

Siemens PLM Software

# What's new in NX 10

Easier and more powerful than ever

## Benefits

### Design

- Perform easy concept design with NX Layout and NX Realize Shape
- Use NX anywhere with a touch-based interface
- Use aerospace-specific functionality for faster design
- Provide full access to Teamcenter information within NX with Active Workspace
- Easily leverage legacy data with Optimize 2D geometry
- Realize easier model diagnosis with face snapshot

## Summary

The latest release of NX™ software continues to improve on the impressive functionality and user-friendliness of previous versions. The most important stage of the design is the early concept phase. NX 10 features tools such as NX Layout and NX Realize Shape™ to make this stage easier and faster than ever. With a new touch-enabled interface, NX 10 brings unprecedented versatility and ease-of-use to advanced design. New multiphysics and composites simulation capabilities make NX CAE 10 a fully capable tool for aircraft engine and airframe engineering. New industry-specific capabilities in NX CAM 10 enable faster programming and better machining quality. More efficient cutting strategies for mold and die machining, streamlined programming and high-quality surface finishes increase your overall manufacturing productivity.

## NX for design productivity

### Concept design

#### NX Layout

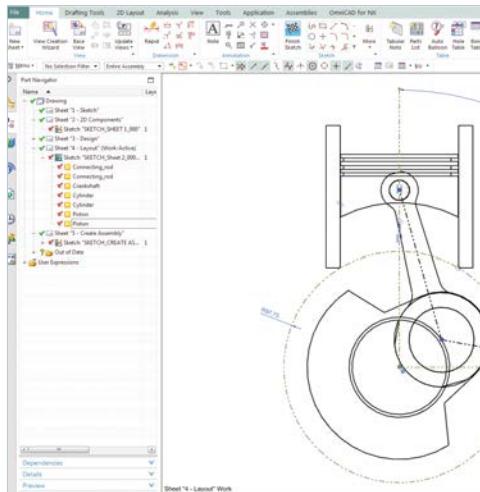
Concept design is easier than ever in NX 10 with NX Layout. NX Layout lets you explore design concepts in 2D for situations in which it's faster and easier to represent design problems. As you'd expect, NX Layout is fully integrated into NX and NX Drafting. This means virtually no learning curve because the interface and commands are the same. It also means that the 2D components you create in NX Layout are organized in the part navigator, and they are parametric, editable and re-usable. You can drag-and-drop 2D components from the Re-use Library to ensure quality and save design time. Finally, anything you create in NX Layout can easily be migrated to 3D to complete your model. The 2D components enter 3D as constrained, parametric sketches that can easily be modified.

# What's new in NX 10

## Simulation

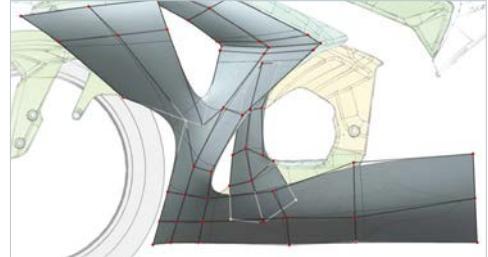
- Streamline thermal structural analyses with new multiphysics environment
- Increase productivity for automotive NVH analysis through integrated vibro-acoustics analysis
- Simulate complex phenomena in composites like delamination and progressive failure
- Simulate fluid flow problems such as the filling or emptying of a tank
- Extract relationships in results data to get a greater understanding of physical behavior
- Leverage Python programming language in NX Open to more easily automate processes

So when you finish your design in 2D, you are already well on your way to a complete 3D model.



Concept design is easy with NX Layout.

**NX Realize Shape** NX Realize Shape is an exciting concept design method that uses advanced subdivision shape creation first introduced in NX 9. It is intuitive: you can create a shape based on a cage that can be extruded, swept, lofted, revolved, copied and more. You can create cage faces from curves or polylines and subdivide them as much as you want to give you greater control with smooth transitions. The end product is high-quality B surfaces in an editable NX feature. This allows for rapid conceptualization of ideas without the need for expert knowledge. NX Realize Shape can be used in combination with or alongside other surfacing and design tools. In NX 10, every aspect of NX Realize Shape has been enhanced to give you more control over your geometry. NX Realize Shape in NX 10 is a complete set of tools for subdivision modeling. When you consider the advantage of full integration with NX, it's the obvious choice for creating complex models quickly and easily.

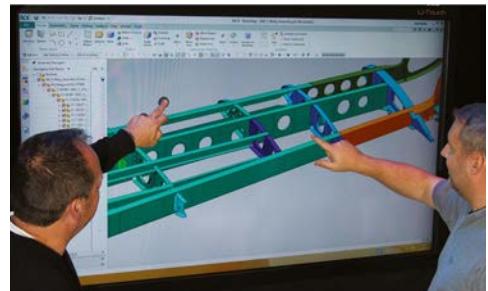


NX Realize Shape makes it easy to create complex shapes.

## Touch

Touch screens can be a quicker, more intuitive way to interact with software. In NX 10, the power of NX is available anywhere thanks to a new optional touch-enabled interface. With NX touch, you have the ability to access the full design capability of NX wherever you are, whether it's on the plant floor, while traveling or just consulting with someone down the hall. Not only does this make it easier to have the information you need wherever you are, it makes you more productive and versatile.

In touch mode, the interface adjusts to make it easier to select items on a touch-screen. The interface is remarkably intuitive and familiar to anyone who has used a mobile device and NX separately. For instance, to zoom you pinch your fingers together; to select something, you tap it, and so on. Given the wide proliferation of touchscreen mobile devices, the average user will be up and running on NX with touch in just a few minutes.



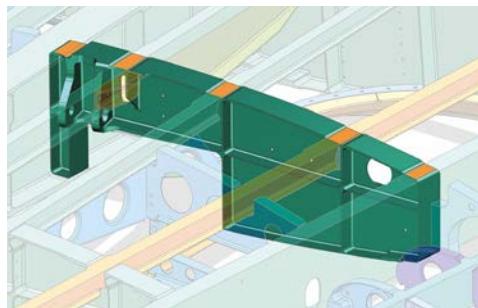
Using NX in touch mode is easy and intuitive.

