

What's new and cool in NX 6

fact sheet

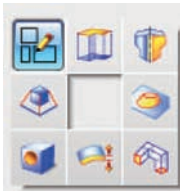
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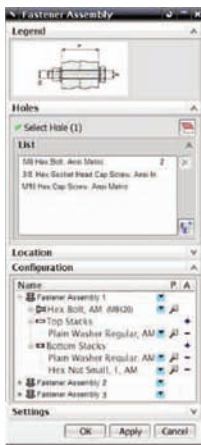
► Summary

Siemens PLM Software's NX™ 6 solution suite delivers significant productivity gains for our customers.

The launch of Synchronous Technology in April 2008 at Hanover Fair (Germany) marks a major milestone in Siemens' commitment to the MCAD market. Using Synchronous Technology as a base, NX 6 provides breakthrough capabilities that significantly benefit today's product makers. NX 6 digital simulation solutions deliver state-of-the-market capabilities that will enable Siemens customers to more effectively compete in today's global marketplace. The release of NX 6 also includes major enhancements across the entire NX solution suite. This fact sheet introduces these NX innovations by describing each NX functional discipline and its newly released capabilities.



User customizable radial pop-ups enable gesture-controlled actions.



Knowledge-enabled fastener sets streamline the selection and insertion of nuts, bolts, washers and related geometry.

NX 6 system-wide innovations

Innovative user interface combines high-end capability with ease of use and learning. NX 6 builds on the role-based user interface introduced in NX 5. Expanding the coverage of the block-based approach across all applications ensures consistency in the core product areas.

To deliver a system that can grow as a user's skill level increases, NX 6 features customizable radial toolbars that reduce mouse travel and enable users to bring their common functions into simple gesture-controlled actions.

Maximum use of screen real estate increases focus on design task. NX 6 allows users to make maximum use of their graphics windows. With full-screen mode the NX user interface and navigators are minimized to allow users to concentrate on the task at hand.

Real-time rendering. NX 6 introduces Trueshade, providing all users with high-quality dynamic visualization of their designs with reflections and environment maps. Trueshade allows users to quickly visualize their designs in a variety of environments and materials.

NX 6 engineering process management

NX product development solutions increase throughput by leveraging Teamcenter® software's engineering data and process management capabilities. With NX 6, the integration of NX and Teamcenter is further extended. NX 6 brings the ability to see more data from the Teamcenter product structure editor inside of NX, which provides a more complete presentation of structure and related data to the user.

Teamcenter project support. NX 6 allows users to assign project data (either single or multiple projects) at file creation or save time. The Teamcenter navigator has been extended to allow users to assign Projects to multiple Items at once. The Teamcenter navigator can be filtered to display objects based upon Projects allowing users clear understanding of the context of the overall design.

4-tier client support. NX 6 has been enhanced to take advantage of the performance gains delivered by Teamcenter's 4-tier architecture. This is particularly important to users working on large distributed networks.



Visual shape searching of Geolus databases from inside NX enables effective re-use.

Supporting re-use initiatives. The re-use library navigation tool in NX has been extended to support Process Studio Wizards from simulation. In addition to this the re-use library now allows features and copied geometry to be quickly stored for future use.

To further streamline the selection and placement of fasteners (nuts, bolts, screws, washers), NX has implemented a knowledge-enabled method that automates the workflow required to select and insert standard fastener sets. The system automatically selects individual component parts based on clamping length and diameter along with compatible mounting features and inserts together with the fastener assembly. This significantly increases productivity by enabling users to create bolted connections within an assembly and standardize on fastener usage within the company – as well as select the right component (by mounting feature pairs).

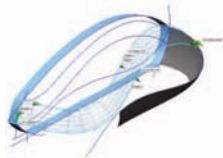
Geolus integration. To facilitate more effective ad hoc shape searching, NX 6 provides embedded access to search any Geolus® software databases that may be present. This searching can be based on geometry characteristics such as size and shape, or the user can create some core base geometry that forms the basis for the shape search function. Geolus search results can include NX parts, JT data or data from other systems such as SolidWorks and CATIA.

