

What's New in NX 7.0

fact sheet

Siemens PLM Software

www.siemens.com/nx

► Summary

Siemens PLM Software's NX™ 7.0 product development solution delivers major productivity gains for design, engineering and manufacturing. With many new and improved capabilities in modeling, digital simulation, NC programming, PLM integration and more, NX 7.0 helps customers boost efficiency and reduce costs throughout the product development process.

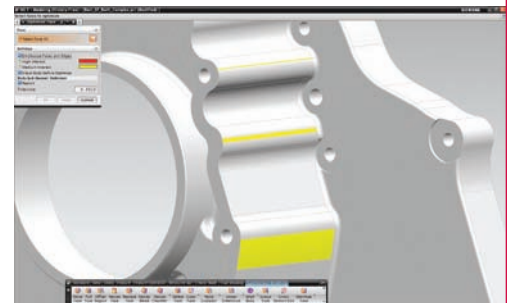
Benefits

- More rapid and efficient product development
- Improved use of imported models
- Faster, easier geometric modeling
- Accelerated CAE model preparation
- Faster, more productive NC programming
- Better leveraging of PLM information
- More intuitive reporting for improved decision-making
- Faster validation and issue resolution
- Improved automation capability

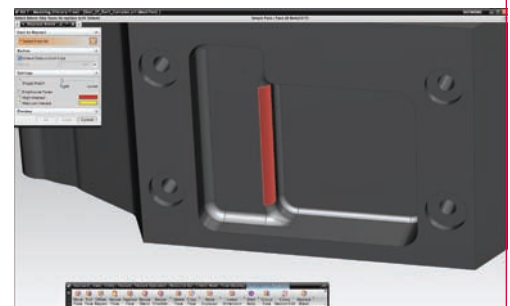
NX 7.0 Design

NX includes many enhancements to synchronous technology, the breakthrough technology introduced in NX 6 that unites constraint-driven and history-free modeling. Improvements in the new release include support for a much greater range of parts and geometries, improved workflows for multi-CAD environments and streamlined methods for geometry re-use.

Streamlined workflows for imported geometry. NX streamlines the use of imported or translated geometry with new capabilities for face optimization and blend replacement. To optimize faces, the software simplifies surface types, merges faces, improves edge accuracy and recognizes surface blends. NX 7.0 can also convert imported B-surfaces to rolling-ball blend faces, which are more easily edited with dimensional parameter changes. On native or imported models, designers can assign chamfer properties to angled faces and resize them, adding offsets and angles, regardless of the feature history.



Face optimization streamlines work with imported models.



NX 7.0 replaces imported b-surfaces with rolling-ball blends for easier editing.

Feature creation option eases subsequent changes. NX 7.0 gives designers the option to create parameterized features when using history-free modeling with holes, edge blends and chamfers. With this option, the feature parameters are maintained so that the geometry can be subsequently changed with parameter inputs.

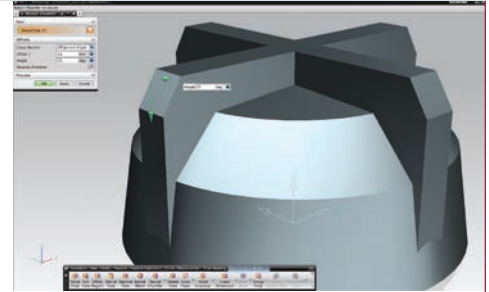
Improved history-free assembly modeling. The ability to move faces in history-free mode has been enhanced in NX to operate simultaneously with multiple component faces in assemblies. Designers simply change the selection scope to include the entire assembly to extend the capability beyond the active part.

Improved pattern modeling. Face pattern operations in history-free mode create pattern features in the part navigator that can be more easily edited. When designers move or pull faces or offset regions on any instance of the pattern, all instances are updated. Other features like blends, chamfers and holes applied to an instance of the pattern are also updated automatically when the pattern is edited.

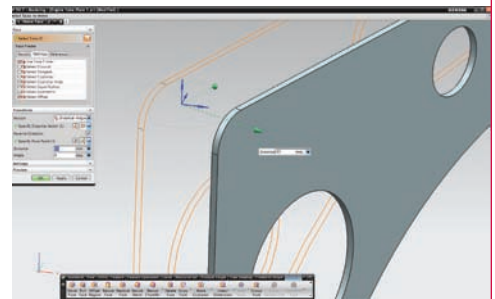
Improved handling of thin-walled parts. Many face editing commands are improved with an option that simplifies selection of faces that are offset from each other. The capability recognizes thickness (for example, ribs) in thin-walled parts, simplifying synchronous modeling with plastic and sheet metal parts.

