What’s New in NX CAD

What’s new in NX 1847 and NX 1872 CAD

By Devin Croswell

10/16/2019
Who I am/What I do:

- **Sr. Mechanical Design Engineer** for Sherpa Design, Inc.
- Over **14 years** of CAD and Engineering
- Multi-CAD User
  - **NX**
  - **SolidWorks**
  - **Creo**
  - **Catia**
- I love to play basketball and go camping
- Raising 3 kids, with my wife, in Washougal, WA
Overview

Topics

Fundamentals
• First Assist, Visual Assistant
• NX VR Enhancements
• Faster Rendering in Ray Traced Studio
• NVIDIA MDL Materials

Modeling, Assemblies
• Shadow Curves
• Snapshots to Review In-context Design Changes.
• Assembly Load Performance
• New Assembly Constraints
• Automatic Work Part Change
• Enhancements –
  o Thicken, Styled Corner, Fill Hole, Virtual Curve

Design for Additive Manufacturing
• Texture Mapping
• Unit Cell Editor
• Filter Lattice
• Added Tetrahedron & Surface Lattice
• Morph Mesh

Aero Design Features
• Aero Flange
• Aero Rib
• Aero Step
• Aero Shelf
• Blend Pocket
What is NX Continuous Release?

Continuous Release:

• Regular **easy steps**
• Same **low effort** for an enhancement release as a maintenance release
• **No distinction** between types of releases
• **No Teamcenter upgrade** necessary for supported Teamcenter versions when upgrading NX
• Deploying productivity enhancements up to six times faster
NX Continuous Release

The Value of Continuous Release

• Receive functional enhancements faster allowing you to boost your productivity

• Consistent schedule for updates allow you to better plan adoption of new technologies

• Increased responsiveness to new ideas and trends

• Reduced deployment cost for incremental updates

• Faster, more consistent response from NX development vs. release dependent hotfixes

• Continued focus on release quality and deployment
• Only 1 active main channel to be supported in Continuous Releases
• Functional enhancement releases every ~6 months (New functionality, enhancements, fixes)
• Update releases every month (fixes)
• Beta and EAP events prior to every functional enhancement release
• Early reports, NXOpen API freeze and Pre-Release can be accessed around Beta and EAP time frame
• Removed focus from versioning, each new release is purely “NX”
Fundamentals

- First Assist, Visual Assistant
- NX VR Enhancements
- Faster rendering in Ray Traced Studio
- NVIDIA MDL Materials
First Assist

Capability

• F1 for help – start seeing initial help topics for select commands in the NX embedded browser

• The Web Browser is available from the Resource bar, and you can double-click its tab to undock it

Value

• Learn more effectively from visual content and videos

• See examples of new or unfamiliar commands

• Link from the navigation bar to additional help topics
Visual Assistant

**Capability**
- Displays thumbnails and descriptions for different scenarios
- Displays a sample model in the viewer
- The viewer allows you to see highlighted options on an object as you zoom, pan and rotate the sample model

**Value**
- Visual Assistant allows for multiple scenarios of sample models with which you can interact, and these can be used to interactively visualize the objects and parameters that you must specify to use the dialog box.
Visual Assistant

Commands that support Visual Assistant:
- Aero Flange
- Aero Rib
- Aero Shelf
- Aero Step
- Contour
- Flange
- Draft
- Draft Body

Additional tools:
- Joggle
- Law Extension
- Through Curves
- Section Surface
- Swept
- Variable Offset Face
NX VR Enhancements

The new Virtual Reality Design Review product via the command Go VR

- Integrated and immersive 3D environment with your models and assemblies for inspection and design reviews

- Inspect and review designs in full scale with a direct connection to NX; no data export or translation needed

- UI has been specifically designed to be used in 3D:
  - Navigation and viewing controls
  - Inspection and review tools
NX VR Enhancements

Recent enhancements in continuous release include:

• New preferences and customer defaults for the NX VR environment that let you specify the up vector, colors, navigation, and performance levels

• New device support for Windows Mixed Reality

• New command to update the VR display based on geometry changes made in the main graphics window
Faster Rendering in Ray Traced Studio

Denoiser removes visual noise from a Ray Traced Studio generated image.

Denoiser supports all render modes: Fast Interactive, Quality Interactive, Photoreal, and Static Image.

Denoiser optimization allows for faster rendering.
NVIDIA MDL Materials

Import material definition language (MDL) materials directly into the **Studio Materials in Part** palette.

