

# What's new in NX

## Achieve cutting-edge innovation with next-generation tools

### Benefits

- Achieve cutting-edge innovation with tools that are the next step in design evolution
- Create better designs faster by naturally capturing ideas as you iterate
- Integrate the latest communication tools across your teams to enhance design understanding and decrease time-to-market
- Create designs quickly and with confidence

### Summary

Building on its legacy of best-in-class customer deployment readiness and data preservation, Siemens Digital Industries Software has enhanced NX™ software, the premiere CAD solution within Siemens' Xcelerator offering, to deliver the next generation of design, simulation and manufacturing solutions. The latest release of NX introduces many enhancements that remove the barriers to innovation and make you more efficient. Our tools empower your organization to meet high demands, innovate your designs and get products to market with exceptional speed. The latest updates allow you to capture ideas naturally as you iterate designs, eliminate errors and challenges early in the design process, collaborate powerfully within and across teams and push your designs beyond what was previously considered possible.

### What's new in NX design

#### Tools to enhance your innovation

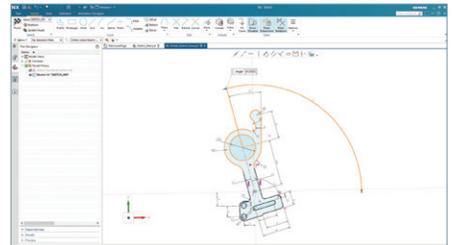
##### Sketching

NX 1926 features a significant leap forward in sketching that can deliver up to a 30 percent reduction in the time you spend capturing design ideas during the preliminary sketching phase. The enhancements that enable this include:

- NX sketching anticipates what you are going to drag or modify

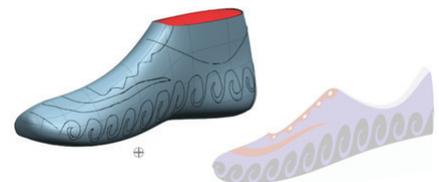
- NX identifies geometric relationships for you based on surrounding geometry
- Like the broader NX design ecosystem, NX Sketch works seamlessly with data regardless of the source

The new approach to sketching allows users to capture ideas as they iterate and make changes as needed, instead of approaching computer-aided design (CAD) after the initial iterations have been sketched out by hand.



##### NX Draw Shape

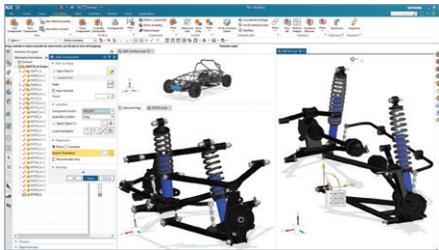
NX Draw Shape is a new add-on module that allows freehand drawing of wireframe shapes on bodies. Designers can draw on planar and freeform geometry. This capability enables users to visually convey and communicate key aspects of designs prior to moving into 3D and enhances NX to support tasks typically performed in niche software today.



# What's new in NX

## Assembly design

NX assembly design has enhanced several common operations to improve user efficiency and generally simplify assembly creation. The newly implemented selection and gesture capabilities include an update to the assembly preference dialog. A "find closest" orientation simplifies creation of touch align constraints.



## Digital mockup and markup

Several new capabilities aimed at enhancing user workflows have been added to digital mockup (DMU). This release features dedicated DMU preferences and customer defaults, as well as new measure and section snapshot properties. NX Viewer now supports DMU and markup capabilities.

## NX Structure Designer

NX Structure Designer is a brand new tool in this release of NX. It includes structural frame design capabilities that help accelerate the design process and reduce time-to-market, including a rich industry standards library and re-usable frames. NX Structure Designer includes:

- Easy-to-use structural frame modeling capability
- Rapid frame drawing assistant that creates 2D skeletons with minimal clicks
- Automatically oriented 3D members placement based on user-defined rules
- Rules-driven and automatically added corner treatments

- Specialty features such as structural corner stiffeners, end caps and mounting feet to create accurate designs



## NX Join

NX Join is a brand new add-on module in this release. It reduces the time required to place and define fasteners and hardware and improves the quality of fastener assembly designs. Integration with other applications ensures a complete solution.

Each join feature is classified within three primary classes of joins: point, line and face. In this release the subclasses are all of the point type and include rivets, fasteners, adhesives, and spot welds.