When a matching design is found, it can be brought into the NX 6 session and investigated for design suitability. SolidWorks data can now be opened directly via the integrated SolidWorks translator.

The integration of Geolus removes the need to exit NX to perform shape searching and encourages the re-use of component geometry, significantly reducing design time and creation of duplicate components.

NX 6 industrial design and styling

Immediate feedback when creating freeform shapes. For industrial design and styling, NX 6 includes many productivity enhancements to the curve and surface geometry creation toolset. Having immediate feedback improves productivity when designing complex shapes. *With these improved styling tools, NX 6 offers 40 percent more styling options and completes designs 20 percent faster than a leading dedicated surface design tool.*



Improved real-time feedback accelerates design and minimizes rework.

Faster reverse engineering from faceted geometry. NX 6 makes facet work easier, enabling users to employ millions of polygons (20-30 million) found in some of the most demanding applications by using JT for the underlying facet body architecture. This effectively removes limitations of data size and enables the reverse engineering of a complete product. The performance gains increase speed and efficiency of features and workflows by 50 percent.

User interface tailored for the designer. As described above NX 6 full-screen mode with its customizable user interface significantly enhances the design experience, especially for industrial design and styling. Benchmarks have shown that the new full-screen look appeals to designers who are currently using niche tools. This makes NX an acceptable choice for industrial design not just in terms of functionality but in terms of user interaction and appeal.



Facet body hole filling enables fast data prep by snipping out bad areas and filling.

Faster, simpler geometry analysis. To help designers evaluate and optimize the functional and aesthetic quality of geometry, NX 6 introduces time-saving tools and workflows for surface analysis. The deviation gauge color map has significantly improved performance and, by supporting for trimmed surfaces, the map provides better “real-time” feedback during the styling and reverse engineering workflow and delivers up to 75 percent performance improvement.



Real-time reflections of design improve realism.

Faster visualization. Coupled with the power of Trueshade, NX 6 also includes a number of powerful rendering tools that speed the generation of realistic product images. The ability to reflect the object being designed in the environment background allows higher levels of realism when performing real-time reviews of designs.

The addition of decals to components has been significantly enhanced, speeding the process of adding customer information and relevant graphical detail to a design.

NX Design

Design freedom enabled by Synchronous Technology. NX 6 expands the system's Design Freedom capabilities with powerful new approaches for product development. NX 6 Design Freedom provides new dynamic editing and creation capabilities that make it easier for non-designers (such as simulation engineers and manufacturing engineers) to use the system's CAD functionality. It also enables data to be used from any source.

Immediate feedback on geometry editing. NX 6 Design Freedom gives real-time feedback on geometry edits; with a simplified user interface the speed of editing geometry to build products is significantly enhanced.

Active selection adds intelligence to designs when needed. NX 6 Design Freedom features new advanced selection methods which automatically imply logical and feature relationships even on non-native data. This action preserves conditions such as tangency, concentricity, symmetry and builds relationships such as common hole sizes at time of editing rather than at creation time.

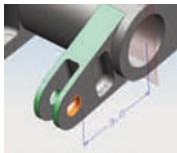
Persistent dimension-driven edits. NX extends the dynamic editing of Design Freedom to allow dimension-driven edits. Dimensions can be added to the 3D model at any stage and used to drive the design to specific values or to build parametric relationships on non-native data.

Scrapbook modeling. To increase geometry re-use, NX 6 supports a new workflow that allows users to cut/copy and paste sections of a model. This can be inside a single file, or across multiple files.

Cut and copy geometry preserves design intent, allowing the geometry to adapt to the new model conditions when it is pasted into place. The system automatically trims the geometry to the new model and reapplies any blend geometry as necessary. This approach significantly reduces design time and promotes geometry re-use.

History-free mode. For concept creation and fast design workflows, NX 6 allows users to opt to work in "synchronous" or "history-free" mode. Geometry is created using standard NX approaches; however, no linear history tree is created, removing the overhead of re-computing the history tree when making model edits.

