What's new in NX 7

Benefits

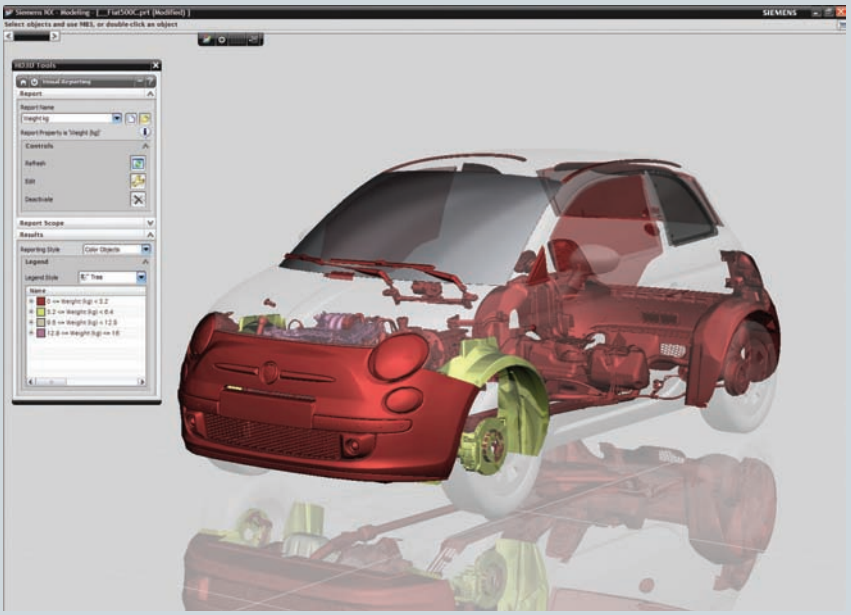
- More intuitive reporting for improved decision-making
- More rapid and efficient product development
- Better leveraging of PLM information
- Faster validation and issue resolution
- Faster, easier geometric modeling
- Improved use of imported models
- Accelerated analysis processes
- Multiple CAE solutions in one system
- Better confidence in CAE predictions
- CMM inspection programming time reduced by up to 80 percent
- NC programs generated for blisks and impellers in half the time
- Tool path calculation time reduced by 50 percent

Summary

Siemens PLM Software's NX™ 7 product development solution redefines productivity in design, engineering and manufacturing. With many significant new and enhanced capabilities in modeling, digital simulation, tooling design and NC programming, the latest release of NX 7 helps companies improve product development efficiency, speed and quality while reducing costs. NX 7 also supports our high definition PLM technology framework to improve decision-making throughout the product development process.

High definition 3D (HD3D)

With the high definition 3D (HD3D) environment, Siemens PLM Software introduces HD-PLM technology into its product development solutions. HD3D unites the powers of NX and Teamcenter® software under the HD-PLM technology framework to visually and intuitively deliver the information you need to understand, collaborate and make decisions in a globally distributed



HD3D Visual Reporting leverages a wide range of company information for key design decision support.

NX

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What's new in NX 7

product development environment. HD3D delivers a product development decision support and analytics solution that places users in the proper context, assists with task accomplishment, presents information intuitively and validates decisions against established rationale.

HD3D enables you to visually comprehend PLM data with interactive navigation, and to drill down to details as needed. By viewing a product's 3D representation, you can easily establish the right context to instantly answer questions about project status, design changes, team responsibilities, issues, problems, cost, suppliers and other attributes. Color-coding, onscreen tagging and legends enable fast visual assessment and interpretation of product development issues and decision criteria.

Information drilldown is enabled using interactive tags and detail dialogs.

HD3D Visual Reporting HD3D Visual Reporting is a new product for displaying and interacting with product and process information directly in the 3D environment. It enables you to interactively create, edit, execute, save and share reports based on product and process criteria. You can select report data from any NX part properties and attributes, or from 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. HD3D reporting helps you 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 Visual Reporting includes a set of out-of-the box reports that provides your design teams with answers to commonly asked questions. On installation, reports related to ownership, check out, part maturity, projects, load status and more are available for instant use. You can readily change the input criteria of any report as needed, or generate custom reports with a report definition dialog.

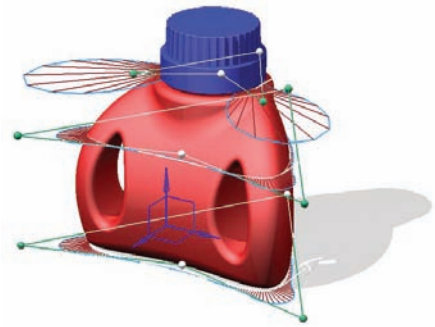
HD3D validation accelerates checking, improves quality HD3D is also implemented in NX Check-Mate, a standards-based validation checking tool that ensures consistency and quality in CAD data and monitors compliance with corporate and industry standards. HD3D enhances the validation tools with a new interactive 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, you can select and open parts directly from the graphic tags to edit models and correct problems.

Improving issues communication, speeding resolution The HD-PLM technology framework also accelerates issue resolution and improves collaboration. The HD3D Check-Mate interface links results of validation checking with issues management tools in Teamcenter. You can instantly create product issues from within HD3D Check-Mate to share visual representations with the product team in Teamcenter's community collaboration solution as part of the engineering review and approval process. The intuitive visual presentation assists product teams in clearly understanding engineering concerns, with systematic tracking for faster issue resolution.