Edit and customize the studio materials, and then export them in MDL or XML format.
Modeling & Assemblies

- Assembly Load Performance
- Snapshots to review in-context design changes
- New Assembly Constraints
  - Joints: Hinge, Slider, Cylindrical Ball
  - Couplers: Gear, Rack & Pinion, Cable
- Automatic Work Part Change
- Shadow Curves
- Enhancements – Thicken, Styled Corner, Fill Hole, Virtual Curve
Assembly Load Performance

Capability
• Minimally Load – Lightweight Display:
  • The option loads less data than Partially Load – Lightweight Display
  • The assembly is displayed early in the load process so you can set up your scene while your components load

Value
• Less load time than other load options
• Helpful for when you work with large assemblies in design, visualization, packaging, and drafting workflows
Snapshot

Capability

- **Snapshot** creates a copy of a body for visual comparison of design changes
- Snapshot window enables comparing the model side by side with the option to synchronize view manipulations
- Snapshot can be overlaid on top of the updated geometry and inspected using a translucency slider

Value

- Visual comparison of a design change greatly improves understanding of the impact of change
New Assembly Constraints

Use joints to constrain two assembly components so that their range of motion is confined to the desired directions and limits.

There are four joint types: hinge, slider, cylindrical, and ball.

You can also do the following for joints:

- Specify whether a joint is driving or driven.
- Set a value for a driving distance or driving angle.
- Set upper and lower limits for distance or angle values.

Joints are shown with other assembly constraints in the Assembly Navigator and the Constraint Navigator.

You cannot convert a joint from one type to another.
New Assembly Constraints

Hinge

A hinge joint between two bodies allows one rotational degree of freedom along an axis.

A hinge joint does not allow translational movement in any direction between the two bodies.

You can set an angle value and limits for a hinge joint.
New Assembly Constraints

A slider joint allows one translational degree of freedom between two bodies along a vector.

A sliding joint does not allow the bodies to rotate with respect to each other.

You can set a distance value and limits for a slider joint.
New Assembly Constraints

A **cylindrical joint** between two bodies allows two degrees of freedom: one translational and one rotational.

With a cylindrical joint, the two bodies are free to rotate and translate relative to each other about and along a vector.

You can set distance and angle values, as well as limits, for a cylindrical joint.
New Assembly Constraints

A ball joint between two bodies allows three rotational degrees of freedom

You can set an angle value for a ball joint, but not angle limits
New Assembly Constraints

A **coupler** is a type of constraint that defines the relative motion of two joints.

You can use couplers to more easily define complex relative motion between two components.

In the **Assemblies** application, you can create three types of couplers:

- **Gear**
  - The angle values of two joints are coupled by a ratio that you specify and an angle offset calculated by NX.
  - The ratio affects how much the second gear moves in relation to the first.
New Assembly Constraints

The angle value of a hinge, ball, or cylindrical joint is coupled with the distance value of a slider or cylindrical joint using a displacement radius and a linear offset calculated by NX.

The displacement radius is the radius where the pinion contacts the rack, which is consistent with Animation Designer.

The distance values of two joints are coupled using a ratio that you specify and a linear offset calculated by NX.

You can couple slider joints or cylindrical joints, or one of each type.
Automatic Work Part Change

Capability
• Work part is automatically set based on selected geometry

Example workflow
• Current Work Part is top level product assembly
• Enter a command, e.g. Chamfer
• Selection Scope is set to Entire Assembly
• Can select edges from any component in the assembly to chamfer
• Once edge is selected, that component automatically becomes the work part

Value
• Speeds design in assembly context workflows
• Removes the need to explicitly set the work part
• Reduces RMB interactions
Shadow Curves

Use the **Shadow Curve** command to extract silhouette curves from bodies, or to project shadow curves of bodies to other geometry.

The silhouette curves and shadow outlines are generated from a point or along a vector.

You can use silhouette curves and shadow outlines to conduct simple visibility obstruction studies, such as identifying blind spots that may restrict the line of sight in your models.
Shadow Curves

Outline the visible and obstructed areas for the painting of surfaces and solids.
Display plots of blind spots reserved for heavy machinery located in manufacturing plants.
Display on surfaces and planes the light pattern formed from a light source in safety systems and advertising.
Display a camera's field of view and plot the visible area, such as for a drone.