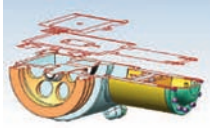
Replacement assistant. Many design modifications require complete changes to geometry. NX 6 features a new replacement assistant tool that simplifies major geometry changes. Previously users would need a detailed understanding of the relationships between parametric objects; the replacement assistant helps speed the workflow when making complex changes.



NX allows 3D dimension-driven edits to add parametric expressions when needed.



NX Scrapbook modeling enables new parts to be quickly created from old ones.



Faster dynamic assembly sectioning.

NX 6 Assembly Design

Productivity gains when dealing with NX assemblies. NX 6 introduces numerous assembly modeling enhancements that improve core task productivity. For example, on-the-fly linking cuts many steps out of the assembly design workflow, building WAVE geometric links when needed, even when the user is using lightweight JT-based facet data.

Dynamic Sectioning improves assembly review process. Dynamic sectioning has been enhanced in NX 6 enabling users to define and display multiple sections controlled via the model navigation tool. Users also can request a separate 2D pop-up window with a grid display to review geometry (e.g., measurements) and clearly view curve geometry.

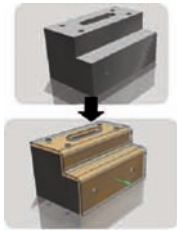
NX 6 Routing

NX 6 enhances its electrical routed system design capabilities by adding physical properties to the system's smart routing tools. These tools facilitate intelligent path routing by enabling the paths to understand the overall assembly's constraints, including its keep-in and keep-out zones.

NX 6 Design Validation

More efficient, automated validation. NX can now be set to run Check-Mate validation checks against parts that have actually changed. This significantly reduces the time needed to validate designs.

Requirements-driven design validation (RDDV). NX 6 RDDV can access Teamcenter's system engineering capabilities to facilitate requirements capture for subsequent validation. This approach enables users to set rules within NX that are dynamically validated.



NX 6 allows complex sheet metal parts to be created by wrapping around a solid.

NX 6 Sheet Metal Design

With NX 6 complex sheet metal components can be quickly generated by developing the solid that is to be encompassed, effectively allowing the user to develop a model of the "void" and then generate an enclosure that surrounds it.

This approach is particularly useful when designing sheet metal that has complex angles and unusual bend conditions. This capability supports an important I-deas® software workflow, which has been ported to NX to maintain Siemens' commitment to the I-deas user community.

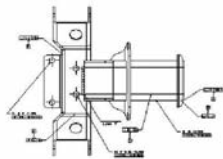
NX 6 Drafting and 3D Annotation

Rapid drawing creation tools. Innovations in NX 6 for drafting and 3D annotation significantly reduce the time and effort required to produce standard compliant 2D drawings and communicate the design and manufacturing intent in the 3D model.

Unified sketch environment. This environment allows users to create 2D sketches inside the drafting system without leaving the NX environment.

Improved dynamic interaction model. NX 6 speeds the process of getting the drawing that the user needs by increasing the use of dynamic interaction for annotating models including dynamic manipulation for centerlines, leaders, notes and labels.

Automated hole tables. This capability enables users to create standard hole tables to document a model, streamlining the documentation process.



PMI re-used in drawing view.

NX 6 Product Template Studio

Maximize the value of parametric design. NX Product Template Studio (PTS) supports re-use strategies by allowing any parametric part to become a “template” with published inputs and embedded validation tools. PTS builds a codeless user interface that ensures that the correct input for a design is collected and applied to the model enabling design variants to be quickly developed.

PTS templates can be combined together to support the generation of parametric systems models that can rapidly be changed as the basis for new generation products.

NX 6 Digital Simulation

Powering innovation with lifecycle simulation

With over 350 user-driven enhancements encompassing over 10 different CAE application domains, NX 6 continues to deliver on Siemens’ pervasive lifecycle simulation vision. A unified, production-ready environment, NX 6 includes a complete suite of robust multi-disciplinary simulation solutions that meet the needs of advanced CAE analysis experts as well as designers and design engineers.