## Manufacturing

- Machine better quality molds/dies and extend tool life with new cutting techniques
- Improve machining efficiency and reduce model preparation with enhanced 5-axis operations
- Machine higher-quality impeller blades faster with enhanced swarfing
- Automatically create accurate and safe inspection paths with NX CMM
- Produce better quality parts with enhanced CMM results analysis
- Quickly design and visualize production lines with Line Designer
- Simulate manufacturing process faster with tooling enhancements

## Aerospace design

NX 10 is a complete, fully capable aerospace design solution. Commands for creating flanges, ribs, shelves and steps were specifically developed for airframe and aero skin designs; these commands dramatically simplify and accelerate the aerospace design process.



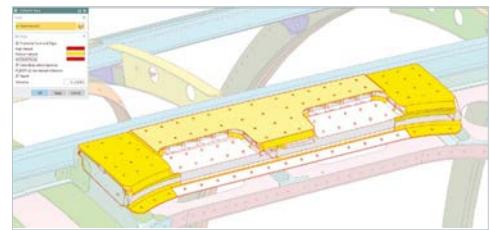
NX 10 features commands specific to aerospace design.

## Synchronous technology

Introduced in NX 9, synchronous technology 2D gives you the same freedom and speed 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. In NX 10, synchronous technology 2D is even more powerful, with T-junction support, point on curve constraints, offset constraints and more. In addition, it's now easier to select the curves you want to modify in 2D sketch. Synchronous technology 2D in NX 10 gives you more ways to modify and use 2D data from any source than ever before. Overall, synchronous technology 2D is up to five times more productive than standard sketching techniques.

Optimize 2D geometry is a new function in NX 10 that cleans up 2D data the way optimize face cleans up 3D data. It gets rid of junk data such as duplicate lines, extra points, tiny pieces of geometry and other extraneous bits that make sketches difficult

to use and modify. The Optimize 2D function can also "planarize" or move the geometry onto one plane. This is especially useful when working with 2D data migrated from another computer-aided design (CAD) system. It's one of many ways that NX enables you to reduce rework and part cleanup time.



Synchronous technology makes non-native data usable.

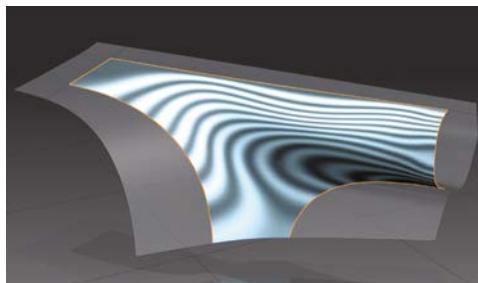
## Part modeling

Model diagnosis can be a time-consuming task when modifying parametric models. The face snapshot command in NX 10 is a powerful tool that saves time by showing you the previous state of any face. This makes it much easier to see any problem areas and fix them.

Patterning has been enhanced to give you greater control and improve performance and efficiency. Creating high-quality blends with more consistent results is easier in NX 10, thanks to tools such as edge blend and face blend. These new blend options let you control every aspect of your blend's geometry and create blends with G2 continuity for finish-quality surfaces used in styling and industrial design.

Creating high-quality surfaces in complex regions is easier than ever. Fill surface enables you to create a surface by picking curves around its boundary with full control over continuity. Curves can now be wrapped or unwrapped across multiple faces. Not only does this increase versatility, it's especially helpful for using manufacturing reference geometry. Trimming and extending sheets and curves has been enhanced to provide more effective results with fewer commands,

increasing efficiency. With new capabilities in the blend corner and fill surface commands, it's easier than ever to create high-quality surfaces that can be manufactured even in complex areas of your model, which was difficult to do in the past.



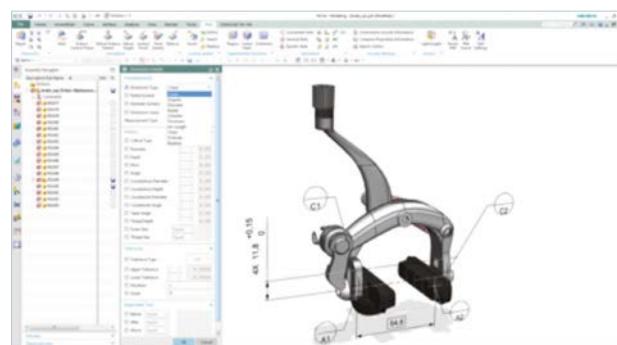
*Fill surface makes it easy to create high-quality surfaces.*

### Documentation

NX Product and Manufacturing Information (PMI) and NX Drafting have many customer-driven enhancements in NX 10. Many of these improvements focus on giving you greater control over the display and placement of your annotation, both on drawings and the model itself. For instance, NX 10 provides a number of drawing format tools to assist in the creation and maintenance of drawing templates. These tools include a title block command as well as intelligent sheet zones. These commands have been enhanced in NX 10 to provide improved workflows.

You can open a JT™ file and automatically create real PMI information from it in NX 10. This includes dimensions, geometric dimensioning and tolerancing (GD&T) objects, weld symbols, centerlines and much more. Since these are real PMI objects, they can be used downstream in a drawing or in manufacturing processes like any other PMI object. With NX 10, you can take advantage of existing information much more quickly than before.

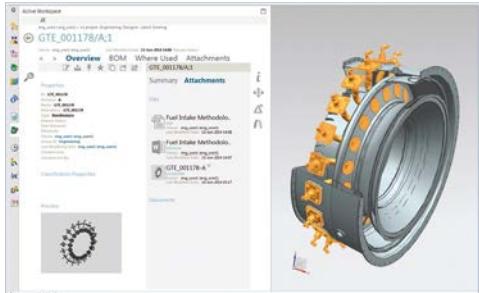
Siemens PLM Software recognizes the cost and effort associated with producing drawings, which is why with each release of NX we focus on delivering tools that support rapid drawing creation with reduced cost and increased productivity. Many customer-driven enhancements to dimension creation and editing have been incorporated into NX 10, as well as a number of annotation enhancements that provide improved compliance with drafting standards. These changes all contribute to simplified workflows and better productivity.



