What's new in NX 5

fact sheet

Summary

NX™ 5 software is a major milestone in the history of UGS' high-performance product development solution. The new release delivers key technology innovations that will help customers realize significant productivity and efficiency improvements.

Benefits

Greater workflow productivity throughout product development

Freedom from system-imposed constraints

Accelerated product development cycles

Better product quality and performance

Faster design-through-manufacturing turnaround

Improved collaboration

Features

Integrated, high-performance CAD, CAM, CAE and PDM for product development

Greater capability for all development applications: industrial design, design, simulation, documentation, tooling and machining

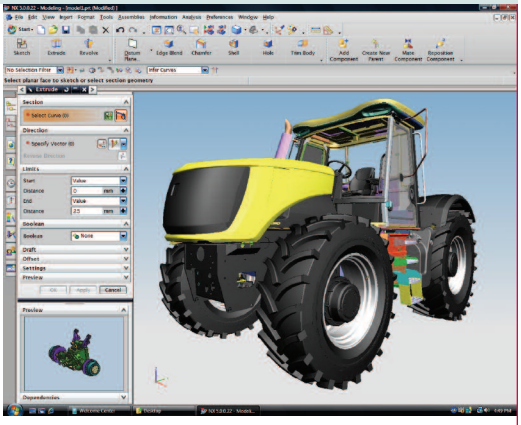
Productive, configurable, easy-to-use interface

Design Freedom technologies for modeling with imported geometry

Multi-CAD Active Mockup for in-context design, massive assembly performance

NX 5 system-wide innovations

Innovative user interface combines high-end capability with productivity, ease of use and learning. NX 5 introduces an innovative user interface design that boosts out-of-the-box productivity and dramatically improves usability. Redesigned menus and input dialogs clearly communicate required input and command steps, and are implemented consistently throughout the system, re-using common elements to enhance operational consistency and discoverability. Input dialogs now include explicit steps and focus on the current task, with commands presented in a top-down flow to help guide users through the operation. Similar functions are grouped together, with titles and commands reinforcing what each command does. Required steps are marked by red asterisks and completed steps are marked by green checks. Advanced options are collapsed and hidden, yet still available when users need them. To ensure the consistent location and presentation of all dialog boxes, users can attach or "clip" most dialog boxes to a "rail" along the upper edge of the graphics window to reduce clutter.



Consistent, structured input dialogs, configurable user interfaces, and Windows Vista contribute to a more productive, usable, and discoverable environment for NX.

With role-specific user interfaces, companies can present NX capabilities as appropriate to the user's function and level of expertise, enabling first-day productivity for novice users and flexibility to adapt to more functionality as their needs and proficiency grow. Developed and validated in partnership with NX customers and a third-party graphic design agency, *the new interface provides a comfortable, attractive and productive environment that will help customers realize immediate productivity gains of 20 percent or greater, and reduce training time by 50 percent.*

NX UGS

Transforming the process of innovation

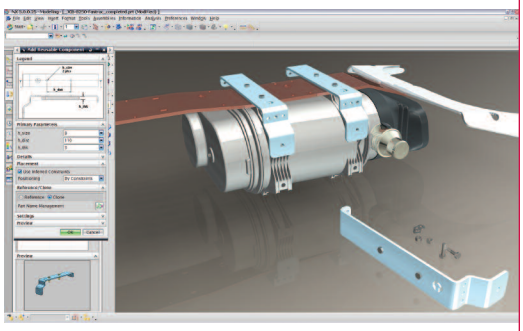
NX 5 engineering process management

NX product development solutions benefit throughout from the engineering data and process management capabilities of UGS' Teamcenter® software. With NX 5, the integration of NX and Teamcenter Engineering is extended, with new user interface utilities and functional capabilities that improve efficiency in searching, browsing, loading, creating, saving and synchronizing data.

Teamcenter management tools embedded in NX. NX 5 allows direct access to Teamcenter via a new Teamcenter navigator resource bar that will eliminate Teamcenter training requirements for most NX users. New file selection options when opening, adding, importing, exporting and substituting components provide greater usability and access to managed information in a style familiar to NX users. Teamcenter advanced search capabilities, with full filtering criteria and saved searches, are available directly from NX, enabling users to quickly locate Teamcenter-managed data. The assembly navigator in NX displays Teamcenter item numbers, revision levels, names and descriptions to provide better visibility and understanding of managed parts and assemblies.

Improved concurrent design. NX 5 will enable companies to reduce rework and improve collaboration by creating "interfaces" to a parts and publishing and managing interfaces in Teamcenter. These can include geometry and expressions defining connection points, locations and interfaces for related components. User access can then be restricted to a part's defined interface, and users have detailed information about dependencies. By publishing part interfaces, companies can simplify impact analysis, streamline change notifications and reduce change conflict in concurrent design scenarios by up to 60 percent.