Silhouette curves extracted from a body using a point light source

Shadow Curve outline of a window projected onto an opposite face
Feature Enhancements

Thicken

Capability
• The **Thicken** command is enhanced so you can now select multiple faces or regions as input for a single thicken feature
• Selecting disconnected faces
• Selecting multiple regions from different sheets
• Selecting multiple faces from single body
Feature Enhancements

Thicken

Select Boundary Curves
lets you select regions from two separate faces, so you can apply a shared thickness to them

Add New Set
lets you select a new set of regions from separate faces

You can now select regions on multiple disconnected faces as input, and then offset them to create multiple solids in one thicken feature
Feature Enhancements

Thicken

You can select multiple disconnected faces from a single body as input to create multiple solids in one feature.
Feature Enhancements

Styled Corner

The Styled Corner command is enhanced so you can create corner faces that perfectly match the surface parameterization of the input blends.

Value

You can create high-quality blend corners where three blend faces converge.

- The degree and segments along curve C1 match those of blend B1.
- The degree and segments along curve C2 match those of blend B2.

\*Note\* If the segments of B1 and B2 are different but *compatible*, the larger number of segments is used.

If the segments of B1 and B2 are different and *incompatible*, the minimum number of segments to fit to C1 and C2 is used.

- The degree and segments along curve C3 match those of blend B3.
Feature Enhancements

Fill Hole

The Fill Hole command is enhanced as follows:

The Fill Hole type can now fill all holes at once when you set Selection Method to the new All option in the Edge Selection group.

A new type, Connect Holes, creates loft geometry between two selected holes, even from two different bodies.

Value

You can modify the shape of facet bodies more efficiently.
Feature Enhancements

Fill Hole

A new type, Close Notch, fills holes on laminar edges of facet bodies. In this context, a laminar edge is a facet body edge that doesn't border any other facets of the same body.

The Bridge Gap type now allows the selection of facet edge chains, even between two different bodies.
Feature Enhancements

Extract Virtual Curve – Tube Centerline

Use the new **Tube Centerline** type to create a new centerline or reconstruct a missing centerline as a curve feature through a circular body shaped like a tube or a hose.

You can create tube centerline curve features from a tube, which can be single or multi-segmented.

All the selected face loops should be closed, or you will get centerlines only for the closed face loops and nothing for the non-closed face loops.
Design For Additive Manufacturing

Carbon M2 allows us to produce isotropic polymer parts similar to those produced by traditional injection molding processes.

There's no stair-stepping like typical 3D printers produce.

We're proud to have been selected as the first Carbon Design Partner in late 2017.

We recently received our second Carbon M2 to speed up our production.
Design For Additive Manufacturing

- Unit Cell Editor
- Filter Lattice
- Added Tetrahedron & Surface Lattice
- Texture Mapping
- Morph Mesh
The **Unit Cell Editor** command allows you to create or edit a basic cell for user-defined lattice structures.

You can create cells from points or existing curves.

You can preview unit cells in a lattice pattern before you create it.
Filter Lattice

Use the Filter Lattice command to remove rods from the lattice based on filter settings.

You can filter for rods to remove based on:

- Their angle from the print normal plane
- Their maximum rod length
- Whether they are dangling
**Added Tetrahedron & Surface Lattice**

Using the Lattice command 

**Tetrahedron Fill**—Fills a boundary volume with a lattice by using its mesh facet edges as rods and adding tetrahedron structures inside.

**Surface**—Creates a lattice on faces by using their mesh facet edges as rods.
Texture Mapping

Select the Texture command to start the Texture task environment.

Use the Texture task environment to create and edit UV parameterization on convergent and analytic faces, and map 3D textures onto those faces. You can also perforate faces with textures. The result is always a convergent body.

In the Texture task environment, you can access several commands from the following groups on the Ribbon bar:

<table>
<thead>
<tr>
<th>Create</th>
<th>Modify</th>
<th>Preferences</th>
</tr>
</thead>
<tbody>
<tr>
<td>3D Texture</td>
<td>Delete</td>
<td>Display Distortion</td>
</tr>
<tr>
<td>Punch Through</td>
<td>UV Parameterization</td>
<td>Show 2D Lines</td>
</tr>
<tr>
<td>Delay Texture Update</td>
<td>Transform Parameterization</td>
<td>Show 3D Lines</td>
</tr>
<tr>
<td></td>
<td>Reverse Parameterization</td>
<td>Line Display</td>
</tr>
<tr>
<td></td>
<td>Make Symmetric</td>
<td></td>
</tr>
</tbody>
</table>
Morph Mesh

The Morph Mesh task environment provides a set of commands that you can use to deform and refine a facet body interactively using a cage structure and controls similar to NX Realize Shape.
Morph Mesh

Use these commands to modify control cages:

- **Extrude Cage**
  Lets you extrude selected control cage faces. The behavior is identical with the command for NX Realize Shape.