NX design

Faster sketch-based modeling NX includes new tools that create geometry from scratch 50 percent faster. With direct sketching, automatic dimensioning and one-pick positioning, you can quickly and easily draw, constrain, position and edit sketches, then extrude, revolve or sweep them to build geometric features. NX 7 infers design intent for creating, uniting or subtracting geometry in Boolean operations to reduce user interaction and accelerate feature creation. Sketches and 2D objects can be easily stored, shared and retrieved through the NX Reuse Library.

Streamlined freeform modeling The latest release of NX transforms freeform design with synchronous technology, delivering an advanced shaping toolkit that works with any geometry – even imported models – and eliminates 50 percent or more of the labor-intensive steps required in conventional freeform modeling.



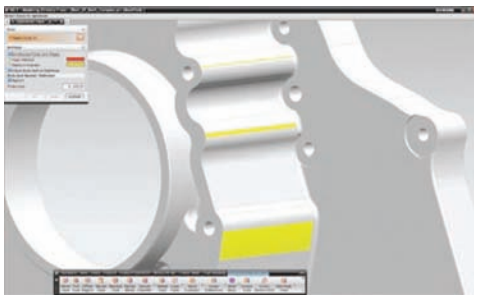
New freeform modeling tools create isoparametric lines for fast and easy modification.

The freeform modeling tools allow you to begin with solid or surface, analytic or B-rep geometry. You can insert isoparametric curves and easily model organic forms by moving constraint points, surface poles and handles. With simple push and pull shaping techniques, you don't have to be a surface design expert to create complex freeform models. Surface selection and analysis tools help you control continuity, blending and inflections to create designs with greater aesthetic appeal. With these new, unique tools, you can create and modify complex freeform geometry with exceptional speed, versatility and control. And, your freeform models are completely re-usable, from concept through production.

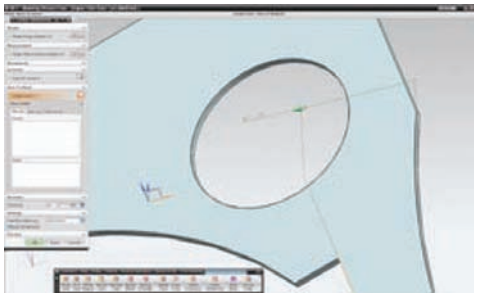
Synchronous technology enhancements accelerate design NX brings you the best of both worlds, combining the power of history-based modeling with the speed and flexibility of synchronous technology. With extensive improvements to synchronous technology in NX 7, you can create and modify designs faster and easier, and work with data from other CAD systems more efficiently.

NX streamlines the use of imported or translated geometry with new capabilities for optimizing imported models. To optimize faces, the software simplifies surface types, merges faces, improves edge accuracy and converts B-surface to rolling-ball blend faces for easier editing. Other synchronous modeling enhancements include improved selection intent options and new tools for working with patterned features, thin-walled parts, assembly operations and cross-section editing.

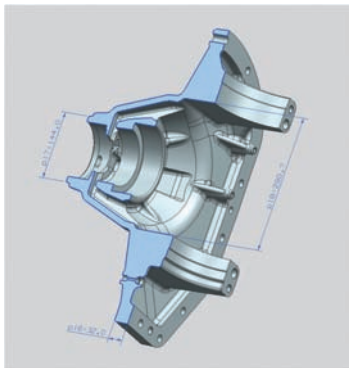
Designers can directly move surface poles with advanced shaping capabilities in NX.



Face optimization streamlines work with imported models.



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

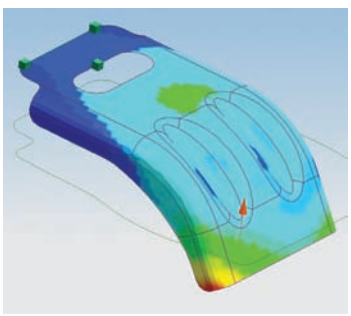


With NX 7, designers can easily change models by editing cross-section curves.

Add design intent and intelligence as needed NX 7 gives you the option to create parameterized features when using synchronous modeling. With this option, feature parameters are maintained so that the geometry can be edited more easily. NX 7 also adds dimension locking and fixed constraints to prevent changes in size or position. These tools effectively add design rules to models that have no history or parametric behavior.

Accelerating re-use The NX Reuse Library has been enhanced with an improved "component first" process for adding library components to NX models. When you add a component from the Reuse Library to an assembly, you can now select the component and place it before creating its mounting holes. Mounting holes associated with components are automatically cut when you insert the component into the assembly. This streamlined workflow is more intuitive and accelerates the assembly process by combining and automating multiple operations.

New add-on application for flattening, unforming and formability analysis NX One-Step Formability Analysis is a new application that can eliminate costly physical try-outs and save time on redesigns. By using fast and highly accurate FEM-based calculations to analyze complex sheet metal part designs for manufacturability considerations, you can predict thinning, stress, strain and springback. You can also use this application to generate an accurate flattened blank profile.



You can predict formability and generate the flattened blank profile for complex-shaped freeform geometry.