*NX PMI enables you to create dimensions more efficiently.*

### Active Workspace

Active Workspace 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. Active Workspace enables you to quickly access Teamcenter® software data from within NX or anywhere else. The tool reduces the amount of time you spend searching for information with its powerful interactive search and filtering capabilities, which can access external systems, eliminating the need to search in multiple places. Because Active Workspace is context-aware, it only shows the information that is relevant to the task being performed, which saves you time. By highlighting issues in visual reports, it helps you see the big picture so you can make smarter decisions more quickly.



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

## NX for simulation productivity

### NX CAE

NX CAE is a modern simulation environment for modeling; structural, thermal, flow, motion and multiphysics simulation; optimization; simulation data management; and simulation-driven design. NX 10 for simulation introduces new capabilities and enhancements in NX CAE to help you solve the most complex problems faster. Highlights of enhancements in the NX CAE 10 release include:

### Multiphysics

**Multiphysics environment** The new multiphysics environment in NX CAE 10 takes simulation integration to a new level to help you connect two or more solvers to streamline the process of performing complex, multiphysics simulation. This environment delivers a consistent look and feel for performing multiphysics simulations so you can easily build coupled solutions on the same mesh using common element types, properties, boundary conditions, as well as solver controls and options.

This initial release of the multiphysics environment provides the ability to solve thermo-mechanical problems in loosely (one-way) or tightly-coupled (two-way) modes. Coupled thermal-structural analysis enables you to leverage the new NX Nastran® SOL 401 multi-step nonlinear solver and a thermal solution from the NX Thermal solver.

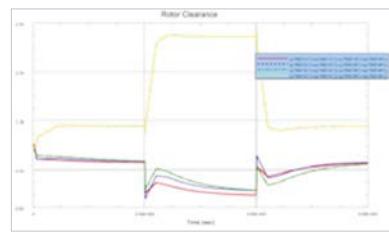
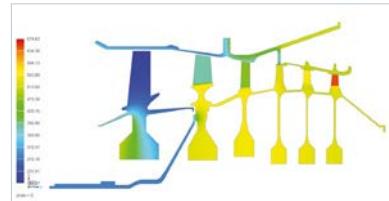
Using NX CAE 10, you can more easily tackle complex simulations, such as blade clearance analysis within aircraft engine systems, or structural analysis of automotive powertrain components within high temperature environments. Other applications for electronic components and metalworking processes are also well suited for the multiphysics environment.

### Simulation modeling and results visualization

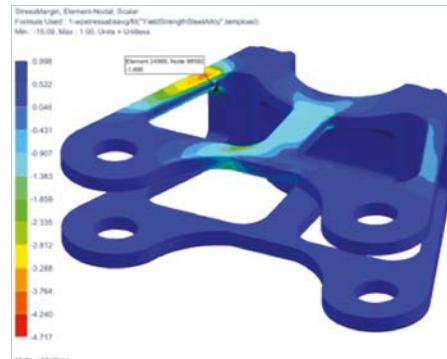
**Updated expression functions, quantities and evaluation system** Some simulation models require highly interdependent loads and boundary conditions (LBCs), some of which are not known until solve time. NX 10 introduces new expression functions, quantities and an evaluation system that simplify the task of defining these interdependent LBCs and can even evaluate expressions at solve time. Expressions can even reference proprietary, user-defined codes, such as thermal subroutines, which make them much easier to manage and use.

In addition, these new expression quantities and functions provide significant benefits in postprocessing operations by leveraging relationships in the data that allow you to develop a greater understanding of the model's physical behavior. For example, you can create user-defined expressions that combine simulation results as a function of time, which you can then plot in NX CAE.

**Adaptive meshing** NX CAE 10 introduces a new adaptive meshing capability for better convergence and accuracy of structural, thermal and multiphysics solutions. Adaptive meshing automates the lengthy and repetitive process of running multiple finite element analyses (FEA) with different mesh densities. The mesh refinement is determined by stress, strain and temperature error estimates associated with element stress and strain discontinuities. Based on these error estimates, the

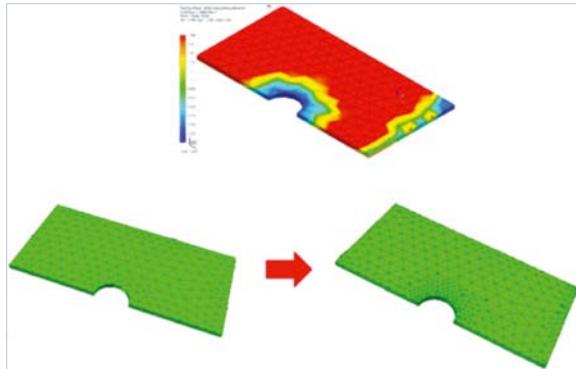


Multiphysics environment used for bidirectional thermal-structural coupling to study blade clearance within an aircraft engine.



Stress margin calculation based on temperature dependent yield stress.

software identifies critical regions on the model that become the target for further refinement of the mesh in subsequent iterations.



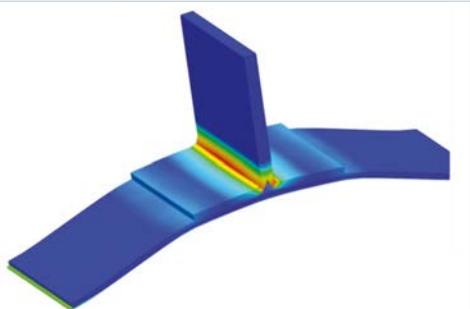
*Adaptive meshing leads to better convergence and accuracy of structural, thermal and multiphysics solutions.*

### Structural analysis

**Extended composites simulation capabilities** NX CAE 10 extends its modeling and simulation capabilities for parts made with laminate composite materials. NX Laminate Composites has enhanced data exchange capabilities with Fibersim™ software so it can be used to import zones onto polygon faces or 2D elements. Additional new enhancements include an interface to CATIA Composite Product Design (CPD), support for the new NX expressions capabilities and the ability to edit multiple laminate physical properties in a single operation. NX Laminate Composites also supports the new NX CAE environment for the LMS Samtech Samcef™ Solver Suite software, which is being introduced in NX 10, and enables you to create cohesive layers between extruded plies to model delamination.