Benefits of NX Join include:

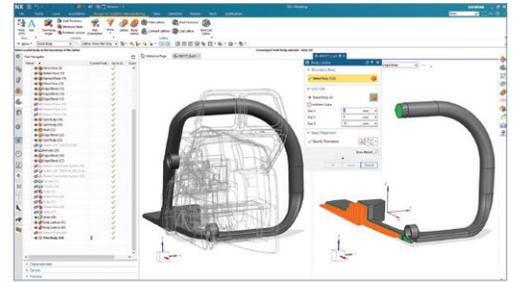
- Features appear in the part navigator
- Information and properties are available from the right mouse button
- Custom attributes can be configured as needed
- Defining join features in the NX model helps capture fastener information in a form that can be used throughout the product lifecycle

## Design for additive manufacturing

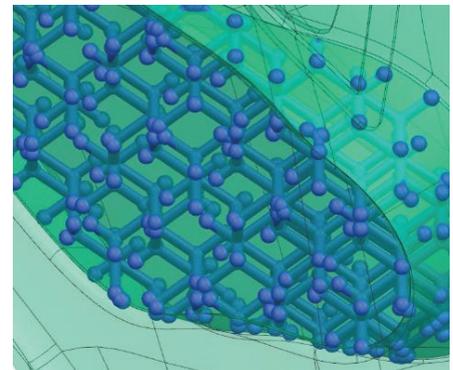
### Lattice

A new body lattice command enables designers to use any CAD body as a unit cell so that any analytic or freeform shape is now possible, which greatly

expands the engineering possibilities of lattices. Additionally, 2.5D lattices are now possible supporting various in-fill requirements.



The lattice tool features several workflow and command enhancements and enables users to place balls at the lattice nodes to further improve lattice durability. The tessellation factor is maintained between editing operations for improved lattice quality.



## 3MF export

Updates to 3MF file format export enable users to output body and face colors and visualize material textures. This capability is aimed at improving collaboration among the community of additive manufacturing original equipment manufacturers (OEMs), suppliers and services bureaus. Manufacturing slicing performance is significantly improved using the lattice graph directly from 3MF.

## Collaborative design and management

### Maximize design understanding with integrated communication

#### Model-based definition

Model-based definition offers several workflow and functional improvements focused on improving design and collaboration efficiency, including:

- Greater support and automation for publishing technical data packages from NX
- New capabilities for comparing product and manufacturing information (PMI)
- Dimension workflow improvements
- Additional standards coverage for creating compliant models

This release features increased support and automation for publishing technical data packages. Users can now multi-select attachments in a single operation and automatically open Adobe Acrobat Reader to view newly published technical data packages, making it easier than ever to communicate within and across teams.

#### PMI compare

The addition of new PMI object properties for enhanced workflows and improved efficiency dramatically reduces PMI and model view checking time by enabling you to quickly and easily identify changes. The updates to PMI compare include:

- Feature control frame tolerance modifier comparison and reporting
- Additional detailed comparisons for hole and thread callout properties
- Custom symbol properties comparison and reporting

#### Teamcenter X

The Teamcenter® portfolio now includes Teamcenter X, a software-as-a-service (SaaS) offering. Users of NX can leverage a cloud-based, instant-on, pre-configured product lifecycle management (PLM) solution that delivers fast return on investment and grows with

you as your business grows. With managed services, Siemens has experts in place with years of PLM experience to operate Teamcenter in the cloud, regardless of customer size.

#### Teamcenter integration

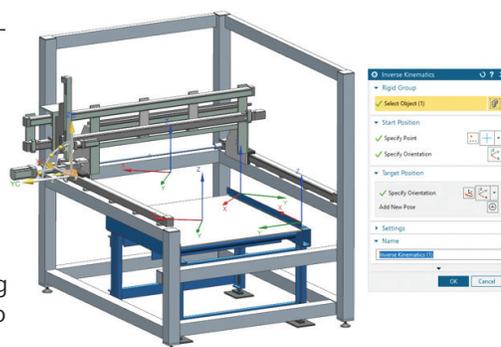
Teamcenter features a further workflow integration update with intelligent part number creation that assigns a unique number to new items created in NX, allowing for better documentation and property tracking.

## Integrated simulation and manufacturing

### Have higher confidence in your designs than ever before

#### NX Animation Designer: enhanced inverse kinematics

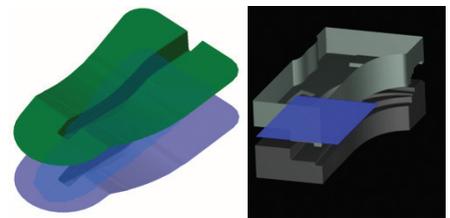
NX Animation Designer has enhanced inverse kinematics that enables users to set multiple travel points between start and end in order to simulate designs in motion. Users can control location and rotations, edit the results based on their needs, independently tune motor speeds as needed and achieve results almost instantly.



#### Fibersim

This release introduces a forming capability that is focused on helping companies design parts for forming. The capability can be used for refining part and tool shapes to make parts more viable for forming through quick feedback within the traditional CAD design space, similar to the classic producibility simulation of the Fibersim™ portfolio for hand-laid parts.

With this release, Fibersim includes a topology check for the design checker capability. Because Fibersim relies on on-surface geometric calculations, it is imperative that the laminate surface is of high quality and free of defects. With the topology checker, users can quickly assess the topological validity of a surface and graphically identify features that yield invalid topology and prevent proper operation of core Fibersim functions.