NX 2D design and drafting

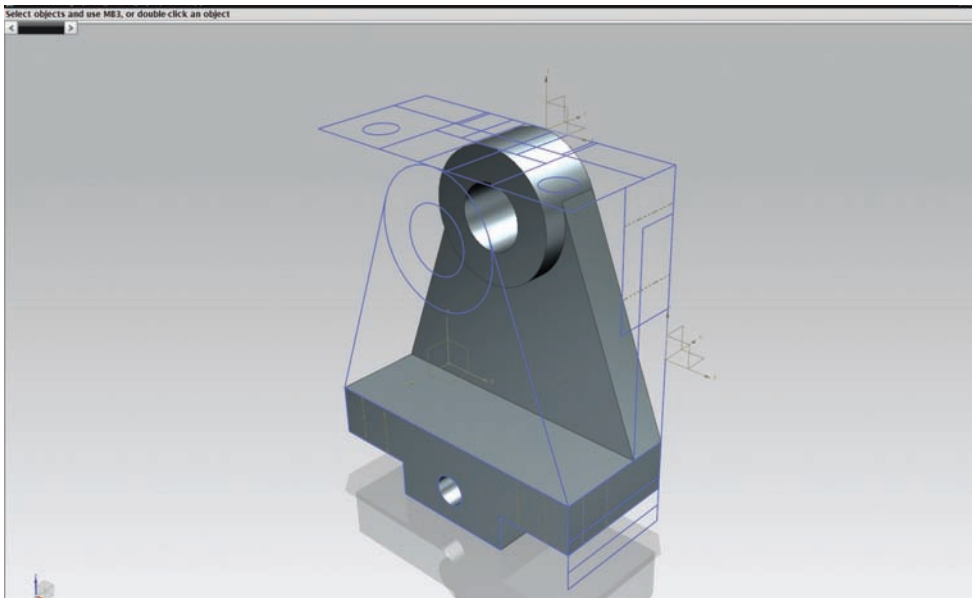
Advanced 2D design and drawing: NX DraftingPlus NX DraftingPlus is a new product devised to meet the requirements of 2D-centric design and drawing, as well as to support 2D-to-3D workflows. It is fully integrated with NX Drafting and offers a set of 2D-centric drawing tools for customers who require 2D design and layout capabilities, or who want to maintain or re-use legacy 2D drawings.

The advanced functionality provided by NX DraftingPlus is accessed from NX Drafting toolbars and menus, enabling use of the NX interface and all interface customization functionality such as roles and radial menus. NX DraftingPlus functionality can be used while creating drawings from 3D models or to create 2D designs from scratch, while maintaining full data compatibility with NX.

NX DraftingPlus provides a robust curve creation and modification toolset for 2D design. New Sketcher functions such as automatic dimensioning and NX DraftingPlus tools like infinite line provide better support for 2D-centric workflows. View creation and orientation tools along with standard views, project to view, dynamic preview and intelligent margins give NX DraftingPlus cutting-edge view creation capabilities.

To provide a single, consistent mechanism for working with custom symbols, both NX Drafting and NX DraftingPlus now leverage the NX Reuse Library for managing symbols. NX DraftingPlus supports direct import of I-deas® software symbols and drawings, ASC/DWG, DXF/DWG data and IGES data. This capability helps you leverage existing investments in I-deas 2D data, or data from most other 2D systems.

The Copy to 3D capability in NX DraftingPlus provides a streamlined path for converting 2D designs into 3D models, without recreating the geometry. This optional workflow enables you to move old 2D designs into new versions as 3D models.



NX DraftingPlus automates conversion of 2D designs to 3D models.

Standalone 2D CAD solution: NX Power Drafting The capabilities in NX DraftingPlus are also available in a fully functional, standalone solution called NX Power Drafting. It can be used to create detailed production drawings using NX 3D solid models and assemblies, or as a standalone high-performance 2D design and drawing system.

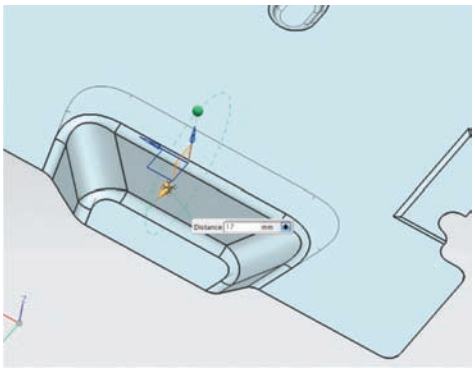
Improved drafting compliance NX has improved local drafting standards support with a recent focus on Chinese (GB) and Russian (ESKD) standards. Designers can select any of these options to configure more than 200 standards-compliant settings in the drafting and 3D annotation environments.