In addition, enhancements in this release for the NX Response Simulation module allow you to recover ply stresses for laminate composites for random vibration analyses.

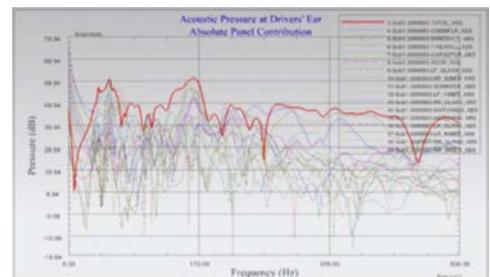
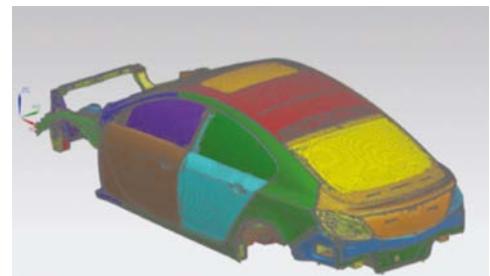
**Integrated vibro-acoustic analysis** New capabilities in NX CAE 10 for simulation modeling and postprocessing mean you



*Simulation of laminate composite material showing delamination.*

can now perform an integrated, end-to-end vibro-acoustic workflow to increase productivity for automotive noise, vibration and harshness (NVH) analysis. In NX CAE 10, you can:

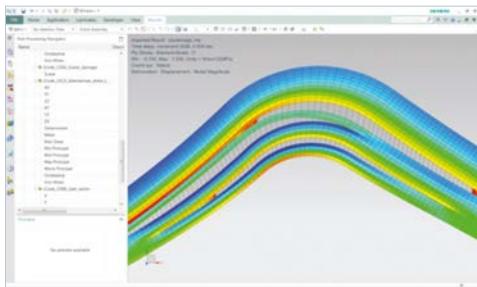
- Easily create the cavity mesh starting from just a structural finite element mesh of the outside structure
- Create load recipes to define loads from external sources, such as measurement data
- Set up and run an NX Nastran vibro-acoustic solution that can include coupling, panels for contribution analysis and absorbent panels
- Postprocess results for sound pressure levels and panel contributions



*An integrated vibro-acoustic workflow increases productivity for automotive NVH analysis.*

**New LMS Samcef Solver Suite environment** NX CAE 10 introduces a new environment for LMS Samcef Solver Suite, which offers a nonlinear finite element solver for structural analysis covering a wide range of applications in the aerospace, defense and automotive industries.

With LMS Samcef Solver Suite, you can run linear and nonlinear static analyses, as well as modal and buckling analyses. The solver includes unique capabilities for the prediction of complex, nonlinear phenomena like progressive damage in the unidirectional and woven fabric plies of a laminated composite structure, as well as delamination with coupling to the damage inside the plies.



The new LMS Samcef Solver Suite environment can be used to simulate complex, nonlinear phenomena in composites in addition to linear static, modal and buckling behaviors.

### Thermal analysis

**Solver-evaluated expressions** NX Thermal and NX Advanced Thermal modules can be used to leverage the new expression capabilities introduced in this release to model interdependencies between boundary conditions. This allows you to define boundary conditions with symbolic expressions containing quantities that can only be evaluated at solve time and updated as needed during the solution sequence.

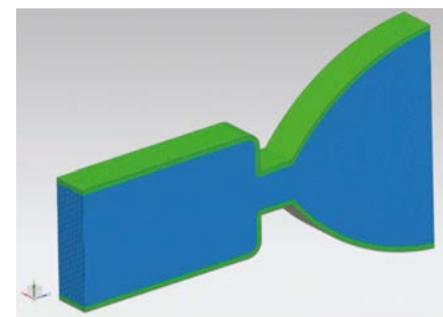
**Printed circuit board components for spacecraft** The NX 10 release of the NX Space System Thermal module includes the functionality to create printed circuit boards (PCBs) and PCB component simulation objects. These new features let you easily define the thermal resistor models for PCBs and electronic components commonly used in spacecraft, which operate under severe thermal conditions over the course of a mission.

**Adaptive time stepping scheme** The new adaptive time stepping scheme introduced in NX CAE 10 handles sharp changes in

temperature at boundary conditions. When there are no abrupt changes in boundary conditions, the new adaptive time scheme accelerates the speed of the simulation without losing accuracy.

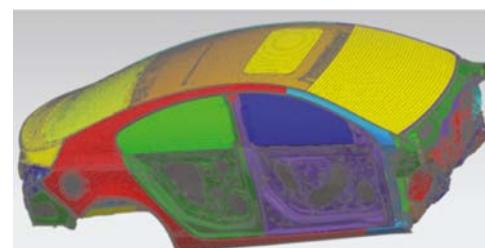
### Flow analysis

**Boundary layer meshing in the FEM** In NX CAE 10, it is possible to create a boundary layer mesh in the finite element model (FEM) file with the NX Advanced Fluid Modeling module. Creating the boundary layer in the FEM file gives you greater control over boundary layer mesh because you can visualize it and use all the mesh controls and quality checks you normally would with any FE mesh. This also makes the boundary layer mesh available for other applications in addition to computational fluid dynamics (CFD), such as for acoustics applications. You can take advantage of the NX CAE geometry and mesh preprocessing speed for use with external CFD solvers by exporting the boundary layer mesh to a CFD general notation system (CGNS) file.