Supporting re-use initiatives. NX 5 and Teamcenter help companies realize cost and time-to-market benefits of design re-use. A new Re-use Library navigation tool in NX displays standard parts and re-usable content in a hierarchical tree structure. It enables designers to directly browse Teamcenter classification hierarchies, Teamcenter folders and operating system directories for company standard parts, catalog parts and design parts.



NX 5 embeds Teamcenter engineering process management to support faster searching of managed data and to promote re-use. The integration also delivers dramatic efficiency gains for typical design tasks.

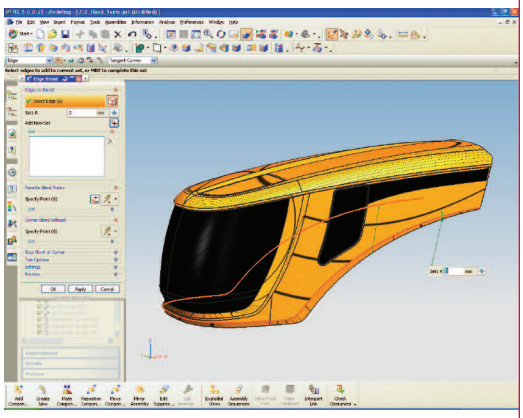
To further streamline the selection and placement of re-usable parts, NX has implemented a knowledge-enabled method that automates part family member selection and placement, based on indicated geometry. Users can simply drag part family icons to the target geometry in the model – the software infers the correct family member and automatically positions the part. *The technique reduces mouse clicks and design time in NX 5 by more than 80 percent over previous releases for typical design tasks.*

A library of industry standard parts has been added to NX that includes bolts, screws, nuts, washers, pins, bearings and profiles. The parts conform to all major international and national standards, including ANSI, DIN, UNI, JIS, GB and GOST.

NX 5 industrial design and styling

More freedom and control for styled geometry. For industrial design and styling, NX 5 includes many productivity enhancements to the curve and surface geometry toolset. Styled sweeps, blends and corners and global shaping tools have added new continuity controls, editing handles and input and selection options for more control and better geometry quality. Curve and surface modeling tools have been enhanced to support easier and faster creation and editing of freeform shapes. *With these improved styling tools, NX 5 offers 40 percent more styling options and completes designs 20 percent faster than a leading dedicated surface design tool.*

Faster reverse engineering from faceted geometry. NX 5 adds new tools for analyzing and creating surfaces from faceted bodies. Facet body curvature analysis calculates the minimum radius of curvature for each facet and creates a color map on the facet body distinguishing areas of high and low curvature. From this curvature evaluation, NX automatically extracts feature lines, or separates regions of different curvature. A rapid surfacing tool generates a curve network and a G1 continuous model of the faceted body.



NX 5 industrial design and styling delivers more freedom and control for curve and surface modeling, and streamlined surface analysis.

Faster, simpler geometry analysis. To help designers evaluate and optimize the functional and aesthetic quality of geometry, NX 5 introduces time-saving tools and workflows for surface analysis. New analysis tools include gap and flushness that analyzes values between two surface panels, compares them to nominal values and tolerances, and displays the results in user-customizable labels. The gap and flushness analysis measures the maximum opening width between two panels at a specified location, and evaluates how large gap and flushness will appear from a specified viewing direction. A new deviation gauge color map makes it easier to interpret deviations between surfaces or a surface and a facet body by color coding areas within, between and outside specified tolerances.

Designers can more easily organize, control and re-use geometry analysis tools, which now appear in the NX part navigator for faster grouping, display and editing. Customizable analysis templates make it easy to apply a combination of analysis tools to any number of features. Designers can create, edit and apply templates to save, re-use and share standard analysis objects.

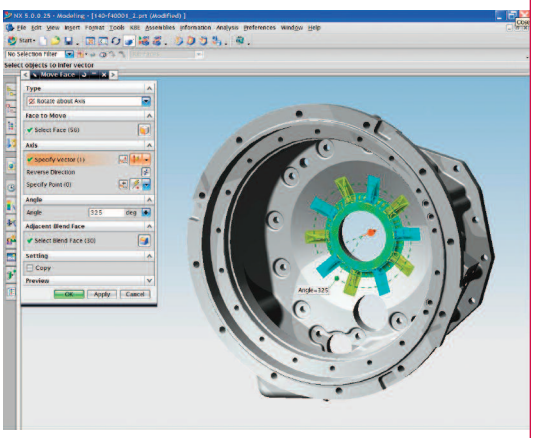
Faster visualization. To accelerate creation of high-quality images with special effects, NX 5 includes tools for creating, saving and re-using visualization parameters. Users can create "cameras" that capture the orientation, zoom and perspective of the current work view. Cameras are displayed in the part navigator or on screen for fast switching among view parameters. NX "scenes" capture background, reflections, lighting, stage setup and other parameters to automate the setup process for photorealistic rendering. Users can simply drag and drop the predefined scene onto a view to set all the visualization parameters, eliminating the manual settings typically required to visualize the model in different environments.