Better positioning, design intent in synchronous mode.

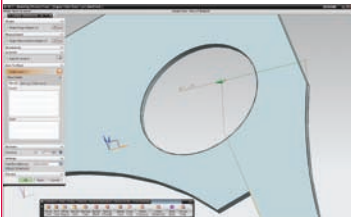
NX 7.0 adds dimension locking and fixed constraints to prevent changes in size or position. A new command adds a 3D fixed constraint to selected faces to establish desired behavior. Linear, angular and radial dimensions include a locking option in history-free mode. These tools effectively add design rules to models that have no history or parametric behavior. Designers can highlight and review fixed constraints and locked dimensions with a new display command.



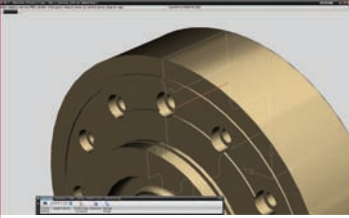
Designers can assign chamfer properties to angled faces to facilitate resizing.



Selection of offset faces simplifies modeling with molded and sheet metal components.



Dimension locking helps fix size or position to maintain design intent.



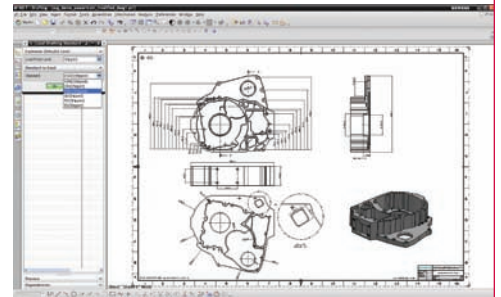
NX 7.0 supports model changes through edits to cross-section curves.

Streamlined cross-section editing. NX 7.0 simplifies cross-sectional based changes to 3D models in history-free mode. Designers can section models and edit the model or its features by changing its cross-section curves.

Better shape evaluation. NX 7.0 includes curve shape analysis in the core modeling toolset. Designers can analyze curves and edges with curvature comb displays, with full control of cap lines, the number and color of needles, scale and scale factor. NX also displays curvature peak points and inflection points. In addition, designers can evaluate continuity between curves and reference objects to check deviations such as variations in position, tangency and acceleration. The new tools are especially useful in surface modeling to verify the continuity between curves used to create surfaces.

NX 7.0 Drafting

Improved drafting compliance. NX incorporates two new drafting options that automatically configure annotation and drafting view preferences to Chinese (GB) and Russian (ESKD) standards. Designers can select either of these options to configure more than 200 standards-compliant settings in the drafting and 3D annotation environments.

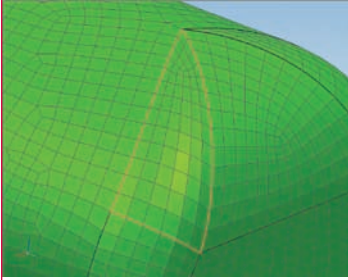


NX 7.0 drafting supports Chinese and Russian drafting standards.

NX 7.0 Digital Simulation

Synchronous technology for digital simulation. NX enhancements to synchronous technology modeling tools benefit simulation by accelerating the CAE model preparation process on native or imported geometry. Synchronous modeling can be used by CAE analysts for geometry cleanup and idealization, providing a more efficient alternative to the limited geometry capabilities of standalone CAE preprocessors and the complexity of history-based CAD. With NX 7.0, CAE specialists can significantly reduce the time required to fix inaccuracies such as gaps or slivers resulting from incomplete import of geometry, and idealize models by eliminating features that are irrelevant to the analysis.

Faster geometry idealization and abstraction. NX 7.0 further accelerates model preparation with improved midsurface generation performance, more precise locations for edge splitting operations, and automatic generation of mesh mating conditions for geometry that has been divided into multiple bodies.



NX 7.0 introduces a quad-only meshing option.