Multi-CAD Simulation Modeling

NX 6 Advanced Simulation directly leverages the new Design Freedom capabilities, powered by Synchronous Technology to enable the rapid evaluation of design modifications based on geometry from any CAD system.

Customized CAE environments structure the NX user interface and FE modeling options in the specific language of the FE solver being used (e.g., Nastran, Ansys, Abaqus, LS-DYNA).

Unique 3D geometry editing, automated abstraction and FE meshing tools make NX the fastest and most productive multi-CAD simulation modeling and analysis environment available today. “Mid-surface” modeling extensions based on a non-manifold geometry engine enable more rapid abstraction and modeling of plate and thin-walled structures.

Additional 1D and 2D FE modeling features, as well as transition elements and multi-point constraints, complement the existing world-class 3D modeling capabilities to provide a complete suite of “bottom-up” modeling and meshing tools for the most complex multi-disciplinary performance simulations in space systems, aerospace/defense, industrial equipment, consumer products and automotive/vehicle applications.

Laminate Composites modeling

NX Laminate Composites features an extensive array of capabilities including ply-based modeling, draping and flattening as well as ply optimization. CAE workflow is aligned to support design and manufacturing practices for composite materials. The product supports the use of NX Nastran as well as the MSC Nastran, Ansys and Abaqus solvers.

Assembly FE Modeling

New capabilities for distributed Assembly FE Modeling is associative to the Product Design Assembly in NX CAD. Teamcenter provides unique flexibility and productivity for CAE analysts in dealing with complex simulation models and integrating the separate simulation modeling efforts of program partners, engineering suppliers and globally distributed design/analysis teams.



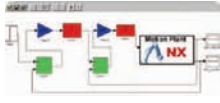
Advanced mid-surfacing and geometry preparation tools



Ply layup, draping and optimization.



Mesh once and re-use.



Co-simulation for Mechatronics design.



Visualization of flow lines for electronics cooling.

Multi-body dynamics

New capabilities in NX Motion Simulation enable the co-simulation of structures linked with and driven by control systems via a direct integration with the widely-used MATLAB SIMULINK product. Interactions between the structural characteristics and the control system characteristics can be evaluated and optimized during the product design process.

Integrated multi-physics solutions

Coupled multi-physics for flow and thermal

Extensions to the integrated Advanced Flow and Advanced Thermal solutions enable simulation of the complete range of coupled multi-physics phenomena including conduction, forced convection, condensation.

The NX Electronic Systems Cooling package can now simulate condensation and dust/particle impact, as well as the couple flow/thermal characteristics of an enclosed electronics system. This approach makes NX Advanced Simulation the only system able to simultaneously evaluate all primary failure modes for high tech electronics using a single master model and common user environment.

New capabilities in NX Response Simulation support the unique product development needs of the automotive, aerospace/defense, space systems and consumer industries. Examples of user productivity enhancements are customized dynamics simulation workflows for drop test and random vibration; function toolkit enhancements such as persistent templates; a graphical means to evaluate which vibration modes of the structure are contributing the most impact to the response at various frequencies.

NX Nastran

NX 6 offers extensions to NX Nastran capabilities across many solution areas including solver performance, glue contact, linear contact, advanced nonlinear, HDMP parallel processing and rotor dynamics.

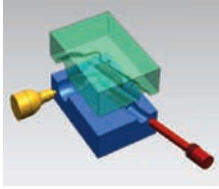
Integration of digital simulation and physical test results

New FE model correlation capabilities enable CAE engineers to validate the accuracy of their simulation results against real-world performance measurements from physical prototypes, as well as other legacy simulation models, and results in better design decisions at increased levels of accuracy.

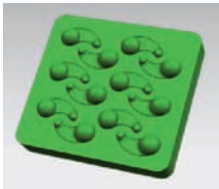
Analytical equivalents of physical test strain gages and sensors can also be defined at discrete locations in the digital performance model within NX Response Simulation. This enables the generation of stress/strain, displacement, velocity/acceleration and reaction force results from the virtual model which simulates the real-world prototype test results.

