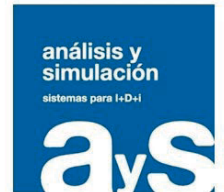




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Answers for industry.

What's new in NX 9

Smarter decisions. Better products.

Benefits

- Execute faster, more intuitive 2D sketching with Synchronous Technology 2D
- Faster, easier industrial design with Realize Shape
- 4th-generation design allows collaborative work on massive designs
- Achieve easier, faster modular design
- Spend less time looking for commands with extensive usability enhancements throughout NX
- Significantly increase update speed

Summary

NX™ software release 9 from Siemens PLM Software is more powerful, faster and easier to use than ever. Modeling improvements such as Synchronous Technology 2D and Realize Shape give you more tools to create geometry quickly. It is easier than ever to access the information you need to make smarter decisions about your design. Usability has been greatly enhanced, particularly in Drafting and PMI, while retaining the customizability that you demand. NX 9 introduces new simulation capabilities for mesh morphing, provides greater control over mesh quality and allows you to specify surface wrapping resolution for better creation of fluid domains. Performance enhancements greatly reduce analysis time for large models, and there are new tools for analyzing dynamics of composite parts. You can boost your part manufacturing productivity with new capabilities that give you more control over your machining strategy, automatically convert from 3-axis to 5-axis

tool paths, review your inspection results in the context of your 3D model and access the Manufacturing Resource Manager.

NX for design productivity Synchronous Technology 2D

Synchronous Technology 2D gives you the same freedom and speed in 2D that the groundbreaking synchronous technology gives you for 3D geometry. It even works the same way as Synchronous Technology 3D, with simple, intuitive push-and-pull commands when using the NX sketch environment. Easily modify 2D geometry regardless of where the data came from or how it was constructed. Sketches modified with Synchronous Technology 2D are automatically updated according to changes in real-time. For instance, if you move or offset a curve, all associated geometry intelligently adapts. The relevant relationships are preserved while irrelevant relationships are removed to enable the intended change to be made with ease. If

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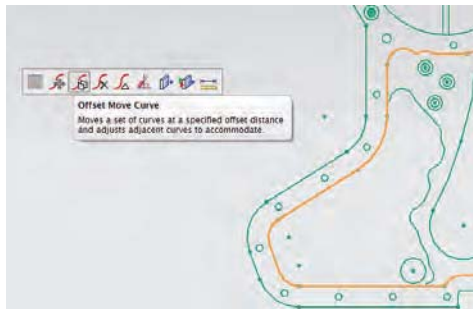
What's new in NX 9

Benefits *continued*

- Reduce memory usage by up to 40 percent for large models
- Achieve a new level of cut control and faster NC programming
- Inspect a wider range of part types, including sheet metal components
- Easily access a fully classified tool library, while working in NX
- Accelerate the preparation of tooling models
- Morph existing meshes to quickly fit modified geometry
- New mesh controls help create quality meshes in fewer steps
- Parallel thermal solver reduces analysis time for large models

you delete a curve, the sketch automatically adapts to “close” the geometry around the deleted curve. Overall, Synchronous Technology 2D is up to 5 times more productive than standard sketching techniques.

Synchronous Technology 2D means that your sketches are now more versatile because you can change the design intent of the sketch while preserving integrity. It's another way of adding intelligence to all 2D data, including legacy and non-NX data. Large 2D layouts are now more usable and efficient because you can quickly edit large numbers of curves at once with no impact on performance. Using Synchronous Technology 2D makes it faster and easier to understand and implement design changes, which means you spend less time working on models and have shorter lead times.

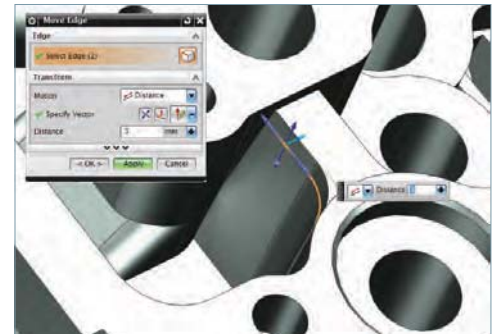


When you move a curve with Synchronous Technology 2D, the associated geometry adapts.

Synchronous Technology 3D

NX 9 is the sixth release with synchronous technology, which is maturing, improving and gaining more functionality with every release. New synchronous technology functionality allows you to edit a shape by directly manipulating edges instead of the usual method of editing faces. Faces adjacent to the edges that you edit adapt to follow their motion.

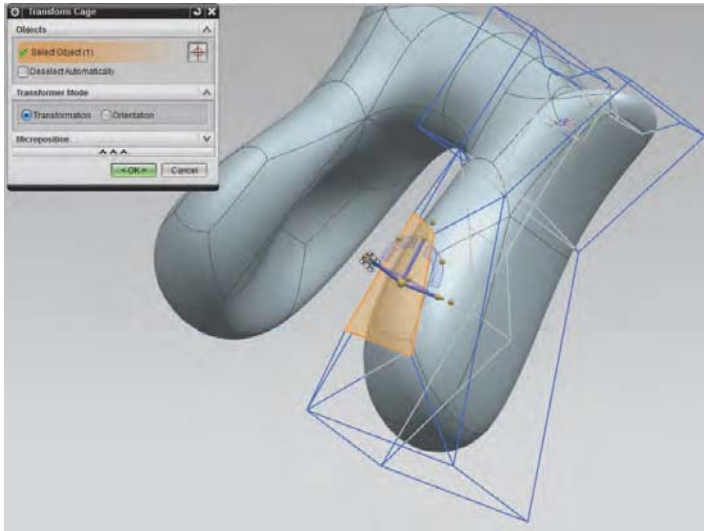
New face selection options make it easier to select the geometry that you want to modify with synchronous technology. It is now easier and faster for you to clean up models with new tools for selecting and deleting blend faces. You can specify a range of blend radii and all blends within that range throughout the model. With every release, synchronous technology adds more tools that let you modify your geometry the way you want.