NX 5 design

Templates reduce manual input, support standards. NX 5 uses templates for creating new files. Users select templates from the appropriate group, such as part or drawing. The templates are customizable, and automatically generate default file names, locations and other settings as defined in company standards. In addition, the templates automatically start the appropriate application.

Better design productivity and clarity. Many enhancements within NX are aimed at improving the efficiency of interactive design. Designers can more efficiently control the display with a consolidated dialog that shows or hides objects by type. A view triad persistently displays the orientation of the model's coordinate system and can be used to dynamically rotate models about a coordinate system axis. Models can also be rotated about part edges, curves or datum axes. A new command finder assists in menu navigation. The part navigator, which provides a tree list of features and bodies that comprise a part, has been reorganized to reduce redundancies and improve clarity. Improved error reporting and diagnostics increase the effective skill level of users and supports faster problem resolution.

Design freedom with advanced selection intent. NX 5 consolidates all geometry selection options in one convenient toolbar for enhanced control in interactive design. The new options enable users to select and group multiple faces, features, curves, edges and components, improving selection speed, clarity and accuracy. Advanced selection intent works in conjunction with NX direct modeling extensions on models imported from any source, delivering the most productive solution for working with legacy data. It delivers huge time and productivity benefits as compared to traditional feature recognition technology.



Advanced selection intent controls and direct modeling in NX 5 give designers freedom to work directly and efficiently with models from any source.

Faster model editing. With the new release, NX offers users the option of saving model data for fast rollback and editing. *The option dramatically reduces model recompute time when making changes, resulting in 30 percent to 50 percent faster editing.*

More productive sketching for assembly layout, feature creation. The NX sketcher has been enhanced with additional background color, work plane and grid options that support a wider array of user preferences for 2D sketching. New commands have been added for making corners, offsetting curves, creating intersection curves and fast trimming and extension of curves. The expressions dialog has been added to the sketcher for capturing complex design intent. Sketch dimensions and associated expressions can be reattached to new target geometry, eliminating the need to delete and recreate the dimension. A new constraint command locks sketch curves in a fully defined position and orientation.

Faster creation of patterned geometry. For designs with multiple instances of geometry, NX 5 adds a flexible, consistent patterning mechanism that supports bodies, curves, curve features, edge and face sets and datums. Designers can create multiple instances of the geometry in associative patterns that are transformed along a line or about a rotation, mirrored, along a curve or in irregular patterns. Instances are associatively linked to the original geometry in this powerful tool for design re-use.

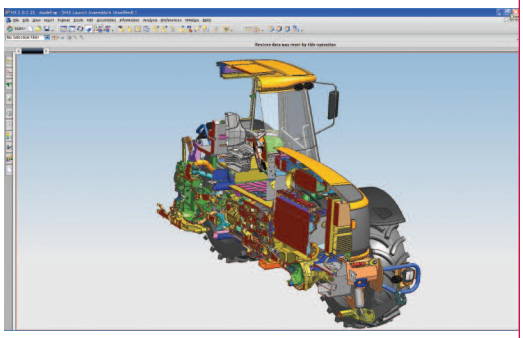
NX 5 assembly design

Performance breakthrough for large assemblies. NX 5 introduces major architectural changes that dramatically improve design functions and performance for large assemblies. NX assembly modeling now employs UGS' widely adopted JT data format and Direct Modeling technology for lightweight, high-performance visualization and multi-CAD collaboration. The use of JT dramatically improves assembly design functions for faceted representations, enabling key operations without requiring the use of precise solid geometry. In addition, the JT format improves NX's large assembly capacity and performance while reducing memory usage and rendering time. *Test cases demonstrate 60 percent or more reduction in memory usage and 65 percent or more increase in display frame rates for typical large assemblies.*

Active Mockup unites mockup, review and modification processes. NX 5 embeds digital mockup capabilities in the assembly design environment, introducing technologies from UGS' Teamcenter Visualization that support multi-CAD assembly layout and design in context. Enabling an instantaneous review-to-modification process, NX Active Mockup unites the design and review processes in a single solution that requires no separate files, eliminating file management overhead typically required with separate design and mockup solutions.