NX 7 digital simulation

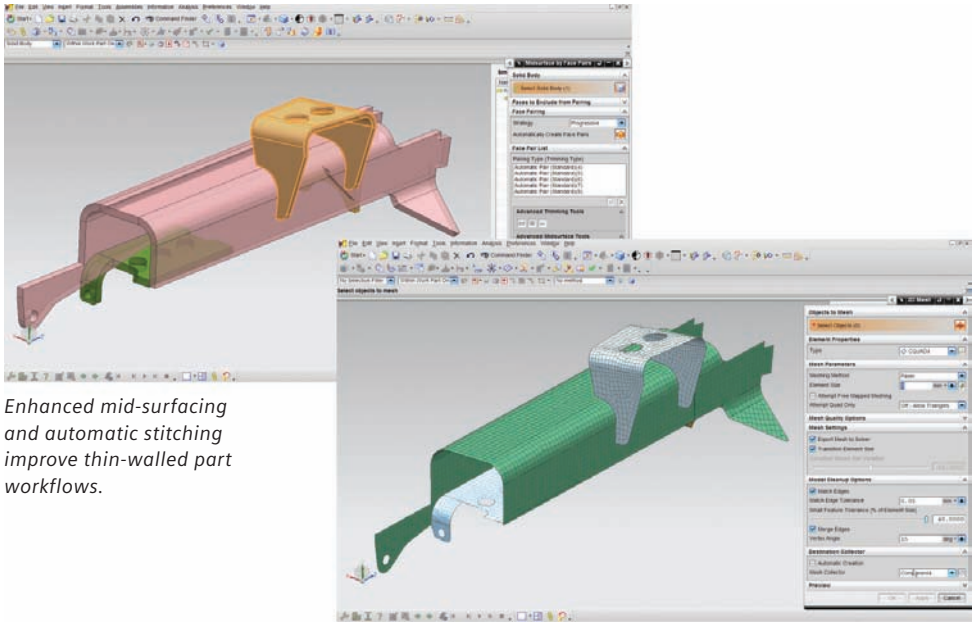
The latest version of NX 7 significantly enhances the NX Digital Lifecycle Simulation portfolio with increased modeling efficiency and an expanded range of integrated solutions. New product offerings in NX 7 include durability analysis and model correlation. Additionally, NX 7 offers major enhancements in the areas of multi-body dynamics, modeling of thin-walled structures, beam modeling, meshing, laminate composites and support of multi-physics solvers.

Integrated design and CAE

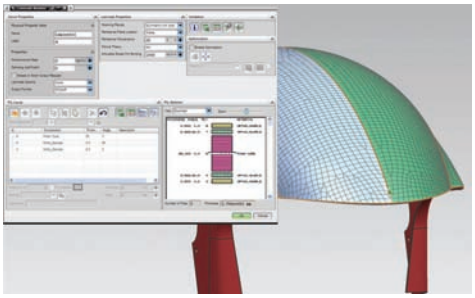
Synchronous technology for CAE The integrated design and CAE environment in NX offers a more efficient alternative to the limited geometry capabilities of standalone CAE preprocessors. Analysts can use synchronous modeling on both native and imported geometry for cleanup, idealization and easy creation of design alternatives. With NX, 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. Improvements in the latest version include enhancements on working with sheet body geometry.



You can use synchronous technology to edit sheet body geometry to quickly analyze design alternatives.



Enhanced mid-surfacing and automatic stitching improve thin-walled part workflows.



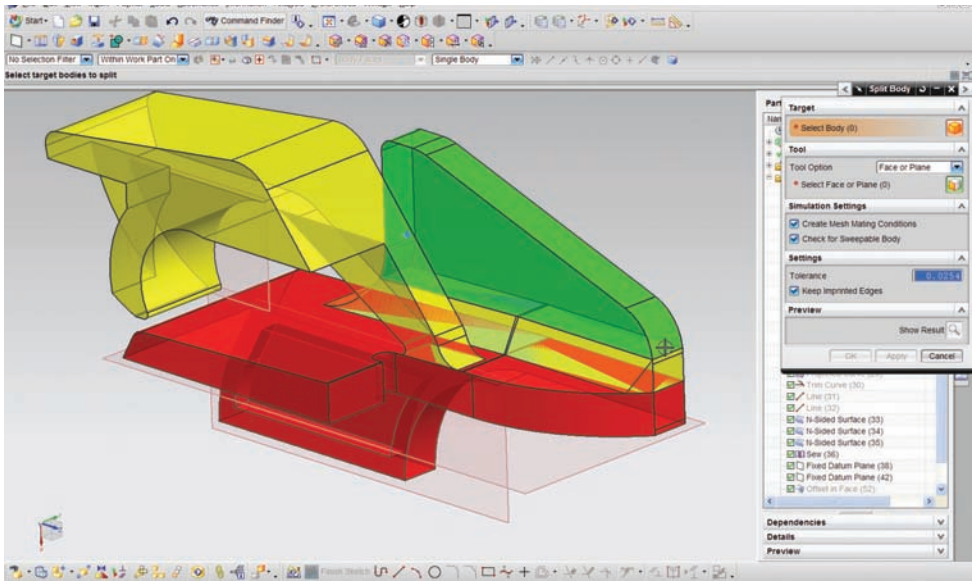
You can now employ easy-to-use laminate composite definition capabilities.

Laminate composites Analysts can evaluate structures made with laminate composite materials using the extensive and easy-to-use ply and laminate definition capabilities in NX. With NX 7, users have the ability to import ply data from FiberSim and also place plies on imported FE mesh models.