As the edge is moved, the associated faces adapt.

Realize Shape

Realize Shape is an exciting new design method for advanced subdivision shape creation using NX. It is intuitive and easy to use. You begin with a primitive shape and manipulate a surrounding “cage” to morph the shape as needed. The cage faces can be subdivided as much as you want to give you greater control. It's easy to create smooth transitions using Realize Shape, and there are many options for shape creation. The end product is high-quality B-surfaces as an editable NX feature in well under half the time needed with previous methods. This allows you to rapidly conceptualize ideas without the need for expert knowledge. Realize Shape can be used in combination with, or alongside, other surfacing and design tools.



Realize Shape can be used to create advanced shapes quickly and easily.

Ray Traced Studio

The new Ray Traced Studio option enables you to view a high-quality image of your design in real time. Taking advantage of multi-core and accelerated graphics processors, Ray Traced Studio can display realistic lighting and inter-part reflections to help you make design decisions earlier.



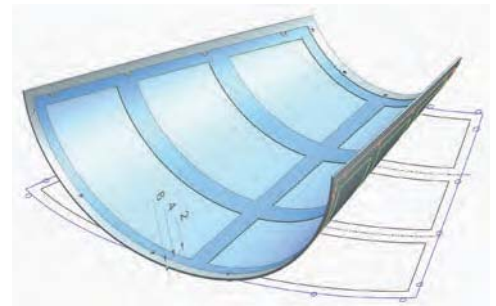
Ray tracing shows real-time high quality images of your designs.

Feature-based modeling

Feature-based modeling is as important as ever, and NX 9 delivers a number of enhancements in this area. It is now possible for you to create complex rib

structures in a single feature from a network of curves; in the past this would have required multiple features. This means less clutter in your part history and faster model regeneration.

You can now sweep a revolved tool along a smooth path. This simulates manufacturing processes, and allows you to create slots that were difficult to create in the past. Patterning is now more consistent between types and easier to use. Using Draft features, you can now use a nonplanar object as a parting line, such as a surface. This saves time compared to the old method of selecting edges.

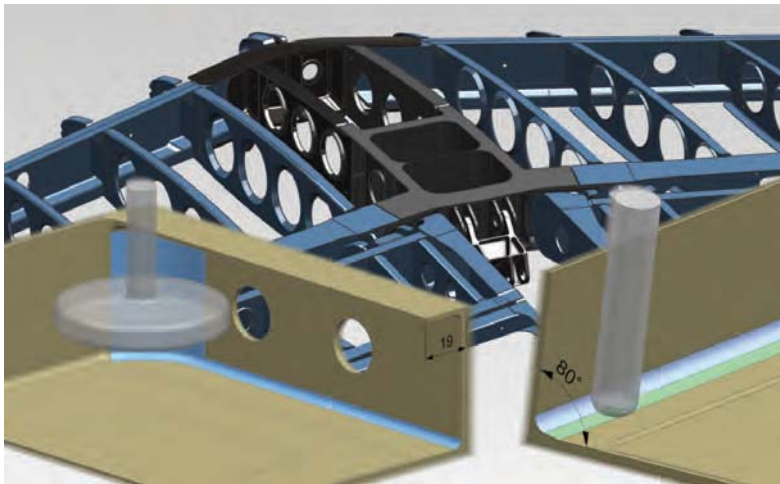


Developable surfaces can be flattened without deformation.

Our customers are a primary driver of our development, and many of the improvements in NX 9 were conceived for the aerospace industry. With the new “developable” option within the Ruled feature, surfaces can be flattened without deformation (i.e. wrinkling, stretching or tearing). You can now also wrap and unwrap curves onto developable surfaces. This enables you to show geometry in both formed and flattened states. These tools are especially valuable in workflows in which parts must be shown in multiple states, such as manufactured and installed.

The thicken command has been enhanced with the ability to select different thicknesses for different regions, greatly simplifying the process of creating chemical mill pockets. Pocket blends are easier to create with tools such as Analyze Pockets, which looks for undercuts, negative angle draft, tool inaccessibility areas and other

manufacturability issues. Blend Pocket not only helps you create blends based on the cutter parameters you supply, but also helps you create the fill geometry, such as cutter ramps. Inter-part design is enhanced as well with updates to WAVE and product interfaces which make it easier for you to link geometry between parts.



Pocket Blends are easier to create in NX 9 with tools like Analyze Pocket.

Modular design

Modular design with Part Modules is enhanced to support several new workflows. You do not have to subdivide a model into separate bodies in order for multiple designers to work on it anymore. Part Modules also supports local feature updates, so you have the ability to update individual Part Modules or perform a full feature update as needed.

Assembly design

NX 9 introduces a number of customer-driven enhancements to assembly modeling. It is now possible to reorder objects in the Assembly Navigator using drag-and-drop or the Reorder Components command. The Assembly Navigator display is cleaned up because multiple instances of the same object are now packed by default. These changes help you set up your assembly in a way that's easy for you to understand and modify.

The new Align/Lock constraint allows you to fully constrain cylindrical objects such as bolts or pins with one constraint, saving clicks and time. Also, NX 9 now supports Component Patterns in assemblies, which offers a greater range of pattern types than in the past. The user interface has been revised to be consistent with Pattern Feature, making it more efficient and easier to learn.