With lightweight loading and direct model rendering, designers can actively work in the full assembly context, with multiple level-of-detail display that supports interactive performance even in huge assemblies. New assembly engineering capabilities in NX 5 include:

- Dynamic clearance analysis
- Spatial filtering
- Faster dynamic sectioning
- Directionless assembly constraints
- Automatic generation of motion envelopes
- Ability to add multiple components to assemblies in a single command
- Automatic collision-free path planning for assembly/disassembly



Active Mockup in NX 5 supports true design in context, even for massive, multi-CAD assembly models. It dramatically improves large assembly performance and unites the review and redesign processes with a single solution. This image depicts a dynamic cross-section of the tractor assembly model.

NX 5 routing

NX 5 improves efficiency and control in design of mechanical routed systems through implementation of the D-Cubed 3D constraint solver. Designers can define more sophisticated relationships between routing segments and components using assembly constraints for position, alignment, angles and rotation vectors. With the enhanced functions, designers can directly edit values of dimensioned constraints, and can better control geometry and components for more efficient editing.

For electrical routing, NX 5 automates design of terminal block connections with new tools that create multiple paths from the end of a path to a set of ports on a terminal block. Journaling has also been implemented throughout the routing applications to capture and automate frequently used routing functions. Designers can easily record journal files, which can be paused, replayed, stopped and edited.

NX 5 design validation

More efficient, automated validation. NX validation checks can be configured in NX 5 to execute automatically whenever a design file is saved, ensuring the consistency of validation and earlier detection of problems. For faster interpretation of results, measurement checks are continuously displayed during assembly sequencing and motion analysis using color-coded backgrounds as visual feedback of the check status.

Supported validation checks have been extended to include double-sided inequalities and lists of approved values, as well as local minimum, maximum and various projected distances, between any combination of geometric entities or components or assemblies. Mass properties checks have been extended to include volume, surface area, radius of gyration and weight. For faster resolution of problems, NX now automatically navigates to part features involved in checking violations.

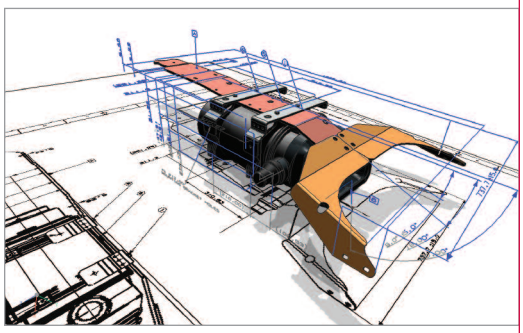
Requirements-driven design validation. NX Check-Mate has been enhanced to validate designs against internal and/or external numerical requirements. Requirements sources for validation checks include XML files, native or managed Excel files and Teamcenter Requirements. Requirements checks can be created more conveniently from the expressions dialog, within measurement commands or from DesignLogic parameter entry widgets.

NX 5 sheet metal design

With NX 5, sheet metal applications include a materials database that enables designers to automatically apply material thickness, bend radius, neutral factor and other characteristics upon selection of the material. Bend tables and bend allowance formulas help companies conform to standards to drive the bending parameters of sheet metal parts. NX 5 adds a new modeling tool that significantly reduces the input required to create commonly used hems, rounded corner flanges and solid punch features. To accelerate design changes, the software now enables direct editing of bend region radius, thickness and k-factor, which also assists in simulating intermediate forming states. For sheet metal part drawings, NX has implemented a streamlined technique for creating drawings with both flat and formed views, with special annotations for bend region parameters. To simplify manufacturing, NX sheet metal exports flat pattern curves and bend region annotations for direct input to machine tools.

NX 5 drafting and 3D annotation

Rapid drawing creation tools. Innovations in NX 5 for drafting and 3D annotation significantly reduce the time and effort required to produce 2D drawings and communicate the design and manufacturing intent in the 3D model. New templates for master model-based and non-master model-based drawings accelerate the drawing creation process with automated title block functionality and preconfigured standards-compliant environments. NX 5 provides a set of predefined standards options that enable customers to quickly configure drafting defaults and preferences in accordance with ISO, ASME, JIS or DIN standards. In addition, with minimal interaction the options can be easily and rapidly customized to support company, industry or country-specific preferences. NX 5 drafting now fully supports shaded views, and provides a number of usability enhancements and a more intuitive interaction environment for the creation of drawing views and annotations.



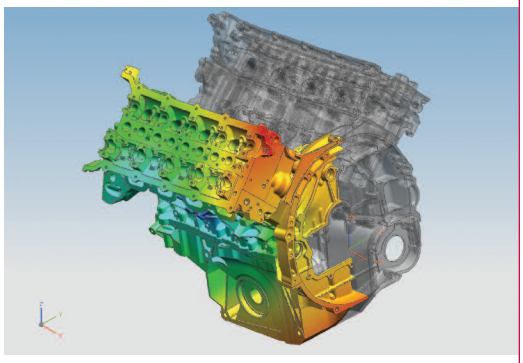
NX 5 dramatically reduces the time and effort required to create drawings and communicate design and manufacturing intent in the 3D model.