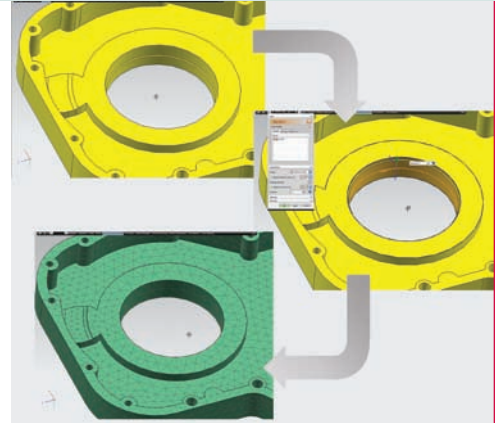
Improved meshing. NX 7.0 introduces support for Nastran pyramid elements and Abaqus gasket elements. Memory management for tetrahedral meshing has been enhanced to significantly improve meshing performance. An option for quadrilateral-only meshing has been added for analyses in which the presence of triangular elements is undesirable or unacceptable. A new NX capability creates a surface coating of 2D elements on top of 3D solid elements, using the nodes and connectivity of the underlying solid mesh.

Enhanced materials capabilities for CAE. NX 7.0

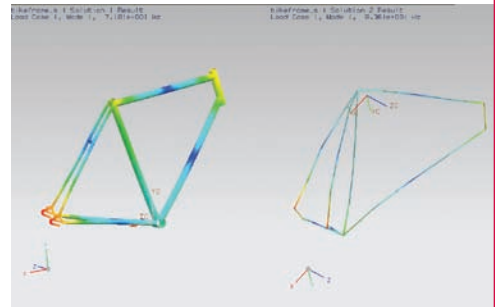
Advanced Simulation includes several enhancements to materials for simulation. Analysts can now maintain separate custom material libraries and create, edit and delete library materials directly in NX. Enhancements for isotropic and fluid materials allow entry of expressions (formulae, functions, references and constants) as property values, as well as specification of units. Plotting of tabular values is also supported. Hyperelastic material models have been added for Nastran, Abaqus and ANSYS solvers.

Improved motion simulation. The NX Motion Simulation Joint Wizard has been improved to automatically convert assembly constraints (as well as legacy mating conditions) to the appropriate links and joints. In previous releases, only mating conditions were supported. For assembly constraints, the wizard now creates the appropriate joint type based on the degrees of freedom in the components referenced by the assembly constraint.

FE Model Correlation. NX Finite Element (FE) Model Correlation software enables users to quantitatively and qualitatively compare simulation and modal test results, as well as compare two different simulations. Tools are provided to geometrically align the models, pair the modes from both solutions, view mode shapes side-by-side and calculate and display correlation metrics. NX FE Model Correlation is available as an add-on to both NX Advanced Simulation and NX Advanced FEM, enabling you to leverage the power and ease of use of the NX environment.



Synchronous technology aids in idealizing models for analysis.



Correlation compares simulation and modal testing results.

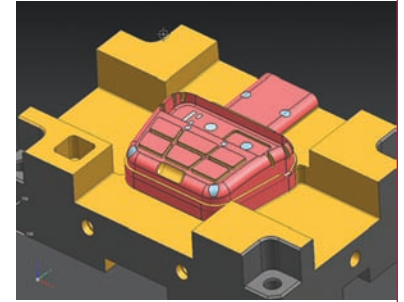


With NX 7.0 companies can more easily automate simulation processes.

Efficiency through automation. With NX 7.0, the NX Open application programming interface has been enhanced to include finite element modeling, solution and post-processing. Using NX Open, companies can automate repetitive tasks and capture CAE process knowledge. Automated simulation processes can then be distributed to other engineering and design departments, enabling them to run simulation and follow CAE best practices. The result is fewer bottlenecks, more efficient design-analysis cycles, and more timely and accurate simulation.

NX 7.0 Manufacturing

Synchronous technology for manufacturing. The new synchronous technology modeling tools in NX benefit manufacturers in several ways. They accelerate cleanup of imported data that has been incompletely or inconsistently translated, and eliminate changed iteration between suppliers and the manufacturing process. Synchronous modeling is also ideal for removing or simplifying features to optimize NC programming (for example, removing features produced by electrical discharge machining) and also in creating as-cast models of blank castings from the as-machined model. For fixture assemblies, synchronous modeling simplifies and accelerates fixture changes when parts are modified.



Synchronous modeling is ideal for optimizing models for NC programming.