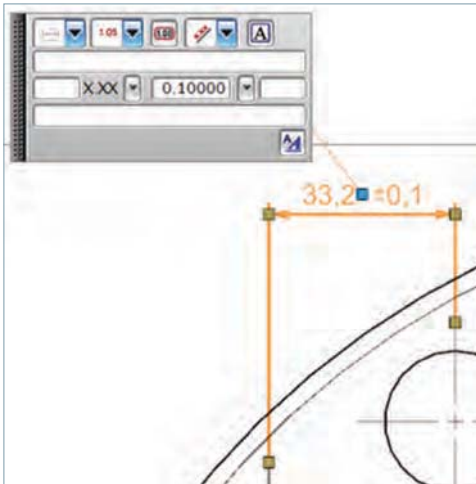
Annotation

Drafting and Product and Manufacturing Information (PMI) are more efficient than ever with NX 9, featuring the most significant usability enhancements in years. Like the rest of NX, Drafting and PMI take advantage of the new customizable ribbon bar interface. The overall user interfaces in Drafting and PMI are aligned more closely with other NX modules to ensure greater consistency and reduce the learning curve for new users. Consistency was also the driver behind making the interfaces the same for modifying and editing annotation objects.

Many Drafting and PMI tasks are simplified and streamlined to reduce mouse movement and save clicks. For instance, the number of dimension creation commands is reduced from 22 to 9 with no loss of functionality. Giving you the right information at the right time is a top priority with NX, so dialog boxes in Drafting and PMI are now configurable based on context, showing only the settings relevant to the task being performed. Using a new settings search tool saves you time when you need to locate a particular setting within the preferences NX provides. As always, NX is constantly updated to comply with all major standards; a number of enhancements in NX 9 make it easier to create fully compliant drawings and models.



Part Module enhancements make collaborative design easier than ever.



The new Drafting interface is simpler and easier to use.

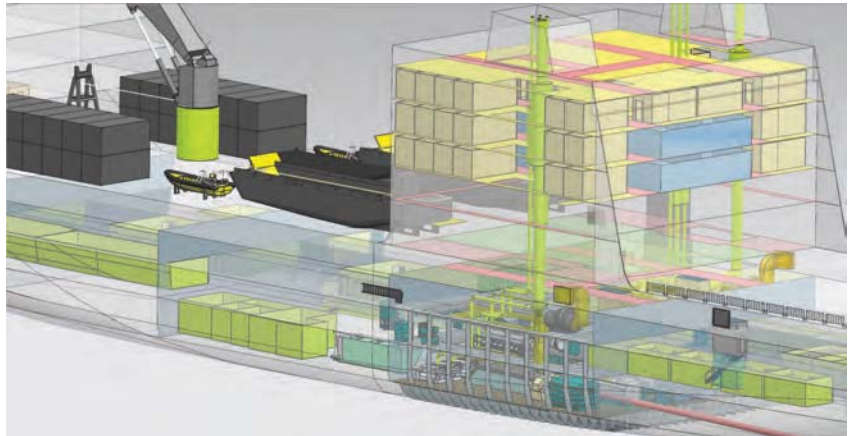
4th-Generation design for the marine industry

Designs and design teams seem to constantly grow. 4th-Generation Design (4GD) is a new design and data management paradigm offering unprecedented efficiencies for large teams working on large designs such as ships. It does this by enabling concurrent and collaborative in-context work. Utilizing a component-based approach, rather than an assembly structure-based one, 4GD works by breaking down the overall design into design elements, which are organized using partition schemes. From there, you can be assigned subsets of the whole design. In NX 9 and Teamcenter® software release 10.1, 4GD is implemented for the marine industry, with other industries to follow in future releases. The component-based 4GD approach enables collaborative design in multiple configurations, is scalable to products with millions of components and is up to an order of magnitude faster in many traditional design-in-context processes.

4GD utilizes the power of NX and Teamcenter together and is seamlessly integrated between them, with consistent user interaction and full support for NX parts and assemblies, so no migration is required. 4GD enables large design teams

to work concurrently with no locking of assemblies; only the actual data to be changed is locked. This is a key part of enabling collaborate work. Performance is enhanced by only loading the design elements needed for the design task, instead of having to "load and hide" the whole assembly. 4GD technology can be used for a single ship or a class of ships. This allows you to organize massive amounts of design data in multiple views, and it supports high performance searches across all product data. With 4GD, decisions about the organizational structure and assignment of design data don't have to be made in advance.

4GD is more than a set of tools – this is an entirely new way of working on large designs, giving you access to the data you need, and only the data you need. This gives you the freedom to work alongside your whole team simultaneously.

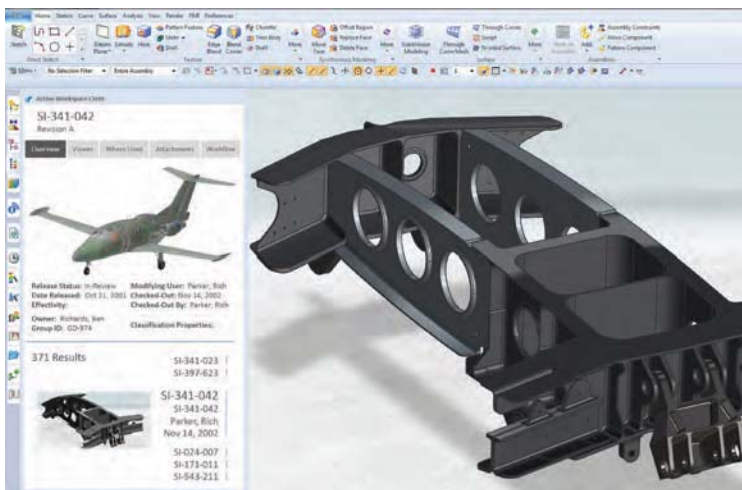


4GD enables large teams to work together on massive designs like ships.