More flexible PMI environment. NX 5 enables customers to create 3D annotations (a.k.a. product and manufacturing information, or PMI) for capturing complete design intent, enabling information re-use and further automating the drawing creation process. New inheritance methods allow users to specify which PMI annotations to be inherited on drawing views, thus reducing or eliminating the need to create 2D annotations for drawings. Using the new search and highlighting tools, designers and draftsmen can more quickly and easily locate PMI related to model topology and features. A new stacking mechanism, new display methods and global dragging functionality wall contribute to a more intuitive environment for the creation of PMI.

NX 5 digital simulation

Powering innovation with Lifecycle Simulation

NX 5 delivers on UGS' pervasive *Lifecycle Simulation* vision. A unified, production-ready environment, NX 5 includes a complete suite of multi-disciplinary simulation solutions that meet the needs of advanced CAE analysis experts as well as designers and design engineers.



UGS Lifecycle Simulation solutions delivered with NX 5 can analyze the world's largest CAE models. The image depicts post-processed displacement results displayed in dynamic cross-section on an engine block model.

NX Advanced FEM/NX Advanced Simulation

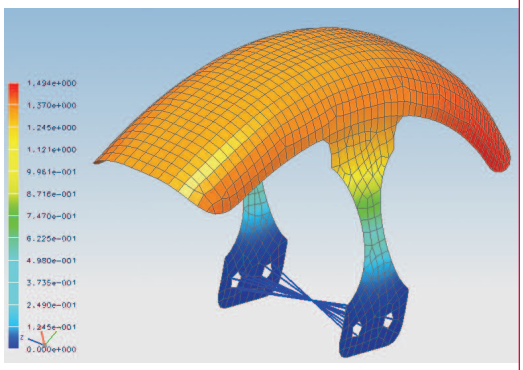
NX 5 delivers a significant number of finite element modeling and results post-processing enhancements to enable a complete environment for advanced multi-physics performance simulation. Building upon robust "geometry-centric" FE modeling capabilities, NX 5 rounds out a complete CAE modeling environment with the "bottom-up, define your own geometry" FE-centric modeling capabilities widely used by dedicated CAE experts. These new CAE modeling and results post-processing capabilities include:

- *Manual node/element operators*
  - Complete node and element manual manipulations
  - Element quality fix functionality
- *Surface meshing extensions*
  - Support for 2D mapped meshing and 3D sweep
  - Smart boundary meshing
- *FE model append*
  - Allow the import of external FE models from multiple sources into an existing combined FE model
- *Mesh mating conditions*
  - Glue coincident – non-manifold surfaces
- *Loads and boundary condition extensions*
  - Flow, thermal, advanced nonlinear, dynamics
  - Automatic face pair detection-contact surfaces
  - Rotational coupling for cyclic symmetry
- *Material properties definition extensions*
  - Advanced nonlinear material types for NX Nastran
- *Physical property tables*
  - UI based on user selected FE solver language

- *FE sets* for CAE expert users
  - Traditional node and element grouping capabilities
- *Mesh collectors* assign attributes to meshes
  - Physical properties table
    - Material properties table
  - Display mesh attributes
  - Drag and drop meshes between collectors
- *General FE modeling tools*
  - Enhanced element quality checks
  - Coordinate systems extensions (cylindrical, spherical)
- *Additional pre/post support for NX Nastran solution types*
  - Dynamics analysis extensions
  - Advanced Nonlinear extensions
- *Integrated CAE results post-processing and visualization*
  - CAE results are a node in the NX Simulation navigator
  - Integrated X-Y graphing support with post displays
  - Display and animate complex dynamics results
  - Synchronous motion animation and tracking on X-Y graphs

What's new in NX 5 for digital simulation

NX Response Simulation – a new add-on module to NX Advanced FEM and Nastran that provides an interactive, visual environment for the evaluation of the structural dynamic response of a system when subjected to various types of complex shock and vibration loading conditions.



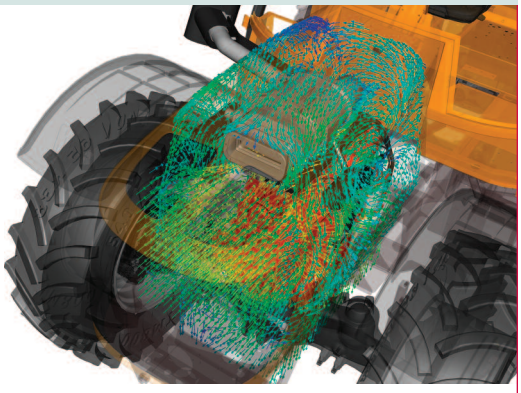
NX Response Simulation analyzes dynamic response from shock and vibration.