Simulation data management and process integration

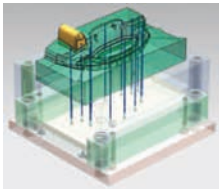
NX Advanced Simulation fully and automatically leverages the new CAE data model and CAE Assembly Product Structure available in Teamcenter's simulation solution. Integration with Teamcenter, the world's leading engineering process and data management solution, enables the global transformation of new product development processes including the integration of data from both COTS and in-house simulation tools.



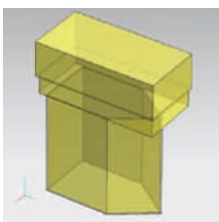
The parting process supports multiple region extraction for core, cavity and sliders.



Merge multiple inserts into a single plate.



Concept Design helps to create a quick first draft of a tool's design.



Undersize electrodes based on user-defined orbit patterns and spark gap.

NX Tooling

Mold Design

Mold Design and Teamcenter Integration. NX 6 continues to enhance the integration between NX Mold Design and Teamcenter for companies aiming to manage data and integrate tool design, manufacturing, and shop floor processes. NX 6 tool design and Teamcenter procedures are enhanced to provide improved performance based on streamlined memory utilization and a powerful mechanism to rename or reference library components.

Design re-use. NX 6 Mold Design offers new out-of-the-box project configurations that deliver improved approaches for collaborating with customers, working concurrently and handling design changes associatively. With NX 6 it is possible to edit the orientation of the CSYS mold and automatically update the mold project without repairing any parting features.

Workpiece and cavity layout. NX 6 adds design flexibility with a new sketch-driven workpiece, enhanced layout dialog interaction and the ability to automatically merge multiple inserts into a single plate.

Parting and core/cavity split. NX 6 continues to streamline the parting process with the ability to extract multiple parting regions for core, cavity and sliders. Parting regions are quickly defined using improved region definition and guideline creation methods. These enhancements are projected to contribute 15 to 25 percent time-savings when designing core, cavity and sliders.

Insert Design. NX 6 provides new tools for quickly splitting an insert body into a sub-insert and subsequently automating the addition of the insert foot.

Concept Design. For companies looking to work faster, Concept Design helps to create the first draft of a tool's design. Concept Design is used to quickly design the mold base and components as simple geometric abstractions that can be saved and loaded into full detail at a later time. Concept Design is lightweight as it does not include individual part files for the mold base and additional components.

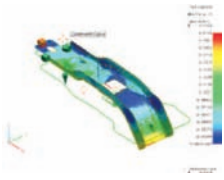
Tool Design Validation. It is important to validate the overall tool design for proper clearances and interferences early in the design cycle. NX 6 enables mold/die designers to quickly perform pre-defined checks and analyze results in a powerful clearance browser.

BOM – The NX 6 BOM delivers added flexibility with limitless columns and string lengths, rapid configurability with right-click ease-of-use, and simultaneous stock size editing for multiple components.

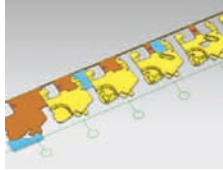
Electrode Design

Copy Electrode. NX 6 supports that ability to copy and re-use electrodes in multiple positions and orientations.

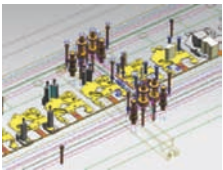
Undersize Electrode. With NX 6, the geometry for undersized electrodes can be created automatically based on orbital motion and spark gap parameters. Circular, square, triangular, spherical and user-defined orbit patterns are supported.



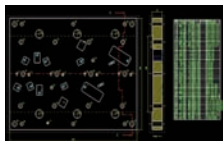
Use One-step Formability to analyze spring back.



Use Strip Layout drag-and-drop to position scraps and intermediate stages on the strip.



Rapidly design the die structure using die base and inserts automation.



Wire EDM start holes are automatically reported in the hole table.

Progressive Die Design

Progressive Die Design and Teamcenter Integration. NX 6 continues to enhance the integration between NX Progressive Die Design and Teamcenter for companies aiming to manage data and integrate tool design, manufacturing and shop floor processes. NX 6 tool design and Teamcenter processes are enhanced to provide improved performance based on streamlined memory utilization and a powerful mechanism to rename or reference library components.