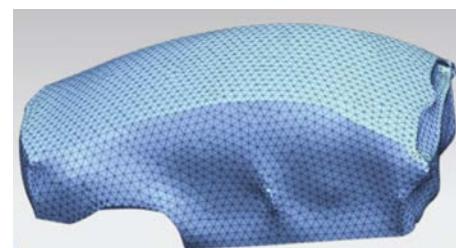


Create and control the boundary layer mesh directly in the FEM for CFD and acoustics applications.

**Mesh wrapping capability** Analysts often need to start their simulations from legacy finite element mesh data, meaning no surface geometry is available. This poses a difficult challenge for CFD analysts who need to create the fluid domain mesh for the cavity inside the part. NX CAE 10 introduces a new mesh wrapping capability that solves this problem by allowing you to generate fluid bodies from models for which you have mesh data but no geometry.

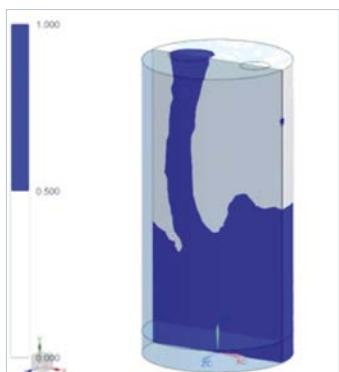


Generate fluid body models when you have mesh data but no geometry for your part or assembly.





*Speed simulation time for even more CFD applications using the parallel flow solver.*



*Simulate open volume enclosures, such as the filling or emptying of a tank.*

**Extensions to parallel flow solver** The NX Flow parallel solver enables successful speeding of large, complex CFD simulations. NX CAE 10 allows you to take advantage of the parallel flow solver for even more CFD applications, such as:

- Two-phase flow
- Homogeneous gas mixture and tracer fluids
- High-speed flows
- Shear stress transport (SST) and K-omega turbulence models
- Non-Newtonian fluids

#### **Two-phase, immiscible fluid simulation enhancements**

With the NX CAE 10 release, enhancements to two-phase immiscible fluid simulation capabilities expand the types of applications you can evaluate. You can now simulate fluid flow problems for a mixture of any two immiscible fluid constituents, meaning you can specify two liquids or gases as long as the two fluids are immiscible. Additionally, you can simulate open volume enclosures, such as the filling or emptying of a tank.

#### **Simulation process automation**

**Python programming language support in NX Open** The Python programming language is added as a language binding for NX Open in NX 10. This allows NX CAE users to record and replay simulation workflows in the Python language. Advanced

users can create or extend NX Open Python journals to develop NX Open applications in Python. These applications can use any features of the Python programming language, including classes, looping constructs, control statements and callbacks. Python journals can be recorded and replayed on Windows and Linux.

#### **NX for manufacturing productivity**

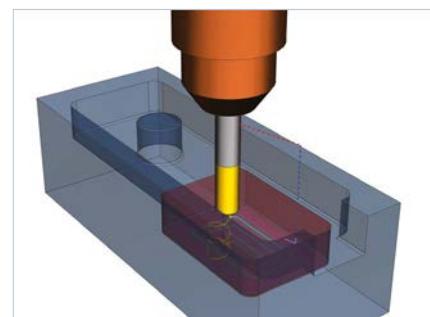
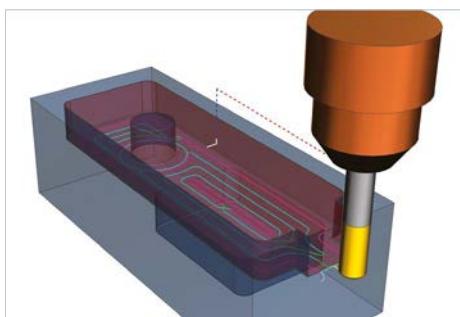
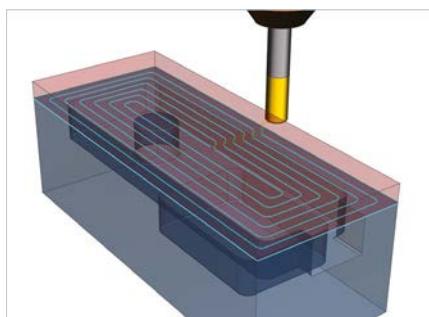
##### **NX CAM**

New industry-specific capabilities in NX CAM 10 help you program faster and machine better quality parts.

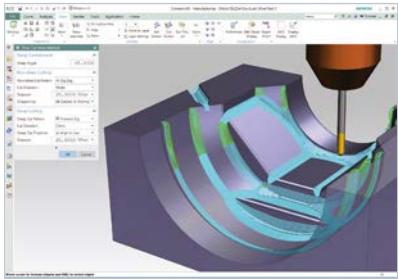
Efficient cutting with adaptive roughing strategies for mold and die machining, streamlined programming of prismatic features and high-quality surface finish of complex parts can increase your overall manufacturing productivity.

##### **Mold and die machining**

**Optimized roughing** You can achieve more consistent tool load, minimize tool wear and extend tool life with the new roughing strategy in NX. This capability is especially useful for more complex parts that require different cutting strategies for different regions. NX applies inward or outward cutting direction, finds the best available start location and uses the right engagement type to enter the material. This automated process is performed region-by-region and level-by-level, ensuring improved cutting conditions in each machined area. For certain types of molds and dies, the programming time can be significantly reduced.

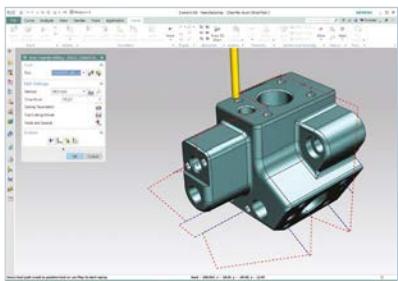


*NX automatically applies best roughing strategies for different regions of the machined part.*

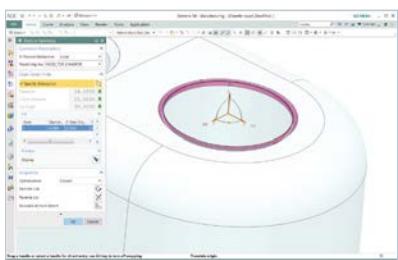


Cut region control enables precise control of rest milling tool paths.