NX Advanced Flow – a new add-on module to NX Flow that extends the range of computational fluid dynamics (CFD) simulation capabilities available within the NX Advanced Simulation environment including:

- Highly compressible flows
- Additional turbulence models, Rotational Frame of Reference (RFR)
- Nonlinear fluid properties (visco-plastic models, etc.)
- Transport of species – humidity, condensation

NX Advanced Thermal – new add-on module to NX Thermal that extends the range of thermal simulation capabilities available within the NX Advanced Simulation environment including:

- Advanced radiation (ray tracing, refraction, infrared signature, etc.)
- Articulating, spinning assemblies
- Duct and hydraulic flow
- Electrical (Joule) heating
- Material transformations – phase change, thermal ablation, charring



NX Advanced Flow Analysis is a new simulation tool in the NX 5 release.

NX Electronic Systems Cooling – an integrated vertical solution combining robust FE modeling with NX Flow and NX Thermal capabilities as well as special capabilities including heat sinks, fan catalogs, PCB data exchange and PCB Modeler (ECAD/MCAD integration); enables the evaluation of the cooling effects of airflow around enclosed, densely packed heat-generating electronics systems used in consumer products, automotive and other industries.

NX Space Systems Thermal – an integrated vertical solution combining robust FE modeling with NX Advanced Thermal capabilities as well as special features for orbital environment modeling and radiation primitives. This new package enables the evaluation of transient heat transfer characteristics for spacecraft and space systems during both orbital and inter-planetary missions.

NX Laminate Composites – an add-on module to NX Advanced FEM that enables the evaluation of products made up of laminate composite materials. The new laminate modeling property definition entity includes support for:

- New material types and ply lay-up definitions
- Failure envelopes, failure index, margin of safety, worst ply results

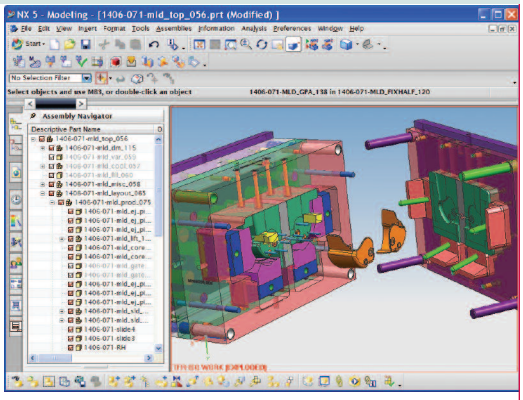
NX Nastran – many NX 5 enhancements across all major areas such as:

- Ease of use/productivity: bolt pre-load, glue connections
- Linear contact and advanced nonlinear extensions
- Structural dynamics, rotor dynamics and DDAM extensions
- Numerical solution efficiency for large model performance
- Flexible body motion analysis with ADAMS or Recurdyn solver
- Optimization and advanced nonlinear extensions

NX tooling

Mold design

Mold design and Teamcenter integration. NX 5 continues to enhance the integration between NX Mold Design and Teamcenter for companies aiming to manage data and integrate processes between tool design, manufacturing and the shop floor. NX 5 has been enhanced to manage 2D drawings associated to standard part data.



NX 5 Mold Design delivers templates for mold design configurations and new mold validation tools.

Design re-use. NX 5 Mold Design offers a new approach for capturing and re-using standard mold design configurations. This new capability allows designers to preconfigure a mold design project template with any number of components (sliders, ejectors, runners/gates, cooling, etc). The template can then be applied at the beginning of the design and the content adjusted as the mold design process progresses. *In many situations, this re-use approach can help save design time by as much as 50 percent and also help to reduce costly design errors*.

Molded part validation. NX 5 adds a corner radius check to the already available wall thickness and draft angle analyses. The corner radius check helps to identify insufficient radii, edges with no blends and also automatically colors radii of specified values/ranges for visual inspection.

Shut-offs and parting. NX 5 continues to streamline the shut-off design workflow with solid patch target control and also improves the parting design change workflow with the ability to store and save existing parting surfaces.

Core and cavity. NX 5 introduces the ability to specifically check the core/cavity design to related molding components for interferences. This new check also maps manufacturing colors and attributes from the article part to specified molding components for manufacturing automation.