Active Workspace Client

The Active Workspace Client (AWC) is a powerful tool for finding information about your products quickly and easily. It is embedded directly in NX, and can also be accessed in any web browser on any device, including mobile devices. This makes it possible for you to quickly access Teamcenter data from within NX or anywhere else. AWC helps reduce the

amount of time you spend searching for information with powerful interactive search and filtering capabilities which can access external systems, eliminating the need to search in multiple places. Because AWC is context-aware, you see only the information relevant to the task being performed, saving you time. By highlighting issues in Visual Reports, you can see the big picture so you can make smarter decisions more quickly.



The Active Workspace Client helps you find vital product information quickly.

Fundamentals

The user interface in NX 9 is more intuitive and faster than ever. Featuring a customizable ribbon bar, the interface is both richer for experienced users and easier for novice users to understand. Commands are organized into logical tabs and groups which provide access to frequently used commands with a minimum number of mouse clicks. You now have access to more commands along with a larger graphics window, increasing efficiency. Also, the ribbon bar is scalable, no matter what your window size is; all commands are available and do not get clipped.



NX 9 features a customizable ribbon bar interface.

Every NX application for design, simulation and manufacturing has a Home tab which contains the most frequently used commands for that application. This means you don't have to spend as much time looking for the commands you use most often. It's also easier to find infrequently-used commands because Command Finder is always visible, so you just type what you are looking for.

Unlike similar interfaces in other products, the ribbon bar in NX 9 is fully customizable. You have full control over the layout, size and appearance of icons and you can create custom tabs and groups. You can still create command bars on the borders and bottom of your graphics area, and you can undock them to get them out of the way if needed. It is now easier than ever to add commands. All customizations are saved in Roles, which can be exported and imported. It's all about enabling you to set up NX the way you want.

Performance Increasing performance is an ongoing priority in NX development. By reducing the number of features which need to be updated after a change, NX 9 provides a significant increase in update speed, especially on the massive parts which are becoming more common. In addition, you now have greater control over updates with powerful but simple part-level feature organization tools, so you aren't always forced to update the entire model. Modular design with Part Modules gives you even more control in this area. Models also display faster because NX 9 takes greater advantage of today's multi-core processors. Adding new functionality while simultaneously increasing performance means that NX is quicker and more responsive, but also more powerful than ever.

NX 9 for simulation productivity

NXCAE

NX CAE is a modern simulation environment for simulation modeling; structural, thermal, flow, motion and multiphysics simulation; optimization; simulation data management and automation; and simulation-driven design. NX CAE 9 includes numerous enhancements to improve modeling efficiency and reduce simulation process time. Highlights of the NX CAE 9 release include the following features:

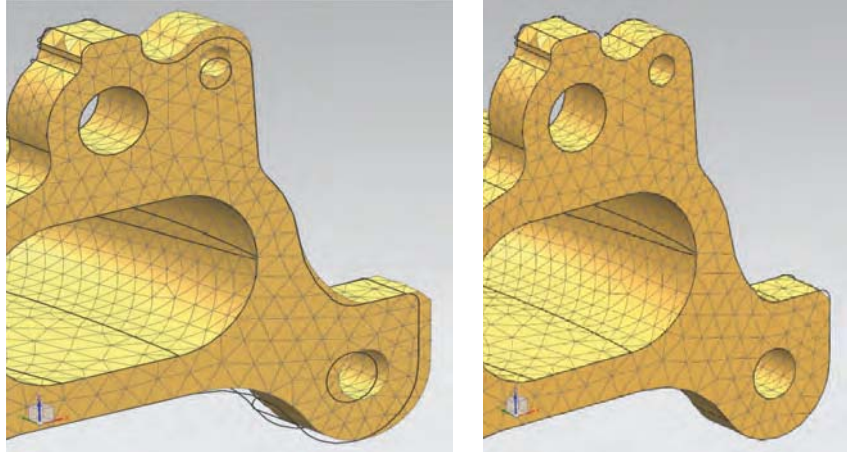
Simulation modeling and results visualization

Mesh Morphing This release includes new commands that allow you to modify an existing mesh to conform to slight changes in geometry by moving node locations to conform to the new dimensions. Morphing is not a remesh, and the software tries to keep the overall mesh topology constant. The software enables you to compute new locations for the nodes in the existing mesh to conform to the changed geometry and expands or shrinks the mesh to fit the modified geometry. This maintains existing node labels and IDs and speeds the update process.

Morphing provides an alternative to the mesh update that occurs within the FE Model Update command. With a mesh update, the software deletes and recreates meshes that are affected by change to the underlying geometry.

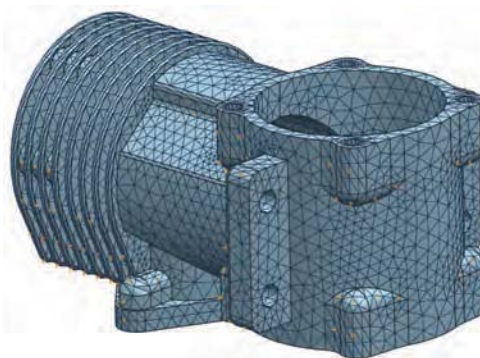
You may want to morph, rather than update, a mesh when:

- It is important that you preserve your current node and element labels
- You want to retain shape and size of the existing elements in a mesh in areas that are unaffected by modifications to the CAD geometry



Existing mesh (left) will be morphed to fit new geometry dimensions which you can see outlined. Morphed mesh (right) is the same mesh except nodes have moved to new locations to match revised geometry.