**Cut region control for rest milling** NX 10 provides enhanced rest machining capabilities that offer better tool life and higher quality surface finish. The interactive cut region control functions have been extended to cover Flowcut, the valley rest-milling operation, and specify a range of cut patterns for steep, shallow and flat areas. To ensure that the best cutting method is used for each area, you can preview, change and reorder the regions before generating tool paths. With the new cut region control for Flowcut, you can quickly and reliably program even the most complex mold and dies. The optimized machining of the corners and valleys can extend tool life and improve surface finish.



Shown is the automated chamfer milling of holes with different orientations and sizes.



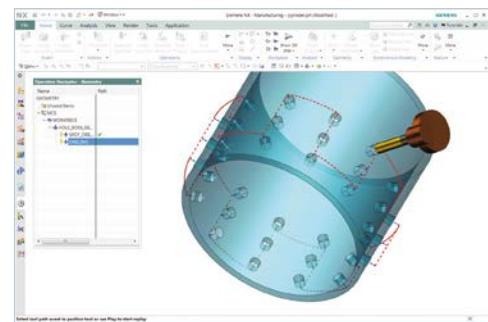
In-process visualization displays uncut material for reliable programming.

### Prismatic part machining

**Chamfer milling of holes** The new chamfering operation applies the powerful hole-milling approach to a common feature with a minimum programming input. NX 10 enables you to automatically calculate the correct tool offset for chamfering holes and generating a circular milling tool path using the familiar hole-making methodology. You can reduce programming and machining time by chamfering multiple holes with one operation, even when the holes have different orientations. And you can use a single tool to machine chamfers of different sizes that save you even more programming time. The in-process visualization displays uncut material after each operation.

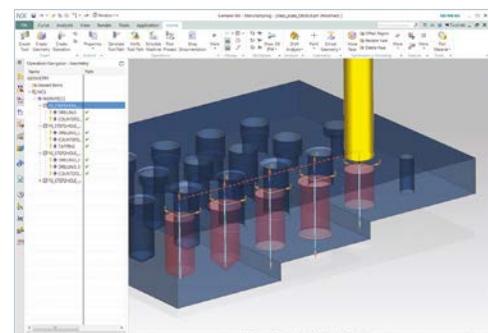
**Efficient drilling paths** Minimizing the travel and improving the machining accuracy can be critical when drilling a large number of holes. The new optimized sequencing in NX enables you to improve drilling operations by specifying the desired drilling pattern and selecting the best start position. The subsequent operation

can start where the previous one ended, reversing the cutting direction, thus further maximizing performance of the machine tool.



Minimize tool travel and improve machining accuracy with optimized drilling patterns.

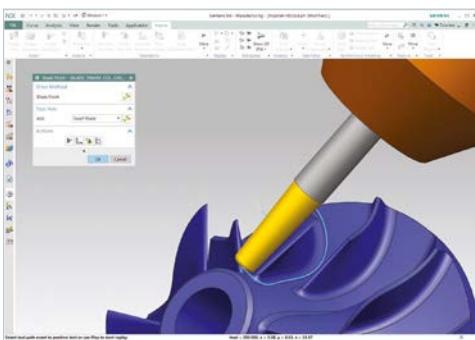
**Group features** With NX 10, you can minimize the number of operations needed to machine parts with many holes. Using feature-based machining, holes are identified, filtered and grouped automatically. Holes sharing similar attributes are programmed together so that they share tools. On the shop floor, the optimized machining process can reduce tool changes, shorten travel distance and increase drilling accuracy.



Flexible grouping capabilities enable faster programming and more efficient machining of similar holes.

## Complex part machining

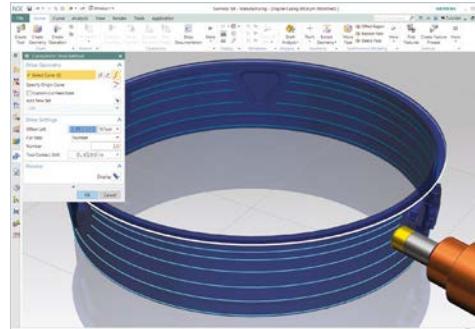
**Swarf cutting of blades** Exact alignment of the tool with the blade geometry can be achieved with the enhanced swarfing in the NX Turbomachinery Milling module. Complex blades can be finished with a single pass using the entire length of the tool, which can produce a high-quality surface finish. The gradual engagement of the tool with the material ensures machining with reduced vibrations that further improve the surface quality. Flat, bullnose and tapered end mills can be used to generate a swarfing tool path that machines the entire blade with a single pass.



Machining high-quality impellers using a swarf operation enables precise alignment of the tool with blade geometry.

**5-axis machining of rotary parts** The enhanced strategies in NX 10 make it simple to create efficient 5-axis machining operations for large rotary parts, such as aircraft engine casings. Select a single edge and NX will generate all the necessary operations to machine cylindrical faces. The lateral tool offset option enables you to easily create accurate, collision-free 5-axis multiple passes. For some rotary parts, this method can save you hours of geometry creation and model preparation.

Also, 5-axis chamfering and deburring operations can be easily generated using this cutting strategy.



Simplified programming of complex rotary parts lets you quickly generate 5-axis tool paths.

This style of rotary cutting performs best when the tool engages material on its leading edge. NX provides control over the contact point and offers options for tool shifts in order to optimize the cutting conditions. This results in better material removal rates, tool life and surface finish.

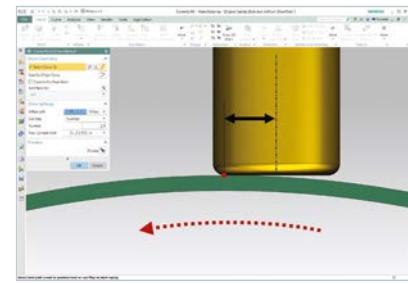
## NX CMM inspection programming

The automated inspection programming capabilities in NX CMM 10 lets you create accurate and safe inspection paths. The results analysis module enables you to import and compare multiple results sets to improve the quality control process.