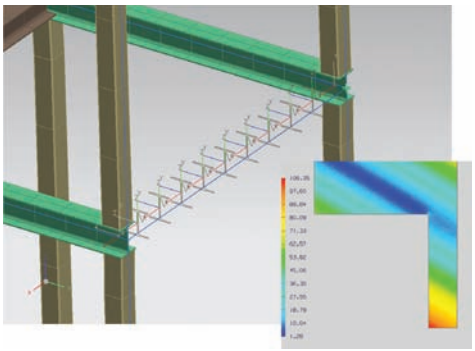
New thin-walled part workflows New workflows in NX 7 deliver up to 50 percent improvements in the time required to replace solid geometry with surface geometry needed to analyze thin-walled parts. NX takes advantage of improved mid-surfacing and automatic free edge stitching capabilities which speed the initial modeling and automatically update the analysis model whenever the base geometry changes.

Improved hexahedral mesh workflows Analysts often resort to trial-and-error steps when trying to partition solid geometry to make sweepable meshes in traditional tools. NX reduces this trial-and-error process by up to 80 percent, depending on part geometry, by visually indicating whether a body can be swept-meshed before analysts even attempt to build their mesh.

Beam modeling In traditional CAE tools, engineers routinely replace complex geometric cross sections with equivalent line elements used for analysis. However, the single-line element representations seen in typical preprocessors are hard for engineers to intuitively understand. NX 7 increases productivity by streamlining beam cross section creation and provides engineers with an intuitive way to rapidly visualize and validate their models.



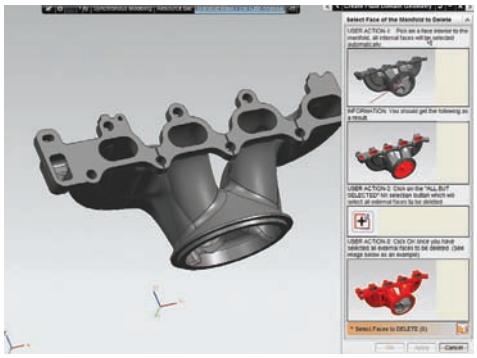
NX indicates whether a solid bodypartition can be swept meshed by color-coding partitions before you mesh.



You can intuitively visualize beam elements using the NX geometry engine.

Automated bolted connection modeling Using traditional methods, engineers spend many hours manually creating rigid element spiders to model bolted connections. NX 7 assists engineers by automatically finding and using the design data needed by this process and enabling them to create large numbers of connections with a single mouse click.

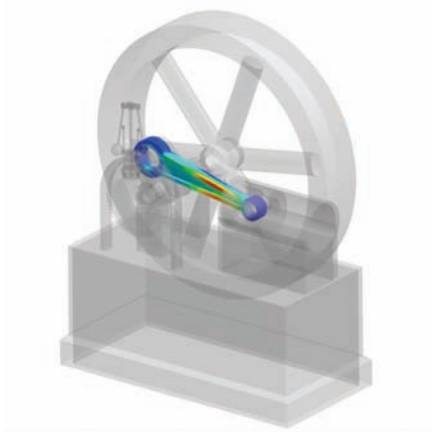
Efficiency through automation The NX Open application programming interface has been enhanced to include finite element modeling, solution processes and post-processing commands. Using NX Open, you can automate repetitive tasks and capture CAE process knowledge. Engineering teams can then distribute these automated simulation processes to other engineering and design departments, enabling them to more easily perform simulation and ensure they follow CAE best practices. This approach results in fewer bottlenecks, more efficient design-analysis cycles and more timely and accurate simulation.



You can automate repetitive analysis tasks with customized workflows and dialogs.

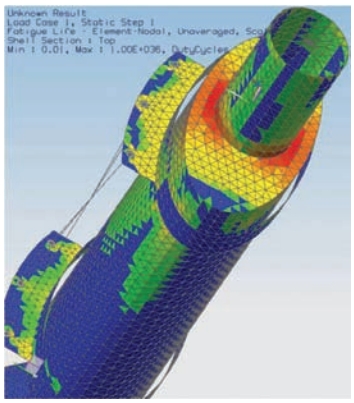
Integrated multi-physics solutions

Flexible-body dynamics. While many products offer kinematic simulation of rigid mechanisms, NX Motion is unique in offering an integrated dynamic simulation environment. Unlike kinematic simulation, dynamic motion simulation takes mechanism compliance into account through the use of bushings, springs and dampers, and now flexible bodies. Part flexibility is important because it changes the geometry of the mechanism and can lead to serious design issues. Engineers can validate designs using higher fidelity models that combine elastic deformation and rigid body motion to evaluate conditions such as sharp impacts, sudden changes in motion or the effect of component flexibility on mechanism performance and durability.



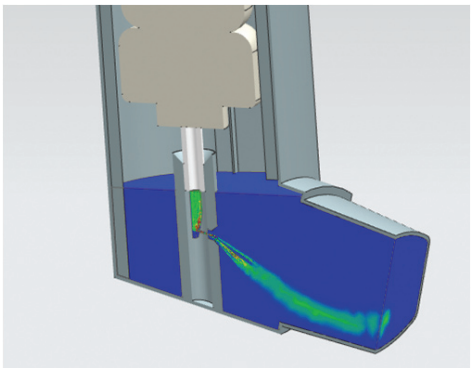
You can use flexible body dynamics to understand part performance under real operating conditions.