Mesh controls The NX 9 release includes new ways for you to control and manage meshing for tetrahedral and surface elements around cylinders, fillets and holes. Improvements for tetrahedral elements allow you to make local 2D mesh requirements part of the 3D process or avoid the need to locally mesh in 2D around cylinders/fillets. Improvements for surface meshing include new controls for fillets and mapped holes, the ability to suppress holes within the 2D free mesh command, and improved controls for target minimum element size.



Tetrahedral mesh utilizing mesh controls for cylinders.

NX CAE 9 also introduces a unique way for you to manage individual control settings, which can also be applied to automated batch meshing processes to deliver better quality meshes in automated processes and eliminate manual rework.

Improved XY graphing integration This release includes a number of new enhancements around XY graphing capabilities to enable more interactive evaluation of simulation results. Examples of these enhancements include extended selection capabilities, orbit style graphs and expanded complex results support.

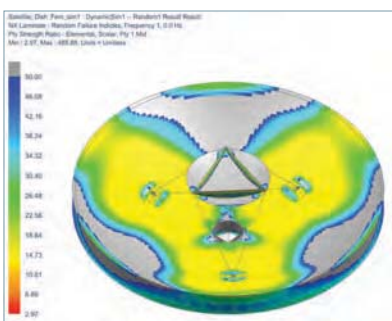
Structural analysis

Rotor dynamics solution support

Beginning with NX CAE 9, you can create a model for NX Nastran® rotor dynamic analysis to help you predict the dynamic behavior of rotating systems, such as aircraft engines or drive shafts. This enhancement is available in the NX Nastran environment and allows you to define items such as bearing supports, rotating and stationary portions of the model, and system-wide and rotor-specific solution options, among others. Supported solution sequences include frequency response, transient response and complex eigenvalue rotor dynamic analyses.

Laminate composite dynamic simulation

The new laminate composite dynamic solution process provides an accurate and efficient means for you to evaluate the performance of composite parts when they are subjected to base-driven random vibrations, which are prevalent in the aerospace and automotive industries. The laminate dynamic solution supports 2D and 3D laminates and enables you to compute results for peak ply stresses, strains, failure indices and strength ratios.

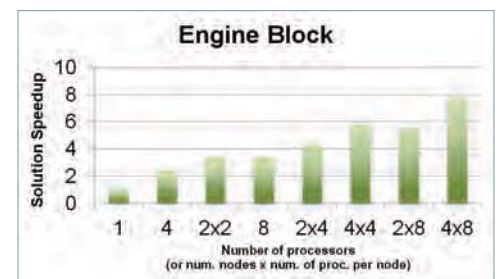


Ply strength ratio computed from a base-driven random vibration simulation for a space antenna made from laminate composite material.

Enhanced durability interface with NX Response Simulation The NX 9 release of NX Advanced Durability includes an enhanced interface to NX Response Simulation that enables you to calculate stress/strain histories on the fly from a Response Simulation event, which is faster than directly reading the stress or strain histories from the RS2 file. This results in a significant time reduction of up to 98 percent since there is no need to solve for time history of stress/strain from NX Response Simulation transient events.

Thermal and flow analysis

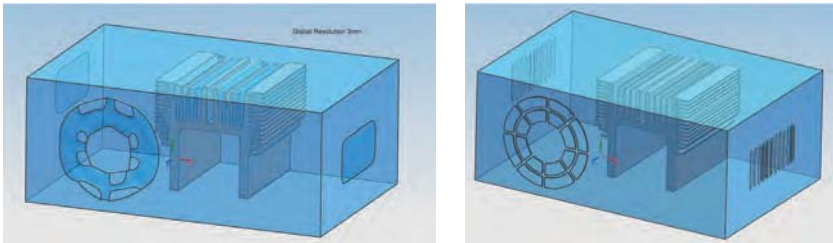
Parallel thermal solver New for the NX 9 release is the parallel thermal solver which helps speed solution time for large models and long transient solutions through parallel processing on multiple CPUs. Depending on size of the model, complexity and number of central processing units (CPUs), it is possible to simulate models up to 8 times faster. The parallel thermal solver uses domain decomposition techniques to help you split the thermal system of equations and distribute the computation workload across multiple processes. This enables the solver to support a large numbers of elements and nodes and a significant amount of conductance.



Speed up using parallel thermal solver for an automotive engine block model with 1.7 million elements and 7.6 million conductances.

Surface wrap local resolution constraint

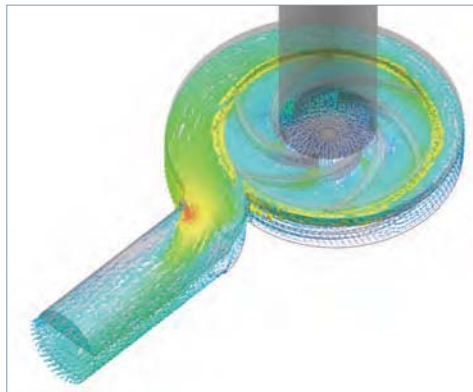
You can apply the new local resolution constraint to surface wrapping recipes to refine or coarsen fluid bodies in selected regions, enabling you to preserve more detail in the fluid bodies generated by the surface wrapping tool or remove unnecessary details. This constraint is easily managed using the simulation navigator, and can easily be copied to other fluid domain recipes within the model.



