient conceptual design as well as tools for general modeling and editing tasks. Models retain their original parametric values, and you can edit them using intuitive dimension-driven techniques.



Synchronous technology

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.



Core Convergent Modeling

Convergent Modeling™ technology is a game-changing technology for working with facet geometry. It is now possible to design with facet geometry using the same parametric, feature-based methods that are used with traditional CAD geometry. You can use familiar feature-based modeling tools (Boolean, trim, offset, shell, scale, taper, blend, chamfer) when working with facet geometry. Designers can work with facet geometry in assembly modeling, drafting, PMI and other applications.

Assemblies

NX X Manufacturing 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.



Basic freeform modeling

Basic freeform modeling is a core suite of tools for creating freeform solid or surface shapes from wireframe geometry. This includes history-based parametric methods. Creation methods include offsets, extensions, rulings, lofts, sweeps, networks and boundary fill. It also provides support for tangent and curvature-continuous shapes.

Advanced freeform modeling

Advanced freeform modeling extends the NX basic freeform capability to include shapes driven by laws and conics, blended shapes controlled by guiding curves, flange surfaces, guided construction of surfaces from facet meshes, midsurface construction and the ability to globally deform solid or surface shapes.

NX X Manufacturing Foundation

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

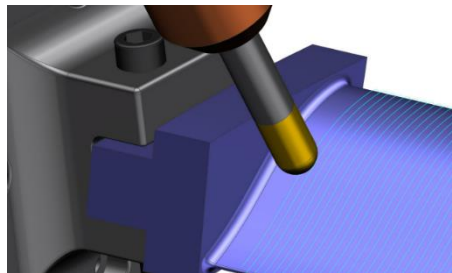
existing postprocessors with the included interactive Post Configurator application, or even build a postprocessor from scratch using Post Configurator templates. You can also use included sample posts or download a post from our support library

Shop Documentation

You can produce HTML set up documents with the included shop doc capability.

Feedrate 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 underutilization of the cutting tool.



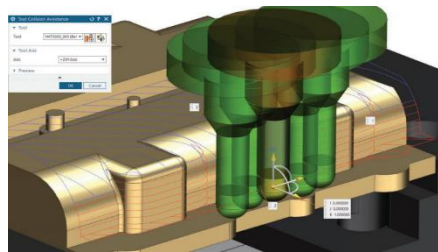
Machining Capabilities

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 flexibility to efficiently machine any hole. Special multi-axis filtering automatically switches machine coordinate system (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 machining of many features, including holes, pockets and slots.



Face Milling

Solids-based face milling automatically respects part and fixture boundaries while efficiently clearing faces. The studio task environment streamlines rendering workflows with a concise set of tools and context-specific commands that streamline setting up rendered images.

Volume-based 2.5D 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.

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.

Toolpath animation

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 recalculations.

Smooth offset cutting

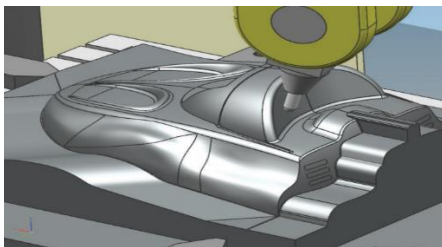
NX X Manufacturing advanced provides smooth offset pattern cuts with special stepover moves and corner treatments to maintain high feed rates and fine finishes. You can cut inside out or outside in.

Smooth transitions

Stepover and traverse motions are smoothed to minimize machine jerk and keep feed rates maximized throughout the entire operation.

Optimized cut regions

NX improves cutting and extends tool life by managing steep and nonsteep cutting regions according to the best cutting strategy. Automatic region determinations and manual control let you apply the preferred patterns to various regions of area milling and rest milling operations. Improve both finish quality and tool life by matching cutting patterns to geometry.



Interpolated passes

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

Automatic Rest Milling

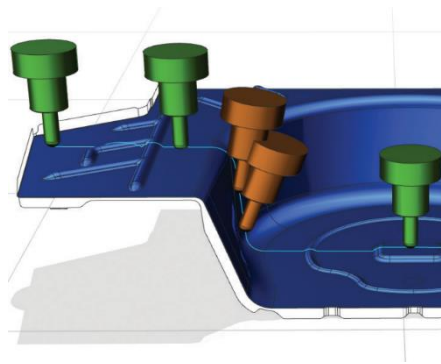
NX discovers uncut valley material remaining from larger diameter tools, including bullnose tools. You can then cut the remaining material in the corners and valleys. NX intelligently organizes valleys and fits patterns appropriate to the steep and nonsteep areas.

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 stereolithography (STL) and JT™ format for numerical control (NC) programming.

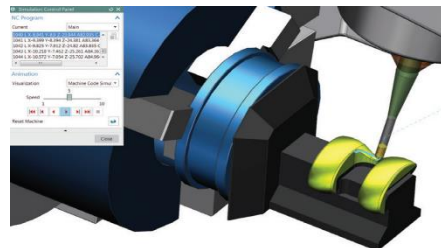


Rotary Milling

This advanced operation streamlines the programming of cylindrical parts. With minimum input required (part and blank geometry), NX generates optimized 4-axis milling operations. The advanced settings, such as controls for tool contact shift and point density, enable creation of smooth, highly efficient multi-axis roughing operations. Additionally, it can be used to machine uncut material from previous operations.

5-axis tool tip cutting

You can trace area patterns across complex geometry while specifying tool axis lead and lag relative to the surface normal.



5-axis roughing

This high-speed adaptive roughing method lets you get closer to the final shape using a minimum number of setups/operations. The adaptive milling strategy reduces heat generation and vibration in the cutting zone, which in turn reduces the wear of the cutting tools, extending tool life.

5-axis area patterns

All of the familiar 3-axis area clear patterns are provided with additional tool axis control. Raster cuts, offset patterns and interpolated cut strokes are all available.

5-axis swarf cutting

You can keep the side of the tool aligned with selected faces while controlling tilt.

Automatic 3-axis tilt

NX CAM invokes automatic tool tilt as a reaction to tool holder collisions in deep cavities. Tilting the tool over to provide clearance for the holder facilitates cutting with shorter tools and higher feed rates.

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

5-axis user control

NX CAM facilitates high levels of user controlled 5-axis milling. You can control multi-axis tool motion with traditional drive, part and check surface specifications. These flexible options make it possible to program virtually any 5-axis cut.

Productivity

Machine simulation

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

Synchronization manager

Synchronize machining operations across multiple channels. This allows you to ensure that the different tools won't start machining at the same time and cause a collision.

Post Hub

Post Hub lets you access more than 1,200 postprocessors directly from [NX X Manufacturing software](#). The extensive online postprocessor library enables you to program a wide range of milling, turning and multi-function machines. It includes postprocessors for many machines and controllers, including DMG MORI, Haas, Makino, Mazak, and Okuma.

[Explore the free 30-day NX X Manufacturing trial.](#)