Durability Engineers use structural fatigue analysis to predict design life or durability. Analysts can check whether in-service loadings are likely to cause early failure in parts subjected to heavy vibration or cyclic loads. NX 7 introduces two new products for evaluating durability – a durability wizard and an advanced durability solution. The NX durability wizard provides an intuitive, guided process for designers and less experienced users to compute strength safety factors and fatigue life. The advanced durability solution assists expert users in performing more complex analyses with support for static and transient loads and the ability to handle bi-axial stresses and notch effects.



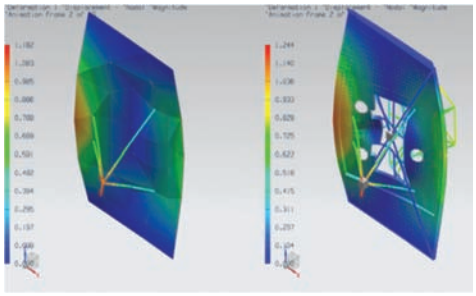
Durability analysis checks whether cyclic loads will cause early failure.

Flow (CFD) and thermal analysis NX enhances flow and thermal analysis productivity with new capabilities to model complex systems. Examples of complex systems include gas-gas mixtures that contain miscible species and systems with rotating and non-rotating flow. Users can get results faster by using the new fractional step integration scheme when solving long, transient problems. Visualization of tracer fluid objects and 3D flow results for relative/absolute pressure and shear resultant provide users with an intuitive understanding of fluid system behavior. NX 7 users also benefit from improved performance and accuracy of the thermal solver for models with articulation. Users will also experience better convergence of simulations involving transient phase changes through the ability to lock temperatures.



Enhanced flow analysis can be used when modeling complex systems.

Solver support NX 7 expands support for multiple solver solutions. New support for hyperelastic material models in Nastran, Abaqus and Ansys helps engineers working with nonlinear materials to better understand their impact on product performance. Users can evaluate the stability of servomechanisms and rotating systems with new support for NX Nastran® solutions SOL 107 and 110 for direct and modal complex eigenvalue analysis. NX also supports NX Nastran SOL 200 optimization analysis and the new NX Nastran flexible body solution.



Correlate analysis results against physical test data to gain confidence in a simulation.

Integrated analysis/test correlation
Before a test can be replaced, engineers need to demonstrate that the analysis model can match measured data. A high degree of match, or correlation, between the analysis model, results and test data gives confidence that the analysis model and solution can be used to predict performance. Two new products enable rapid model correlation and model updating.

NX finite element (FE) model correlation
This new product enables users to quantitatively and qualitatively compare simulation and modal test results, as well as compare two different simulations. The insight provided by viewing test and analysis mode shapes side-by-side is supplemented by a range of options to calculate and display correlation metrics.

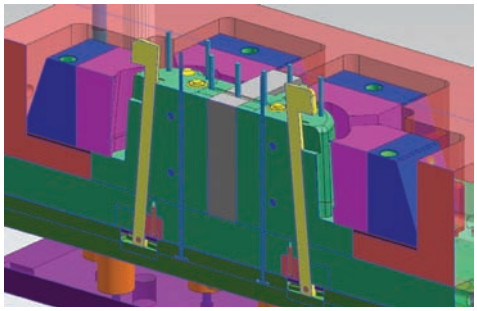
FE model updating NX 7 introduces this new product to rapidly update analysis models to better match test results. Through an intuitive workflow, analysts can quickly update an FE model through an optimization process which calibrates and fine tunes model parameters, such as stiffness or mass properties, to better match the analysis results to the test results.

NX for tooling and fixture design

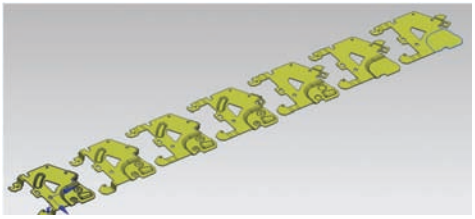
Improved mold design NX 7 significantly reduces the time needed to design and validate molds. The new HD3D visualization tool for mold design validation assists both part and tool designers with the task of checking and correcting part design issues related to manufacturability – such as poor data quality, undercuts and draft angles.

The new cooling circuit design provides onscreen drag handle interaction for rapid creation and adjustment of cooling lines. Cooling lines can be automatically positioned to fit within a specified 3D zone surrounding the part.

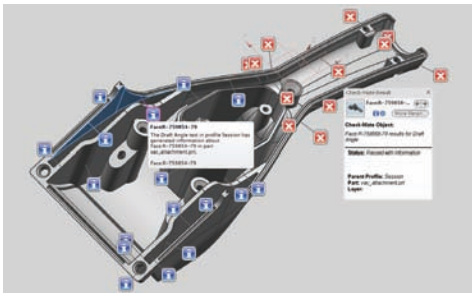
The new motion simulation enables you to dynamically analyze a fully detailed tool operating in the press machine in order to validate and correct interferences and collision issues prior to build.



Validate and correct complex dynamic interference and collision conditions prior to physical try-out (applied to molds and dies) with the new motion simulation capabilities.



Easily create, visualize and re-order bends to design a strip layout optimized for cost and speed.