(Left) Fluid domain geometry resulting from a global resolution only doesn't capture vent opening and heat sink features accurately. (Right) New surface wrap local resolution settings used for vent openings and heat sink create a more refined and accurate fluid body.

Parallel flow solver enhancements

The NX 9 release includes various enhancements to the parallel flow solver. For example, performance enhancements include reduced memory usage of up to 40 percent for large models, and increased performance to speed overall simulation



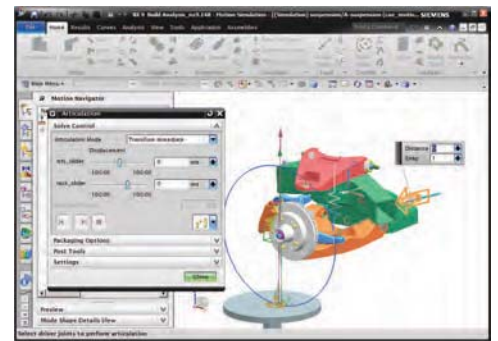
Flow simulation using rotating frames of reference is now supported by the parallel flow solver.

time. Other enhancements to the parallel flow solver include extended support for more simulation objects such as rotating frame of reference and mixing planes and modeling objects, such as tracer fluid and non-Newtonian fluid.

Motion analysis

Interactive articulation methods

Articulation in NX Motion has been extended to include two new methods which are useful for interactively checking interference at critical parts of the motion envelope. The transform immediate option enables you to interactively drag joints to new positions in the graphics window, and then you can use NX to solve and animate the mechanism immediately. The transform delay option lets you articulate the mechanism to a target position by specifying one or more joint displacements. You can then use NX to solve and articulate the mechanism according to those specifications and then solve and animate the joints simultaneously. Joint limits can also be turned on or off for articulation.



New interactive articulation options let you drag joint displacements directly in the graphics window.

Full journaling support All NX Motion preprocessing, solution and postprocessing commands now provide full journaling support to enable the automation and customization of motion solutions using NX Open.

Multiphysics

Condition sequence manager The new condition sequence manager allows you to define and manage a model's duty cycle across solution types. Condition sequences can include any combination of structural, thermal or flow parameters. An example of a condition sequence would be the conditions applied to an aircraft engine during takeoff, climbing to altitude and level flight. Using NX CAE, you can import and manage condition sequences and use them to drive boundary condition values at specified time steps. If your company uses a standard format for defining condition sequences or follows standard practices regarding missions and duty cycles, you can import this data to capture these practices in your analyses and ensure consistency across teams and platforms.

Create and manage sequences for missions using the new condition sequence manager.



NX 9 for manufacturing

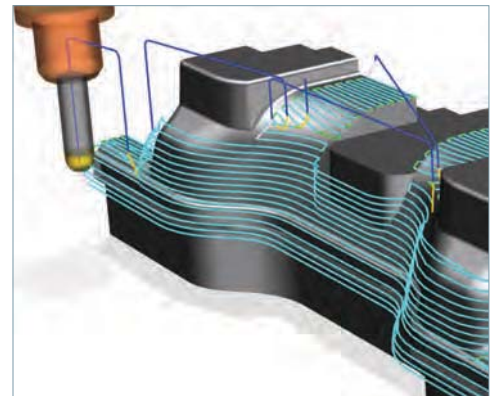
NX CAM

Increase your manufacturing productivity with new industry specific capabilities in NX CAM 9.

Enhanced control over cut regions for mold and die parts, faster programming of multiple-part setups and improved machining efficiency of complex parts can maximize your NC programming efficiency.

Mold and die machining

New Interactive Cut Region Management provides you with a new level of control in planning machining strategies. You can preview, change, reorder or accept the regions and the associated operations before generating tool paths. This capability is especially useful for more complex parts for which careful process planning is required and when you want to define exactly how a part will be machined, region-by-region.



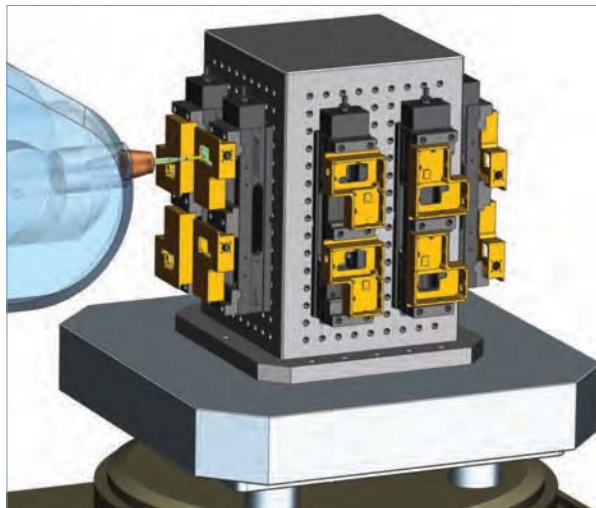
Interactive Cut Region Management provides advanced control of machining strategy.

NX 9 delivers new rest machining functions that offer hybrid machining strategies for greater efficiency in addressing uncut areas. Being able to use a range of cut patterns in rest milling enables you to achieve the optimum quality surface. Intelligent top-down sequencing of uncut corners and valleys enhances cutting action and extends tool life.