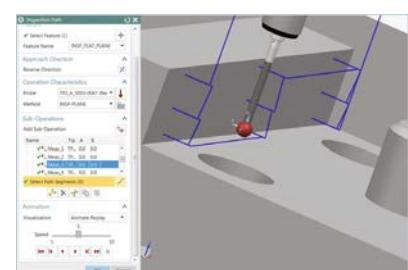
## Automatic collision avoidance enhancements

NX CMM makes it easy to program measurement sequences across faces and features, creating safe transfer moves. The system monitors each motion for possible interferences, then adjusts the approach vector or tool axis as needed to keep the program collision-free.

NX CMM 10 expands these automated collision avoidance methods to include movement of the measurement point. For example, a point positioned too close to another surface will be automatically moved far enough to avoid any interference.



Accurate control of the contact point can optimize cutting conditions.

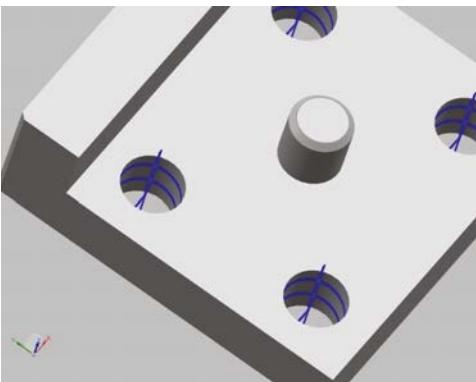


NX CMM automatically adjusts inspection paths to prevent interference with adjacent part features.

Another new option in NX 10 enables you to remove measurement points that cause interference conditions. Arrays and scanning patterns can be applied quickly; then only the points that can actually be measured safely are kept for the final program.

### Scanning paths automated with link to PMI

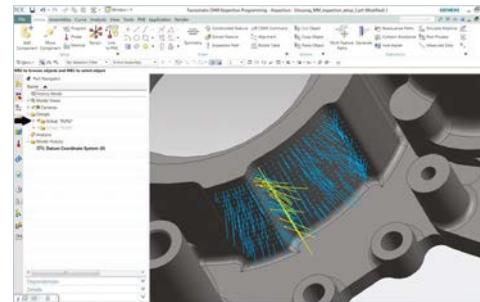
The NX CMM link to PMI uses the solid model's product manufacturing information to automatically program measurements against the provided positioning and tolerance information. The programming automation of the link to PMI has been enhanced to provide scanning operations in addition to touch points. This greatly increases application opportunities for this powerful automation method because scanning measurements are being used more frequently.



Scanning inspection paths can be automatically created by using the embedded PMI data.

### Saved analysis results

NX CMM can be used to analyze measured results by bringing them back into the graphics system to compare against the as-modeled geometry. In NX CMM 10, these measured results can be stored in the NX part file for further analysis. NX measurement tools can be used to display deviations and create annotations for reports. Subsequent measurements can be compared against previous measurements as manufacturing processes are refined and quality is improved over time.



NX CMM lets you save and analyze multiple results sets for improved quality control.

### Machine-specific output

NX CMM uses the same proven, flexible postprocessor as NX CAM in order to provide production-ready output to the wide variety of measurement machines found in industry. With NX CMM 10, inspection programs may include user-defined events (UDEs) to make postprocessors even more flexible and provide machine-specific features. Several UDEs are provided as a starting point, but users can create UDEs for their specific application.

### Line Designer

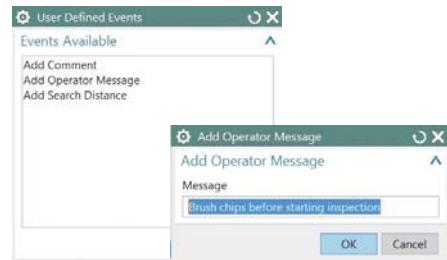
#### Design complete production layouts on a single platform

NX 10 introduces Line Designer, an advanced solution to design and visualize layouts of product lines. The integrated Siemens PLM Software platform enables you to easily associate the designed layout to manufacturing planning.

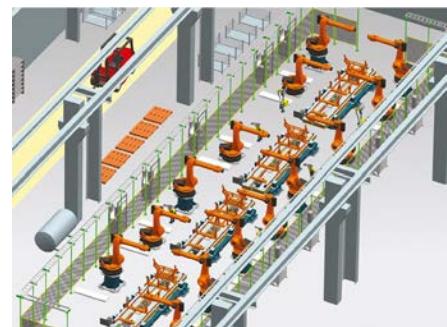
This close integration with planning allows you to efficiently manage the entire manufacturing process. You can easily optimize the process by specifying each production step down to managing a single manufacturing resource, such as a robot or a fixture.

You can perform accurate impact analysis and drive efficient change management by using the parametric resources that are associated with the manufacturing plan.

Having a complete solution for line-level design that is integrated with manufacturing planning is essential to define optimized production processes.



UDEs help you generate inspection programs with application-specific instructions.

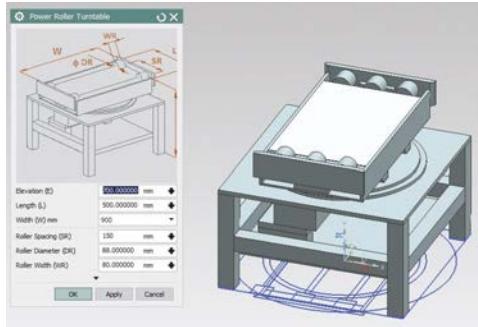


Design and visualize layouts of production lines with Line Designer.

## Lay out the production concept in NX

Line Designer is a complete production layout solution for manufacturing engineers.