Blank creation and prebend definition. The workflows for creating the blank and prebends for both straight-break and freeform sheet metal parts have been enhanced with NX 6. Intermediate stages are now created associatively. Up to six prebends can be defined for any bend. One-step Formability Analysis supports additional spring back constraints and has improved quality and performance. The blank and intermediate state creation enhancements are projected to contribute an additional 15 to 25 percent time-savings when preparing parts for progressive die projects.

Scrap design. In NX 6, individual pieces of scrap are defined using the NX Sketcher and Selection Intent. Scrap profiles and overlap/overcut details are associative and update to design changes. Scraps can be assigned to stations and then automatically placed into the correct strip layout position.

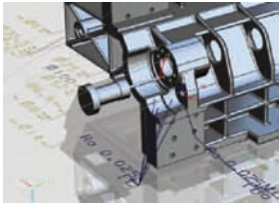
Strip layout. NX 6 provides drag-and-drop and automated strip layout workflows. Scraps and intermediate stages can be placed at the appropriate stations using drag-and-drop and copy and paste. Stations can be easily inserted and deleted. The strip pitch, width and feed direction can be edited and the strip updated associatively. Scraps and intermediate stages with predefined station numbers are automatically placed into the proper strip location. The strip simulation has also been enhanced with associativity.

Die base and inserts. In NX 6, piercing punches/dies are enhanced with design automation for freeform holes/scraps and a variety of piercing punch heads is provided. A new set of z-bend punches/dies with enhanced placement and sizing automation is also available.

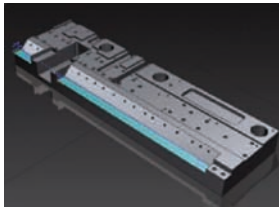
Tool design validation. It is important to validate the overall tool design for proper clearances and interferences early in the design cycle. NX 6 enables die designers to quickly perform predefined checks and analyze results in a powerful clearance browser. NX 6 also provides automatic report generation and attribute definition from piercing force calculations.

Documentation. NX 6 provides the ability to specify a wire EDM start hole that can be automatically reported in the downstream hole table. The NX 6 hole table has been enhanced to support a wider variety of hole types with improved labeling. The NX 6 BOM delivers added flexibility with limitless columns and string lengths, rapid configurability with right-click ease-of-use and simultaneous stock size editing for multiple components.

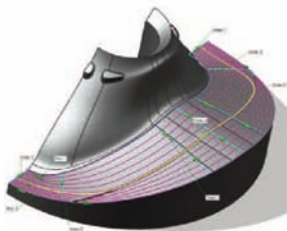
Design change efficiency. NX 6 Progressive Die Design includes many enhancements for streamlining the design change process. Intermediate stages can be created associatively and updated to design changes. WAVE Update Control can be used to identify components affected by design changes and control design propagation. And, the strip pitch, width and feed direction can be easily edited to update the entire project.



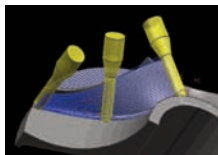
Feature Machining can be driven by standard PMI data from the NX CAD model.



Expanded feature recognition covers a wide range of new slots, pockets, multi-stepped holes.



5-axis streamline patterns can finish many different surface layouts.



NX CAM

Key topics

PMI-driven machining. NX 6 reads product and manufacturing information (PMI) directly from an NX part model and uses elements of that data to drive NC programming in NX CAM. This is one of the first examples in the industry of standards-based PMI model annotation being used to automate a downstream process, in this case CAM. The solution is closely linked to the enhancements to Feature Based Machining. The PMI data such as the specified dimension, tolerance and surface finish is used as input to the selection of feature machining operations. We are seeing significant interest in the use of standardized PMI data as a means of documenting requirements for component suppliers.

Knowledge-driven feature machining. NX 6 has incorporated technology from the specialist Tecnomatix® software Machine Line Planner system into NX CAM for extended feature recognition and a new, much more efficient technology for building each feature machining process from a knowledge database.