#### What's new in NX for manufacturing

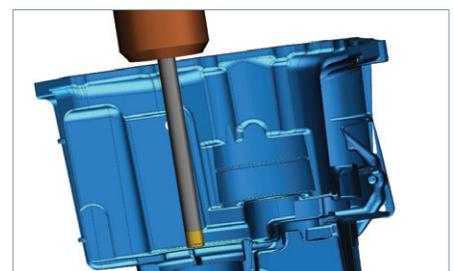
New capabilities in NX enable faster computer numerical control (CNC) programming, higher machining efficiency and streamlined 3D printing job preparation. The improved coordinate measuring machine (CMM) inspection programming further automates the creation of inspection paths.

#### NX CAM

NX CAM software provides comprehensive CNC programming capabilities for a wide range of jobs – from 2.5-axis milling and turning to high-speed machining and multi-axis milling and robotic operations.

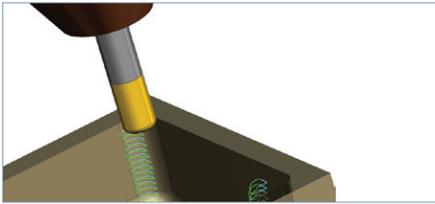
#### Report shortest

The "report shortest" tool streamlines programming of parts with deep features, ensuring collision-free milling operations. It helps you use shorter tools to minimize tool deflections resulting in improved surface quality.



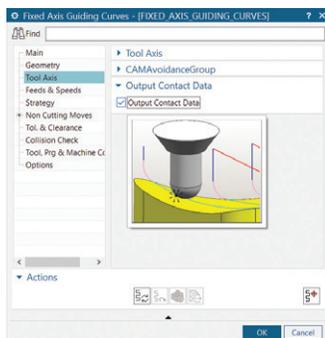
### Smooth transitions

Smooth transitions help you generate optimized transitions between rest milling cutting passes, enabling use of higher feed rates and improved machining conditions to reduce machining time.



### 3-axis guiding curves

The 3-axis guiding curves advanced finishing method can now calculate tool-paths with contact points significantly faster, shortening the programming time of complex parts.

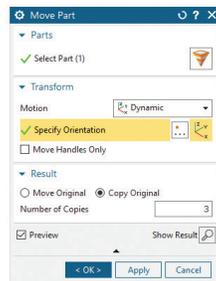
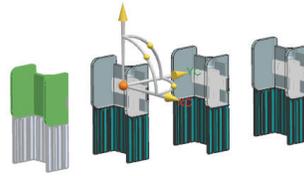


### NX additive manufacturing

NX additive manufacturing offers comprehensive capabilities, from design to print to post-print validation, in a single integrated system.

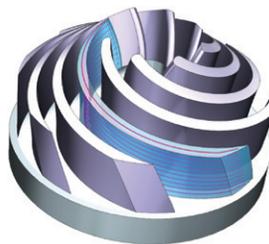
### Copy and paste of parts and supports

Copy and paste of parts and supports enables quick positioning of multiple parts with their support structures in the build tray.



### Additive multi-track buildup

Additive multi-track buildup is a new operation that streamlines the programming of multi-axis parts with near-constant wall thickness. By using a constant number of additive beads, this new capability ensures a consistent thickness and height.



### Apply rules

The apply rules function enables insertion of defined events into a multi-axis additive operation based on time or distance. This capability allows for different multi-axis additive technologies to insert rules or commands specific to their unique requirements.

### Tilt tool axis on finish passes

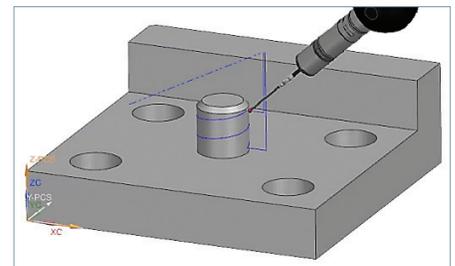
Tilt tool axis on finish passes allows for proven NX multi-axis functionality to control the 5-axis movement of finish paths within operations. This enables complete collision checking, as well as smooth multi-axis output, allowing for a higher quality output on the additive machine.

### NX CMM Inspection Programming

NX CMM Inspection Programming software gives you advanced capabilities that help you automate programming and use the latest inspection methods.

### Boundary scan curve on plane

Boundary scan curve on plane automatically generates inspection paths for tactile scanning, enabling faster and more accurate part inspection. By eliminating geometry creation, such as that used by traditional CMM inspection systems, it standardizes the process and speeds up inspection programming.



### Path order optimization

Path order optimization helps you output the most time-efficient inspection program with a single click. It optimizes the program for probe changes and head angles, and minimizes the distance between paths.

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