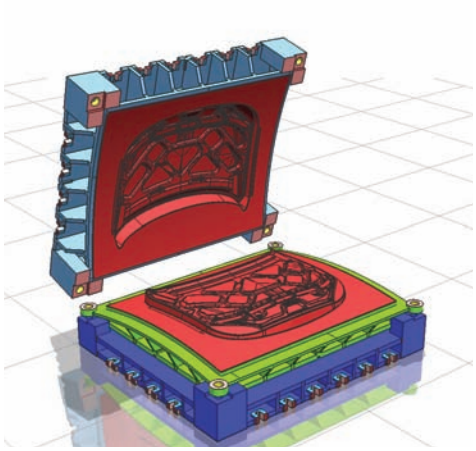


Identify and correct problematic data quality, undercuts and shallow draft issues with visual reporting tools for mold design validation.

Improved progressive die design NX delivers new capabilities that maximize speed and flexibility when designing progressive dies; these capabilities also can save on tool cost. The new workflow for unbending and flattening assists you with creating, visualizing, manipulating and re-ordering bends. Full and partial bending capabilities (including bend allowance) and the freedom to visualize bend progressions from “blank to part” and “part to blank” provides the flexibility to quickly optimize the strip layout for cost and speed.

Die structure design and detailing have also been enhanced to automate design of piercing punches, forming punches and dies. You can now create complex forming punches, including auxiliary mounting and strengthening features, with only a few mouse clicks.

The new motion simulation helps you find and correct dynamic interference and collision issues prior to tool build. You can simply simulate the detailed die as if it was operating in the press machine – including strip lifting and advancement.



Use a library of standard sections to accelerate addendum and draw bead design.

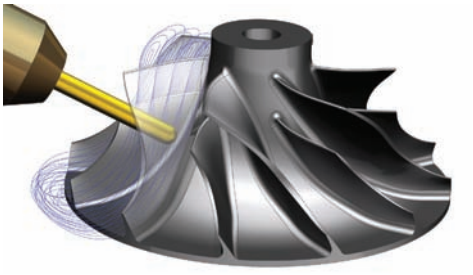
Improved stamping die design NX delivers new capabilities that increase die development productivity. The new die operation line-up assembly structure allows multiple designers to concurrently design many detailed die operations. Additionally, the die face design module has been enhanced for design productivity. It includes access to user-defined addendum and draw bead sections defined in the Reuse Library. The creation of complex binder shapes (planar, cylindrical and conical) and patch openings (joggles and notches) have also been automated. The newly enhanced springback compensation capability can morph the as-designed die face to a calculated springback shape (finite element mesh or cloud of points), and a new function that automatically reduces die face radii has also been implemented.

Improved electrode design You can save material costs by using the same electrode in multiple positions. New tools assist in designing multi-use electrodes with multiple sets of burn surfaces that correspond to different burn positions on the workpiece. Automated checking ensures that these complex electrodes do not produce unexpected burns in their various positions.

NX for machining

New application for blisk and impeller machining NX Turbomachinery Milling is a new application that accelerates the NC programming process for complex 5-axis multi-bladed rotational parts. This application puts you in the context of blisk and impeller machining and offers a set of automated, process-specific functions that simplifies the generation of smart tool paths for these types of parts. The application steps you through the process of programming with the specialized operations for 5-axis roughing, rest milling and finishing of blades, splitters and hubs. Precise control is provided for details such as cut patterns, cut level offsets, edge treatments, tool angle, smoothing options and optimized non-cutting moves. Built into NX CAM, this application is complete with simulation, tool path optimization and a wide range of supporting functions.

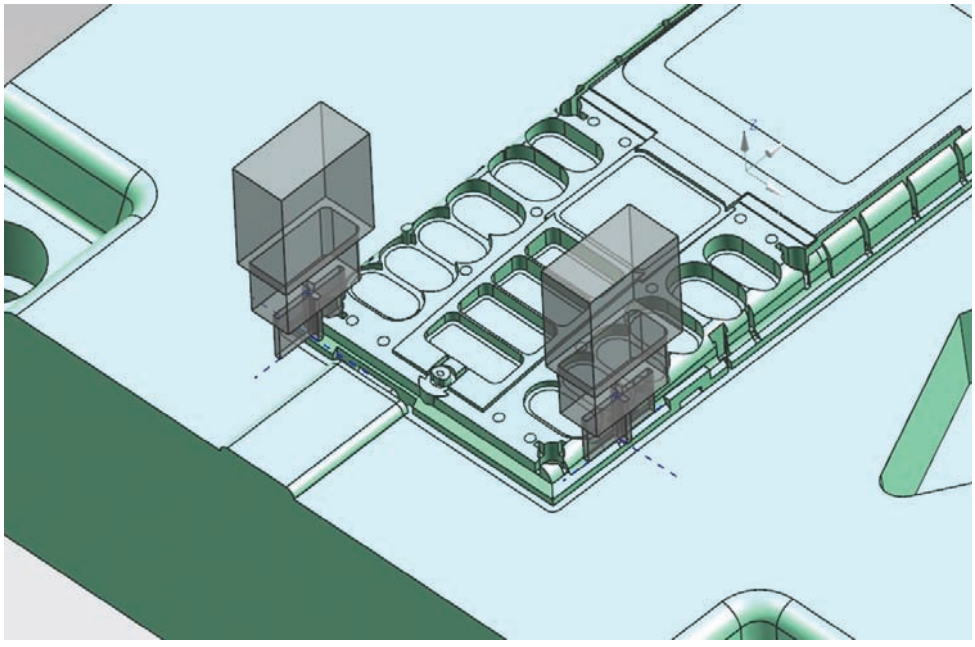
New interactive programming controls You can use manually-driven tool axis and tool path control capabilities to rapidly create and adjust tool paths with a "virtual jog mode" interaction. This control is especially useful for creating simple clean-up passes and also for editing complex 5-axis tool paths when precise tool angle control is needed.