The power of feature recognition has expanded greatly, with the recognition of more features in more scenarios than ever. Multi-stepped holes, including chamfers and rounds are recognized. Additional flats, slots and pockets are recognized also.

Knowledge-driven feature machining builds (or generates) the sequence of machining operations most suitable for each of the identified features from a database of core machining operations (the machining knowledge database). As it builds (generates) each machining operation sequence, the core software (the inference engine) checks for any relevant PMI data – such as surface finish or tolerance that will affect the selection of a specific machining process from the knowledge database.

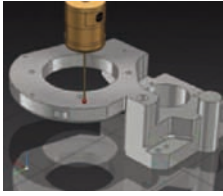
The new machining system feature eliminates the duplication of machining processes across a complete feature set which could cover thousands of permutations. The process building blocks that exist just once in the machining knowledge database can be defined as preferred machining processes and the criteria that determine their use are dramatically easier and faster than with traditional FBM systems that can have individual processes copied many times in each variation of a feature machining process sequence. No computer programming skills are required to use the machine knowledge editor.

Applying feature-based technology can save as much as 90 percent of programming times for the features implemented. Easier implementation delivers this advantage more quickly. The new system is far easier to maintain and enables continual refinement and optimization of preferred machine strategies and tooling.

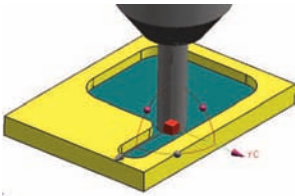
Streamline toolpath. NX 6 builds on the introduction of a new toolpath capability (first provided in NX 5). The Streamline toolpath is particularly well suited to high-speed finishing. It enables the NC programmer to define a machining strategy in which the cutting direction and pattern is optimized to the shape of the part. Unlike many other systems, NX 6 does not require the programmer to construct new geometry or repair poorly modeled or translated surfaces. The Streamline toolpath provides a high degree of flexibility in working with any model and in controlling the flow and patterns employed. Flow and cross curves are automatically positioned and more can be added and/or edited to drive the flow and direction of the cutter path relative to key elements of the part topology.

The Streamline toolpath in NX 6 is particularly powerful as a 5-axis application with the streamline functions as a very flexible drive surface specification. The Streamline toolpath adjusts both the cutting pattern and the tool axis with a smooth transition from one end of the cut region to the other.

The Streamline toolpath also provides automatic dual contact support for special cases where the limits of the region to be cut are determined by the last possible pass of the cutter at the “wall and floor” dual contact position.



Probes can be defined, programmed and simulated providing cycle output for on-machine probing.



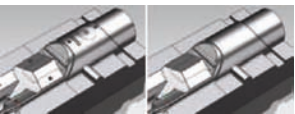
Dragable tool preview makes it easy to use the right tool for the job.



Operation Navigator includes additional information.



New tutorials are available.



Direct modeling makes it easy to handle shutoffs, patches and other model tweaks that aid CNC programming.

On-machine probing using Renishaw probe cycles. Probing tools can be defined, programmed and simulated with the new capabilities. Probes can be fully defined as a solid and maintained in the tooling library. Standard Renishaw motion cycles can be directly programmed, and the results are simulated.

Other CAM topics – Ease of use

Dynamic tool preview display. Users can instantly see if the selected tool fits the geometry situation by quickly dragging the tool display around your NC environment. No toolpath is required, and the suitability of the tool for the job is instantly seen.

More informative Operation Navigator. The Operation Navigator is the information hub of the NC program. Now it is more informative than ever, with the inclusion of machining time and User Defined Event displays. Also handy is its ability to cut and paste operations from a different NC file.

New operation status. Operations can be marked “approved” to avoid them going “out of date” due to minor changes in previous operations or minor, well-understood geometry updates.

New toolpath status. Toolpath can be marked as “locked”, preventing it from being recalculated accidentally. This approach keeps certified programs from incurring unexpected updates, and maintains strict user control over any new calculations.