High-speed noncutting moves with new techniques for engage, retract and stepover motion maintain the fastest feed rates on highspeed machines, while producing high-quality surface finish. Optimized curves (splines) are fit from one tool pass to the next, minimizing machine stress and maximizing your ability to control a drive at its highest feed rates. Robust collision detection along with alternative connection strategies ensure that transitions are safe as well as fast.

Prismatic part machining

New multiple part programming speeds the development of tombstone and fixture configurations, intelligently distributing tool paths to multiple components in assemblies. You can quickly assemble the fixtures and multiple work pieces common in production machining using the extensive assembly modeling capabilities of NX. Complete machining sequences developed for one work piece can be immediately distributed to other work pieces. In this way, complete jobs can be prepared in the same time it takes to program a single part out-of-context. By working in the full setup context, you can easily see accessibility and clearance issues and adjust tool paths only as needed.



Multiple-part programming intelligently distributes machining sequences to the assembly's components.

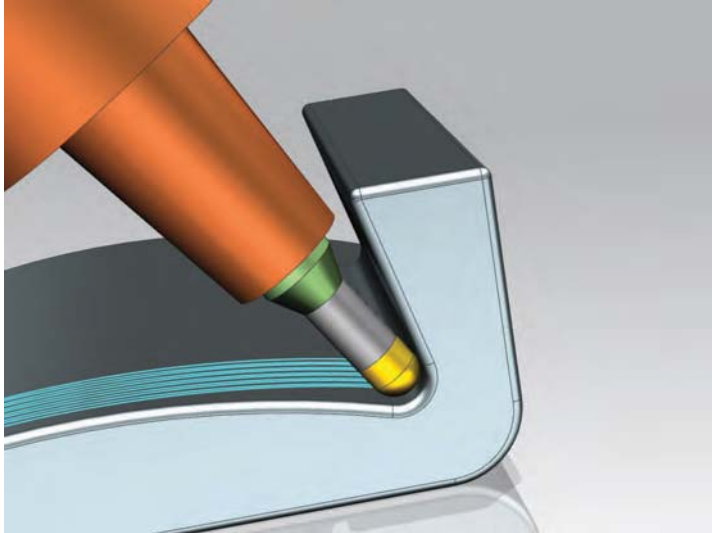
NX 9 brings the speed and convenience of feature recognition to the ordinary hole drilling process. Holes are identified, filtered and grouped automatically, which simplifies programming. Conveniently grouped features now underpin the entire spectrum of hole machining from the most manual to the most automatic. No tedious geometry selection is necessary – you just identify the feature group and drill. Errors associated with geometry selection are reduced. No holes remain uncut, and holes sharing the same attributes are cut together.

New techniques for using T-cutters to create horizontal grooves mean that you can precisely address both top and bottom faces. With a single programming step, the T-cutter can be directed to rough the center of the groove and finish both top and bottom using the T-cutter's separate top and bottom contact points.

Complex part machining

The ability to avoid holder collisions via automatic conversion of 3-axis tool paths by tool tilting is greatly extended in NX 9. The new functions can now be applied to roughing operations or 5-axis operations as needed for additional holder clearance or to avoid machine rotation limits. Tilting away from holder collisions enables you to use shorter tools, which can be especially beneficial in maintaining higher material removal rates for roughing. In all cases, new controls are available to specify the tilting preferences so that the result is well-suited to the geometry and tooling situation. This capability is well-suited to making large molds with 5-axis equipment.

The enhanced 5-axis roughing and finishing in the Turbomachinery Milling module improve machining efficiency and surface finish. Flat and bullnose end mills are now included as roughing tools, increasing roughing performance in many cases. Blades with the appropriate shape can now be swarfed, finished in one pass with the side of an endmill for the best surface finish and fastest cutting.



Automatic 3- to 5-axis tool paths conversion enables efficient machining of parts with complex geometry.

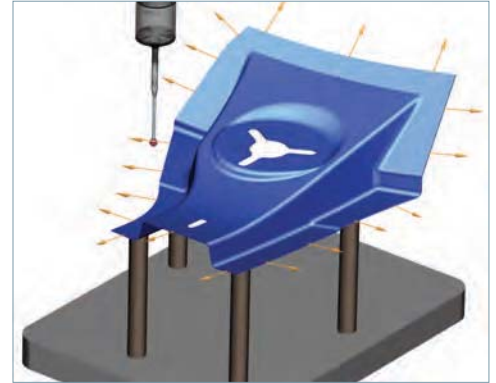
For programming of cylindrical parts, NX 9 enables you to use a specialized 4-axis cutting method resulting in optimized and safe machining. Pockets and slots with cylindrical floors are machined with optional lead angles and smart avoidance by tool tilting. This approach provides efficient programming and effective cutting for this family of parts.

NX CMM inspection programming

Inspect more types of parts, including sheet metal components with the new capabilities in NX CMM 9. The results analysis module now offers new options and graphical display of measured features in NX.

Sheet metal inspection

The new CMM programming method for surface edges automates the programming of sheet metal parts. Edge geometry is measured with probes that can touch or scan the part boundaries. This streamlined process provides advanced options to define offset from the edge and to automatically create relative measurements on adjacent faces. These enhancements greatly increase the utility of NX CMM inspection programming for this class of parts.



New functions automate CMM programming of sheet metal edges.

Expanded CMM programming capabilities

New enhancements in NX 9 provide the capability to create inspection paths on multiple features, significantly reducing programming time. New advanced sensor definitions create additional options for measurement strategies. Automatic path sequencing further optimizes the inspection process on the machine. Additionally, NX 9 provides you with new inspection templates, postprocessors and machine models for simplified programming and quick machine deployment.



Reduce programming time by creating inspection paths for multiple features.