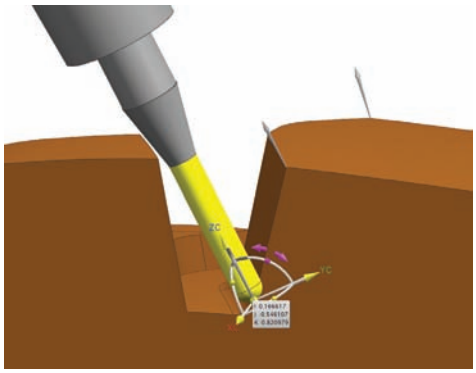
Simplify the generation of smart tool paths for blisk and impeller machining with specialized 5-axis operations for roughing, rest milling and finishing.

Automated feed rate optimization You can improve machining productivity and increase tool life by letting NX CAM automatically analyze tool loading and adjust feed rates to maintain a uniform material removal rate along tool paths.

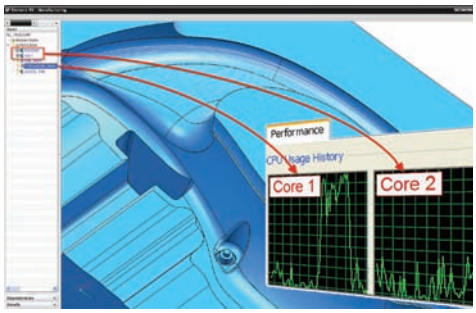
Enhanced machine tool simulation You can use the new tool path-based simulation to analyze and verify manufacturing operations earlier in the NC programming workflow by simulating machine tool motion without a postprocessor or machine tool driver.



Design and validate electrodes that can be used in multiple burn positions.



Rapidly create and adjust tool paths with a "virtual jog mode" interaction.



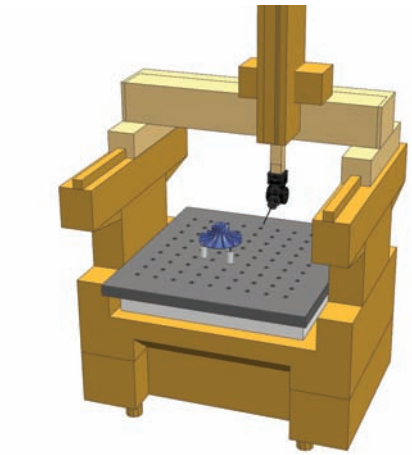
Speed up tool path processing by using your computer's multiple cores to calculate tool paths.

Faster tool path processing Don't wait for complex tool paths to generate before working on another operation. You can use your computer's multiple cores to speed up tool path processing. NX CAM generates operations in the background while you continue working. The result is a 50 percent reduction in tool path calculation time, depending on hardware.

Extended feature-based machining (FBM) capabilities You can achieve greater levels of NC programming automation by leveraging the extended feature-based machining capabilities in the areas of turning, wire EDM and 3- to 5-axis machining.

CAM postprocessing for greater machine efficiency NX 7 provides both automatic and user-selected options to ensure that the postprocessor for SINUMERIK controllers provides optimized output for high-speed cutting, 5-axis machining and cycle support. NX 7 also provides the option to output cutter paths defined by mathematical splines that match with the surface calculations in the Siemens SINUMERIK 840D controller, thereby producing better quality machined surfaces and faster processing times. This is an alternative to the standard method of outputting thousands of "go-to" points that have to be resolved back into a smooth cutter path by a controller.

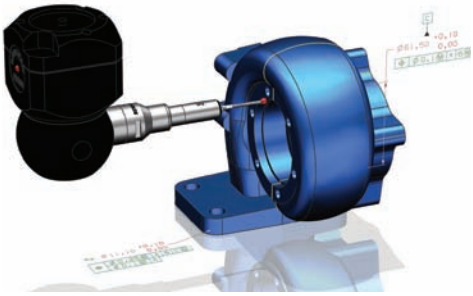
Enriched part manufacturing data management You can manage and distribute postprocessors via Teamcenter. In addition, you can store postprocessor-calculated machining time and machining/cutting tool utilization time, as well as manage multiple versions of CAM templates.



Maximize efficiency of offline CMM programming by working in the context of a 3D solid model environment that includes machine and part.

NX for CMM Inspection Programming

New application for CMM Inspection Programming NX CMM Inspection Programming is a new offline programming application that helps you program CMMs faster and more accurately by working in a 3D solid model environment. You can reduce inspection programming time as much as 80 percent by using the product and manufacturing information (PMI – including GD&T and 3D annotation) on the CAD model to automatically generate inspection programs. You can also ensure that inspection programs are collision-free by using 3D CMM simulation. The application can be used to generate postprocessed output including the DMIS standard and machine-specific formats. You can ramp up quickly by using the context-based inspection programming environment and online workflow tutorials.



Dramatically reduce CMM inspection programming time by using PMI on the CAD model to automate inspection feature and path creation.

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