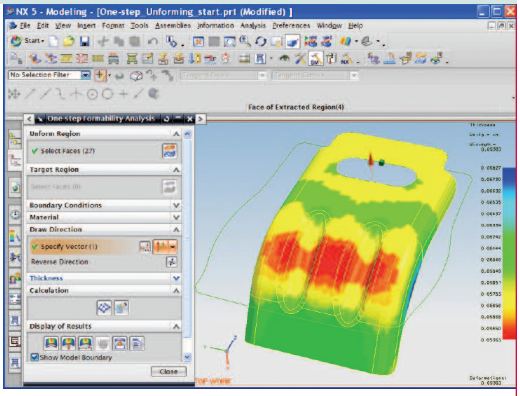
Tool design validation. It is important to validate the overall tool design for proper clearances and interferences early in the design cycle; and, NX 5 enables mold/die designers to quickly perform pre-defined checks.

Design change efficiency. NX 5 offers an extensive toolset for controlling propagation and validating design changes between related components within a mold design project and also into manufacturing. New designs can be compared to originals, swapped and then mapped to enable robust downstream feature updates. Additionally, WAVE Update Control is introduced which is useful for identifying components affected by design changes based on geometric differences. These enhancements are projected to contribute between 15 percent and 30 percent time-savings when making various design changes to mold projects.

Progressive die design

Progressive die design and Teamcenter integration. NX 5 enhances the integration between NX Progressive Die Design and Teamcenter for companies targeting to manage data and integrate processes between tool design, manufacturing and the shop floor. NX 5 supports the ability to manage standard part data with associated drawings.

Blank creation and pre-bend definition. The workflows for creating the blank and pre-bends for both straight break and freeform sheet metal parts have been enhanced with NX 5. Creating pre-bends and over-bends is now easier and takes less time due to the ability to handle multiple step bends and independently control the k-factor and BAF for individual bends. One-step Formability Analysis has also been introduced which adds the ability to analyze and report stress, strain, thinning and springback. *The blank and intermediate state creation enhancements are projected to contribute between 20 percent and 30 percent time-savings when preparing parts for progressive die projects.*



NX 5 tooling includes one-step formability analysis that reports stress, strain, thinning and springback for formed parts

Scrap design and strip layout. In NX 5, individual pieces of scrap can be quickly sketched using the NX sketcher and then automatically recognized for placement within the Strip Layout.

Die base and inserts. A new set of bending punches/dies is available with NX 5 that incorporate parameters for springback compensation. These new punches/dies are enabled with the same placement and sizing automation as the standard NX inserts.

Tool design validation. It is important to validate designs for proper clearances and interferences early in the design cycle and NX 5 enables die designers to quickly perform predefined checks.

Design change efficiency. NX 5 includes an enhanced set of tools for managing design changes within a progressive die project. The new Swap capability enables the ability to replace the original sheet metal part design input with a different component while Design Change Guidance offers advice for effectively completing the change process. Lastly, the hole table is now associative and can be easily updated to a new revision with the press of a button.

Electrode design

Pallet integration. To assist with proper EDM machine setup and validation, NX 5 has been upgraded to support pallets for electrode manufacturing procedures. A library of standard pallets can be easily configured and applied to any electrode project.

Design efficiency. With NX 5, electrodes are automatically tagged with attributes specifying sparking area and sparking type based on a configurable spreadsheet. These attributes can then be used to automate machining of electrodes with compensation for spark gap and orbital motion. Additionally, electrode drawing creation has been improved with additional placement automation capabilities.

Design change efficiency. In many cases, multitudes of electrodes are required to manufacture complex work pieces; as a result, identifying which electrodes are affected during design change sequences can be tedious and time-consuming. NX 5 has been enhanced with WAVE Update Control which automates and assists with the identification and update of only the electrodes that are affected by design changes. These enhancements are projected to contribute between 20 percent and 30 percent time-savings when making design changes to mold projects and then propagating the changes to and updating the associated electrodes.

NX 5 CAM

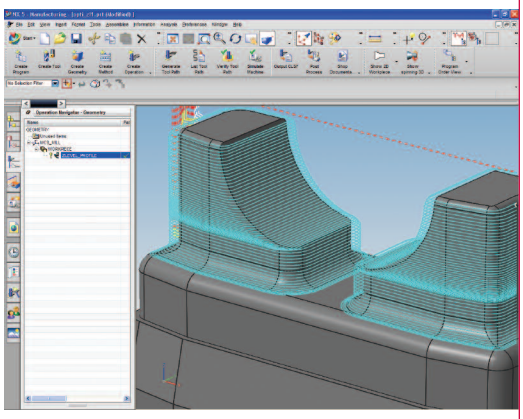
User experience. NX 5 continues to deliver an outstanding user experience. By continuing to drive consistency and efficiency into the interface, each release of NX CAM makes its power and flexibility even more accessible. NX 5 delivers significant satisfaction across the UI, with particular emphasis on all types of non-cutting moves.