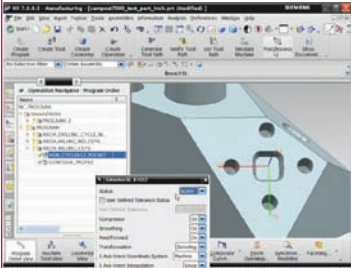
Faster tool path processing. NX 7.0 accelerates NC programming with parallel generation of NC tool paths, which allows simultaneous NC programming and tool path processing using interactive multi-process computing. NX CAM enables users to generate tool paths in an external process while NC programming continues, and takes advantages of multiple processors and cores when available. The result is a 50 percent reduction in tool path calculation time, depending on hardware.

CAM, postprocessing for greater machine efficiency. The latest release of NX CAM enhances the NC programming and postprocessing with built-in support of Siemens SINUMERIK-controlled machine tools. The SINUMERIK controllers feature many unique, high-productivity capabilities, including cycle extensions and specialized commands to maximize machine performance. These capabilities are explicitly supported by NX CAM, and the integrated post builder includes templates specifically tailored to the controllers.

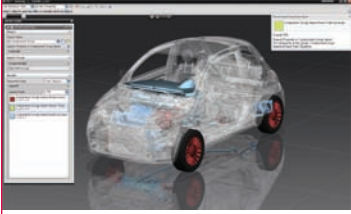
HD3D Visual Reporting and Validation

NX 7.0 introduces HD3D, a new capability for displaying and interacting with product and process information directly in the 3D environment. HD3D combines the powers of NX and Teamcenter® PLM solutions to visually deliver the information you need to understand, collaborate and make decisions in globally distributed product development.

HD3D delivers an intuitive, easy-to-use method for collecting, collating and presenting information. It visually reports product and process data with 3D product models, enabling quick comprehension, interactive navigation and drill-down, and immediate answers to critical questions.



NX 7.0 Manufacturing includes built-in support for Siemens SINUMERIK controllers.



HD3D Visual Reporting uses color-coding, legends and transparency to visually synthesize product and process data.



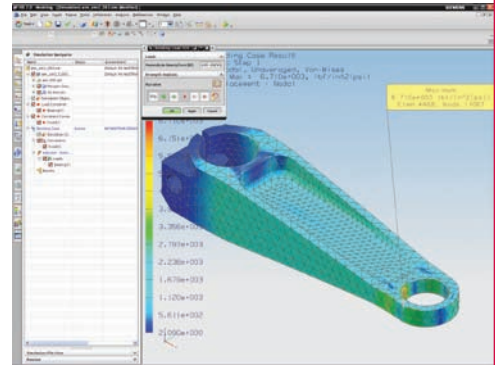
HD3D Validation visually presents results of validation checking.

HD3D Visual Reporting facilitates decision-making. HD3D Visual Reporting, a new module available with NX 7.0, enables users to interactively create, edit, execute, save and share reports based on product and process criteria. Users can select report properties from any NX part properties and attributes or any Teamcenter-managed properties. Reports can be tailored to a selected scope of components. Report results are displayed in the NX graphics window, with color coding of component models, tagging, legends, transparency and charts. The visual presentation and analysis of information provides a superior, more efficient alternative to navigating, collating and processing lists and tables of data that are typical of PLM reporting. HD3D Reporting helps users to efficiently determine project status, identify recent changes, recognize and resolve issues and problems, determine ownership, suppliers, costs and other information through ad hoc queries.

HD3D Validation accelerates checking, improves quality. HD3D is also implemented in NX Check-Mate, a standards-based validation checking application that ensures consistency and quality in CAD data and monitors compliance with corporate and industry standards. HD3D enhances the validation tools within Check-Mate with a new interactive visual user interface for defining validation and running validation checks, reviewing results and resolving issues in active NX sessions. Graphic tags can be displayed on 3D product models to indicate validation test results, for fast visual identification of test status and issues. For faster issue resolution, designers can select and open parts directly from the graphic tags to edit the model and correct problems.

NX 7.0 Product Template Studio

Improved automation and re-use. NX 7.0 greatly expands the capabilities of Product Template Studio (PTS), a tool for building re-usable templates from existing models. PTS now creates templates from finite element and motion analysis context, and can also create drawings when a template is executed. The support for simulation objects and solutions enables companies to easily capture and re-use best practices in motion and finite element analyses. Visual rules added to PTS in this release allow companies to add rules-based logic for control and configuration of templates using graphical techniques rather than programming code. The visual rules significantly lower technical barriers to development of more advanced product templates.



Product Template Studio now creates templates that include simulation and drafting.



Contact

Siemens PLM Software

Americas 800 498 5351

Europe 44 (0) 1276 702000

Asia-Pacific 852 2230 3333

www.siemens.com/plm