- **Delete Cage**
  Lets you delete independent cage regions.

- **Subdivide Cage**
  Subdivides selected control cage faces.

- **Merge Face**
  Merges selected control cage faces.

- **Split Face**
  Splits selected control cage faces.

- **Project Cage**
  Lets you project selected control cage elements to a curve or plane outside of the morph mesh feature.

Use these commands to modify control cages and the morph bodies:

- **Transform Cage**
  (Cage only, or cage and body) Lets you transform selected control cage elements, such as faces and vertices. You can transform the control cage independently, or transform the body along with the control cage.

- **Morph Constraint**
  Constrains control cage elements associatively to external geometry.

- **Delete Constraint**
  Removes associative constraints of control cage elements to external references.

- **Twist and Bend**
  (Cage and body) Lets you bend and twist the input body using a temporary control cage whose size and angles you can interactively set.
Morph Mesh

Use these commands to create cages or cage geometry:

- **Create Cage**
  Lets you create control cages to manipulate the input body. Control cages must always be closed so their deformation volume can be determined, but they do not need to encapsulate the input body entirely.

- **Cage Polyline**
  Lets you create cage polylines. The behavior is identical with the command for NX Realize Shape.

- **Extract Cage Polyline**
  Extracts a cage polyline from a curve outside of the task environment.

Use these commands to set preferences and display options:

- **Morph Mesh Preferences**
  Lets you manage control cage display and selection preferences.

- **X-ray Cage**
  Controls whether see-through display is turned on or off.

- **Allow Backside Selection**
  Lets you select control cage elements that are hidden behind other elements.

        Displays a grid on the XC-YC plane.
Aero Design

- Aero Flange
- Aero Rib
- Aero Step
- Aero Shelf
- Blend Pocket
Aero Flange

Use the Aero Flange command to create a flange body by offsetting a region of skin that forms the exterior of a vehicle or vessel.

Aero Flange is designed for use on airframe parts, but you can also use the command on any other type of parts.

You can specify both open and closed aero flanges.
Aero Flange

To control the location of the aero flange, you select geometry to construct guides relative to the skin.

You can construct a guide where the skin intersects a datum plane or faces, or where curves are projected onto the skin.

You can control the width of the flange by specifying:

- The width of the flange.
- The offset distance from the skin to the flange.
- The symmetry of the flange about the guide.
Aero Flange

To control the length of an open aero flange, you can specify a distance value from one or both ends of the flange.

You can specify the end location of an aero flange based on a distance from the ends of guide curves, or from a selected object.

You can control the thickness of an aero flange by specifying a thickness value, and whether the flange lies on the skin or is offset from the skin.
Aero Rib

Use the **Aero Rib** command to create a thin-wall rib body by extruding a guide curve from a face chain. You can create both open and closed ribs.

![Diagram of Aero Rib](image)

- **Vertical aero ribs as part of web**
- **Closed aero rib as part of web**
Aero Rib

You control the height of an aero rib by specifying:

• The dimension type.

• A height value.

You control the length of an open aero rib by specifying a distance value from one or both ends of the rib, or by selecting an object at one or both ends of the rib.
Aero Rib

When the **To Selected** end option is used, you specify the ending condition of an aero rib as **None** or **Skin**.

You control the thickness of an aero rib by specifying:

- The dimension type.
- A thickness value
Aero Step

Use the Aero Step command to create a step feature.

A step offsets a face region to add clearance where a part joins with another part.

Aero Step is designed for use on airframe parts, but you can use the command with other types of parts.
Aero Shelf

Use the **Aero Shelf** command to offset faces that share an edge to form a shelf along the edge.

The widening and narrowing capabilities of Aero Shelf simplify the process of creating and editing support structures for vehicles, aircraft, and vessels.

You can create two types of aero shelves:

- **End**
- **Middle**

You can use Aero Shelf to add or remove material.
Aero Shelf

You have many options for specifying the length, width, and depth of an aero shelf. You can specify:

- The method used to determine length.
  - Full
  - Shelf from End
  - Step from End
  - To Selected

- The width of the shelf.
- The method used to determine the depth.
  - Offset
  - To Next Wall
Aero Shelf

Example Uses

- Ramp
- Shelf extension
- Rib support
- Pocket shelf
Thank You
Applied CAx
The strongest support for Siemens PLM Software in the US

WE ARE
NX

FEMAP • NX CAD • NX CAM • SIMCENTER 3D • SOLID EDGE • STAR CCM+ • TEAMCENTER PLM