High speed machining

Streamline tool path. NX 5 introduces a new tool path type to the CAM suite. The Streamline tool path is particularly well suited to high speed finishing because it matches the cutting pattern to the specific part geometry of interest. The NX CAM approach to streamline cutting patterns is groundbreaking in its flexibility and utility:

- Any surfaces, trimmed or untrimmed, rectangles not required
- Dual contact boundary condition
- Spiral patterns
- Patterns including points
- Includes 5-axis cutting patterns

Z-level finishing with tool axis tilt. NX 5 CAM introduces tool axis tilt to its Z-level finishing tool path. Tilt can be specified as simple "away from part" or specifically constrained by a point or curve. Tilting the tool allows shorter tools to reach the bottom of the cavity. *Because tool flexibility increases with the square of length, tool axis tilt can result in large feedrate increases. Reducing a tool's length by 20 percent can increase its rigidity, and correspondingly its feed rate, by 50 percent.*



NX 5 machining optimizes Z-level cuts for the part curvature to automate cut level specification and reduce machining time.

Optimized Z-level cut depths. NX 5 continues to make cut level specification more automatic and more optimized for high speed machining. Part curvature determines whether smaller cut steps are required to obtain the specified finish. *Compared to reducing the overall step size, this approach can reduce cut time by 50 percent.*

Multi-function machining

3D machining environment. NX 5 CAM further emphasizes the 3D nature of the programming environment, featuring better support for slant beds and B-head+C-chuck mill-turn configurations. This 3D programming environment provides for better tool control, collision checking and visualization of the program performance on these complex, multi-function machines.

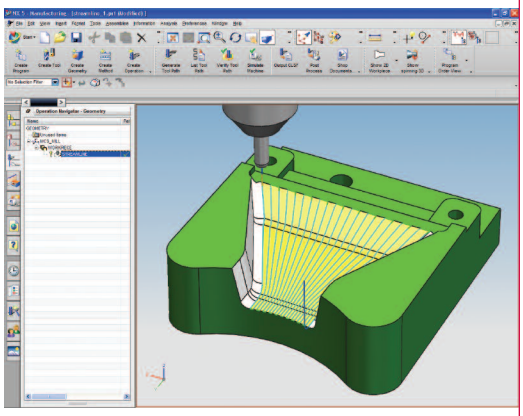
Multi-function machine simulation. NX 5 continues to provide the best synchronization and simulation of these important machine types. Improvements in Post Builder, Synchronization Editor and driver technology keep NX CAM in a leadership position.

Machine tool kits. NX 5 adds to the available machine tool kits, which provide a ready-to-use post processor, machine tool model and simulation driver. Supported machines include the Mazak Integrex and Mori NT machines. *Installing a full kit like this can slash the time it takes to start delivering working programs to a new machine.*

5-axis machining

Z-level finishing with tool axis tilt. The introduction of this function to NX 5 CAM emphasizes the application of 5-axis technology to the tooling industry and other industries that have not traditionally employed 5-axis methods. Adding a tilt capability to the planar finishing pattern deliberately blurs the lines between 3-axis and 5-axis methods. *Each year more 5-axis machines are deployed into the tooling industry. Projections indicate that 5-axis technology will be the driver for the next level of productivity in this market.*

Streamline tool path. The new Streamline pattern in NX 5 is particularly powerful as a 5-axis application. In the 5-axis context, the streamline functions as a very flexible drive surface specification. The Streamline tool path adjusts both the cutting pattern and the tool axis with the same smooth transition from one end of the cut region to the other.



A new Streamline tool path in NX 5 machining automatically adjusts the cutting pattern and tool axis for smooth transition from one end of the cut to the other.

Generic motion specification. NX 5 introduces a completely flexible motion definition that is suitable for all kinds of non-cutting tasks that are required from your machines. This includes transition motions from one operation to the next, moving around a complex part in 5-axis. This is also the way to specify probing motion, or other non-cutting chores that you ask of your machine tool. Being able to simulate these generic motions is critical to staying confident in the safety and efficiency of these various motions. *Using these tools prevents hand-editing and other improvised approaches that can lead to costly errors. One prevented crash can pay for a seat of CAM software!*

NC programming automation

Machining feature navigator. NX 5 CAM introduces the Machining Feature Navigator as the central manifest for all machining features. This new organizational tool makes it even easier to identify and track the progress of each feature. Customizable views of the features and their parameters make it faster than ever to apply the best processes to the feature set.

Feature-based machining. NX 5 CAM introduces, as a preview, the ability to define holemaking processes using Excel. This method further simplifies the description, management and implementation of rules for feature machining. Applying feature-based technology can save as much as 90 percent of programming times for the features implemented. Easier implementation brings this advantage to more features more quickly.

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