The parametric engine in NX enables you to efficiently work manufacturing components and to easily accommodate any changes. When adjusting the size of individual components or modifying the layout, the entire production line automatically updates.



Define smart components with NX parametric modeling.

For each phase of the layout design, you can use the right digital representation of the manufacturing components:



**2D representation:** quickly position the components and generate drawings.



**Simplified representation:** design the 3D layout with a minimum number of components.



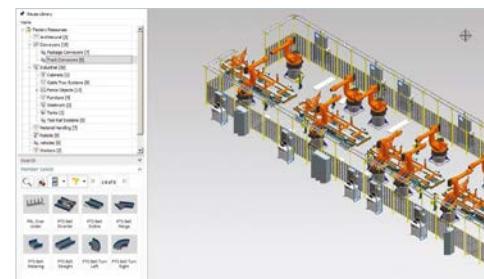
**Detailed representation:** simulate and detect interferences by using all the equipment's details.

The fully classified equipment library is managed by using Teamcenter, a complete system for data and process management. The solution connectivity allows manufacturing engineers to connect to the library. By connecting to the Teamcenter library environment, you can utilize the powerful search, view and retrieval capabilities across a fully classified library directly from NX.

To efficiently handle a large amount of complex data, NX provides advanced technology such as fourth-generation design (4GD) and JT. The component-based 4GD approach enables concurrent design in multiple configurations, and is scalable to layouts with a large number of components. JT is a lightweight data technology that provides high-performance visualization and collaboration capabilities.

## Define the complete production process in one integrated solution

Siemens PLM Software provides a unified platform for product, tool and production system design. It supports the entire workflow – from product and line design to virtual commissioning.



Share the same library equipment with the Siemens PLM Software suite of applications.



Pictured is the factory layout design and manufacturing planning workflow.

The layout designed with Line Designer can be used to validate the manufacturing process using Tecnomatix® software for digital manufacturing. Using the Process Simulate application, you can validate a wide range of robotic applications allowing you to simulate complete production systems, including cell validation and robot placement optimization. By simulating production processes, you can define the optimum manufacturing process.

With Tecnomatix virtual commissioning solutions, you can correct your programmable logic controller (PLC) codes in a virtual environment before using them on real equipment. By simulating and validating your automation equipment virtually, you can ensure proper operation and significantly reduce system startup time.



*Validate the designed production lines with Process Simulate.*

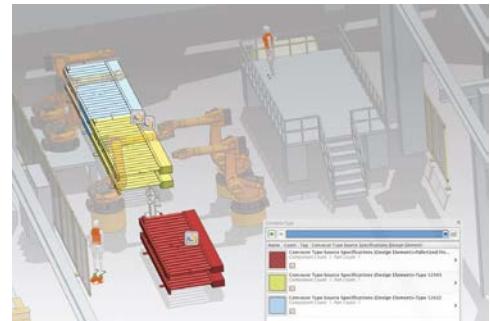
Using the Siemens PLM Software unified platform provides efficient change management and direct access to a shared library of manufacturing assets. Re-usable best practices can be synchronized across the solution. You can further expand the solution to efficiently work with suppliers and system integrators.

#### **Visual reporting and documentation**

You can use Line Designer to directly access a layout's PLM information in Teamcenter. Line Designer can display relevant information about each component, including type, design changes, suppliers, investment cost and build dates.

With high-definition 3D (HD3D) NX Visual Reporting, you can browse product lifecycle management (PLM) data and view details in an interactive navigator. Visual reports can be configured to display color-coded information on manufacturing equipment models based on its values and properties. So you can quickly and intuitively visualize components in make-or-buy categories, identify long-lead items or identify all suppliers of a full line of equipment.

Visual reports can be managed and distributed to benefit the entire enterprise.



*Directly access and display the component properties using NX Visual Reporting.*

#### **Tooling**

Tooling design in NX 10 contains a large number of incremental changes designed to make it easier to simulate your manufacturing processes.

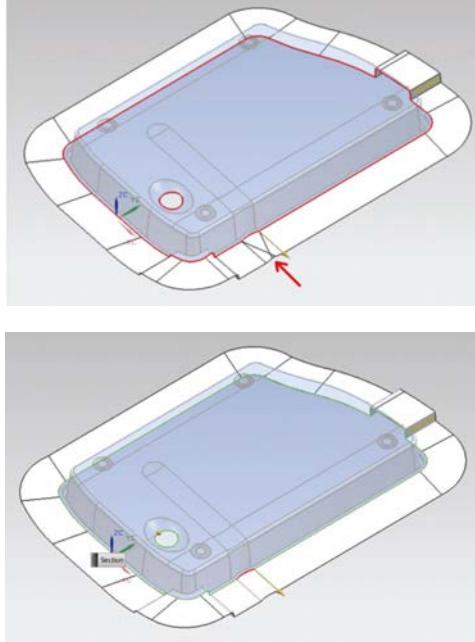
The create box command is used to create a block or cylinder around a group of objects that you select. In NX 10, this command creates associative features for easy tracking and modification. You have full control over the creation method, shape and references of the box. These enhancements automate a significant portion of the mold design process and simplify workflows.

It's quicker and easier to find the standard parts you need in NX 10 because you can define a search-and-save-it in the Re-use Library. You can even place the search directly on the NX ribbon bar for quick access.

It is simpler to create parting surfaces with the new guided extension option. This lets you create an associative parting surface based on a set of connected curves or edges. With guided extension, you can create parting surfaces in areas where other methods cannot.

Progressive die design has a large number of customer-driven enhancements in NX 10. Functionality improvements in the areas of prebending and unbending, shim design, burring and pad creation help simulate actual manufacturing processes as well as provide methods for correcting die issues. Together they make it easier to create large progressive die shapes.

You have more control over tool motion simulation than ever in NX 10. Kinematic models can be easily changed, as can control data. There is support for more cam types and lifters as well. You can also create user-defined motions such as linear movement along a vector, or angular movement along an axis. With these options, you can validate your mold and die assemblies more quickly and accurately.



*Guided extension simplifies the creation of parting lines.*

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