New tutorials. Getting started is easier than ever with additional built-in tutorials for the new user. These tutorials now include special options for:

- Airframe parts
- High-speed milling
- Post processor installation

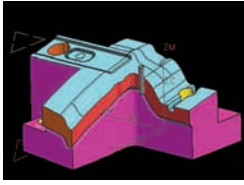
Model editing. NC programmers often have to make edits to models in order to program just the motion they want. A full set of model editing tools are available to the NC programmer, including assembly tools, surface patching tools and direct modeling.

Other CAM topics – Milling, high-speed machining

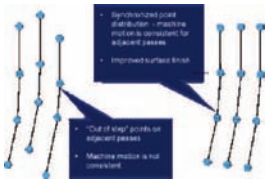
Curve/edge machining. Updated curve/edge machining extends the popular Streamline selection style into the machining of curve and edge geometry types. The curve/edge operation types can machine relative to solid edges without any supporting wireframe geometry. Negative stock allowances work with wireframe, edges and even text objects (including 3D text) to make tracing and engraving applications easy. Multiple offsets are provided and the capability works in 3-axis and 5-axis scenarios.

Solids-based 3D profiling. 3D profiling is ideal for cutting die components, where vertical faces are cut along their edge chains in 3 axes. The solids-based approach stays associative should the faces and their edges be modified plus it provides the safe, intelligent toolpath that is the standard for all solids-based cutting approaches. Edge chains at the top or bottom of the targeted faces can be traced, with multiple depths and side passes.

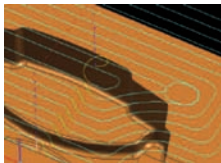
Synchronized point distribution. Streamline operations and other 3-axis area milling operations take smooth interpolation to the next level by maintaining coherence of point distribution from one cutting pass to the next. No matter what interpolation options are employed for control, maintaining similar point distribution across many passes results in the best possible surface finish. This approach extends the range of methods for achieving good surface finish.



3D profiles associatively track vertical edges in space.



Synchronized point distribution works with your controller's interpolation scheme to provide the smoothest finishes.



Advanced corner rounding keeps feedrates high and extends tool life.



Fringe plots show the amount of uncut material on in-process models.

Toolpath splitting. NC programmers can take shortcuts in the programming process by calculating all the cutting motion for a large or deep area once, then splitting the result across several tools. Toolpath can be split by time or distance or by specific motion events. Most significantly, toolpath can be split at the point that holder collision occurs, leaving remaining motion to be performed by a longer tool. In this way, several tools for increasing length can be applied to a deep cavity with only one quick toolpath calculation. Programming is quick and easy, and the use of the stiffest, fastest tools can be maximized.

Toolpath editing. A revamped toolpath editor provides interactive tools to quickly and easily make minor adjustments to toolpath results such as moving, trimming, checking gouge motion, reversing cut order and inserting additional motion elements.

Corner rounding. Roughing cuts are faster and smoother than ever, with an updated approach to smoothing out any 2 1/2-axis pattern. Corners remain smooth and consistent, whether very tight or relatively open. Smooth climb cutting is always maintained, regardless whether extra loops are required to reach all the material. Smooth, fast stepovers can be invoked as ess-shaped transitions from one pass to the next, or they can be incorporated into the corners themselves for the ultimate in clean, fast area coverage with no wasted motion. Depending on geometry and control options, these optimized corner treatments can result in roughing cuts that complete 10 percent faster or better, compared to conventional offset patterns.

In-process workpiece fringe plots. While displaying the in-process workpiece or displaying cut material, programmers can get a clear view of the remaining uncut material by invoking the new fringe plot display. Different colors indicate the thickness of remaining material relative to the actual part faces. Programmers use these displays to easily develop targeted semi-finish cuts.

Other CAM topics – Production-ready output

Machine tool 3D model samples. Several new machine samples are included, complete with machine tool drivers for simulation. These cover an even wider variety of machine configurations, making it easier than ever to simulate the latest models from the machine builders.

Post Builder updates. The interactive Post Builder application is one of the most powerful post processor development tools available and NX 6 provides new capabilities for handling user defined events (UDE) with more granularity. New drag-and-drop options make the use of Post Builder easier and faster.



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