CMM data analysis

CMM results can be read back using NX as .mea or .dml files. Inspection results, including status and deviation, can be instantly reviewed using the operation navigator. With NX 9, you can graphically display the measured features in context of the 3D part model. By comparing as-measured results directly in the as-designed

model context, it is easy for you to identify the sources of out-of-tolerance measurements and suggest corrective actions.



Review inspection results in the context of the 3D model.

CAM data management

The Manufacturing Resource Library (MRL) provides an advanced tool library capability based on Teamcenter technology. With the new MRL Connect for NX you can easily connect NX CAM directly to a standalone installation of MRL while operating within the NX environment.

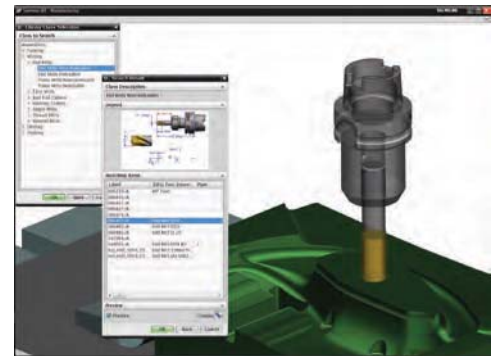
Manufacturing Resource Library

NX 9 CAM can be used to connect to the Manufacturing Resource Library (MRL), a data and process management application built on Teamcenter technology, which manages all of your tooling components and assemblies in a dedicated library. The MRL can be loaded with imported data from tooling vendor catalogs and custom components. These components can then be assembled into complete tools, including the associated 3D solid models used for machine simulation in NX CAM. The administrator of the MRL can access the system to add or edit library content through the MRL interface. NX CAM users can access and search the MRL content to find, select and retrieve tools for use in CNC programming from within the NX user interface.

MRL connect for NX

For shops that use Teamcenter only for their tooling library, this new option allows you to connect directly to a standalone

installation of the MRL without using all the capabilities of Teamcenter. CNC programmers use the regular NX CAM functions for tool retrieval, but are also connected to the MRL's robust tooling library environment, in which they can utilize the powerful search, view and retrieval capabilities across a full classification system.



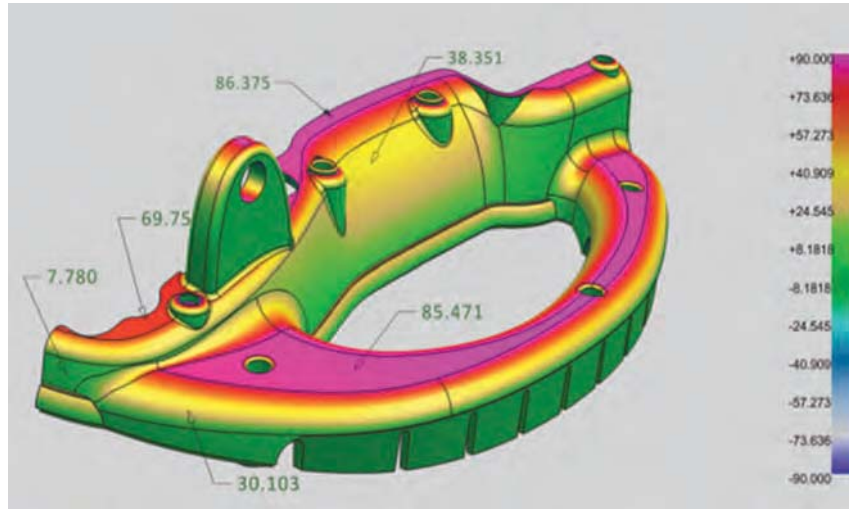
NX CAM is connected to the Manufacturing Resource Library (MRL).

Tooling Design

With the new modeling and analysis capabilities in NX 9 you can reduce the time required to prepare tooling geometry for downstream machining. The new Ejector Pin Table command makes manufacturing document preparation more efficient by enabling you to create template-based drawings of all the ejector pins and sleeves in the mold assembly. The Mold Base Library has been integrated into the NX Reuse Library, making it more efficient and easier to learn. This also adds more functionality, such as the ability to search for specific mold bases and edit them in the assembly navigator or onscreen by right-clicking. Thanks to this integration, it is now easier to access the Mold Base Library during the design phase. The new Object Attribute Management command enables you to easily add, edit or remove attributes on objects in your mold or die assembly. These attributes, which can include face colors, are used for downstream manufacturing instructions. This method is more efficient than the previous way.

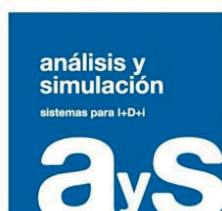
Enhanced stamping and progressive die design workflows enable you to eliminate many manual modeling processes, reducing overall design time. For instance, you can now create or edit a Lower Binder feature in an assembly with several options, including rough sheet metal and multiple closed binder profiles. You can also use templates from the Reuse Library to add details such as ribs or keyways. The new Draw Punch and Draw Die commands let you visually differentiate die faces that require different precisions and tolerances and assign colors to them. This is helpful during die manufacturing. You can now define multiple die tips for any type of stamping operation, so you can process multiple parts at the same time, increasing material utilization and reducing individual part cost.

It is now possible for you to perform sheet metal formability analysis entirely using NX. The Geometry Preparation command allows you to prepare your die, punch, binder and blank geometry, after which the Mesh and Solver command enables you to automatically mesh the geometry and run the LS-Dyna solver. Since you no longer have to move between two different CAD and CAE systems, the process is faster and more efficient, making it easier to verify your design early.



You can now do advanced formability analysis right in NX.

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