



**SIEMENS**

*Ingenuity for life*

# What's new in NX

June 2019

## Breaking down the barriers to innovation

### Benefits

- Faster delivery of functional enhancements through continuous release
- More efficient design with adaptive UI powered by artificial intelligence and machine learning
- Rapid modification of complex shapes with morph mesh feature
- Expanded support and automation for publishing technical data packages (TDP) to Teamcenter
- Improves performance and reduces design time with assembly load enhancements
- Reduces NC programming time with automated creation of complex tools
- 10 times faster toolpath regeneration provides more flexibility to optimize toolpaths
- Reduces setup time and eliminates errors in production with G-code driven robotics simulation
- Maximizes build tray capacity of industrial 3D printers with automated 2D nesting

### Introducing NX

Building on its legacy of best-in-class customer deployment readiness and data preservation, Siemens has enhanced NX™ software to deliver the next generation of design, simulation and manufacturing solutions. In this latest release, NX has introduced many enhancements that remove the barriers to innovation and make you more efficient. Your focus is on innovation and speed to market, and our tools empower your organization to meet high customer expectations and demands.

### NX architecture

#### Continuous release

Siemens Digital Industries Software is now delivering NX using a continuous release methodology. This new delivery model provides faster access to new enhancements and quality improvements while reducing the effort needed to effectively deploy NX. Siemens is the first major CAD/CAM/CAE vendor to deliver products through continuous release.

#### User interface

One of the most noticeable enhancements to NX CAD is the adaptive user interface (UI). Using machine learning (ML) and artificial intelligence (AI) capabilities, the UI can predict next steps

and help you work more efficiently by displaying commands for the most likely next steps.

With adaptive UI, the user interface automatically adapts to the needs of different people across multiple departments. This improves ease of use, resulting in higher adoption rates, accelerated learning and faster system adoption.

New functionality to reduce the learning curve within NX is introduced with First Assist command coverage. In addition, the electronic visual assistant offers you an on-demand graphic window inside NX for hosting specific visual assist content. Visual learning tools within NX improve ease of use and give you an engaging and quick learning experience.

### Token licensing

Beginning in summer 2019, NX product engineering will support tokens, also known as value-based licensing. This licensing will apply to many of the add-on modules that can be purchased to expand the capabilities of the NX Mach core seats. While this licensing approach is often used in the CAE world, Siemens is the first major vendor to introduce it for product design applications. The introduction of value-based licensing is part of an ongoing strategy to deliver flexibility to our customers, both in their approaches to deploying NX and in their ways of doing business with us. This flexibility is a competitive differentiator for NX, as unlike our competitors we are not forcing licensing approaches or contractual terms on our customers.

# What's new in NX

## NX for design

### Electromechanical design

Disjointed mechanical and electrical design systems create issues with lost information and limited traceability and visibility of the impact of changes. NX delivers a holistic design approach that integrates the mechanical, electrical, and electronic disciplines, enabling you to respond to changing demands quickly and easily, increasing confidence in product performance.

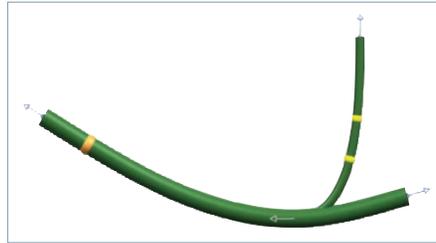
### NX PCB Exchange

NX PCB Exchange has many key enhancements that address the ever-increasing complexity of electronic printed circuit boards (PCBs). Radio frequency (RF) components are designed in shapes that have specific functionality. In order to address these complexities, the latest enhancements within NX enable easier collaboration and management of RF design data to help decrease design time.

The IDX format for sharing data between electrical and mechanical CAD systems includes enhancements to workflow notifications. The notification system uses conventional office tools to keep users up-to-date with the latest design iterations. There are several thermal analysis enhancements that improve design efficiency and the overall user experience. These improvements enable better collaboration and boost productivity in the design process.

### NX Routing

NX Routing includes key new features that improve ease of use, productivity and collaboration for routed systems design. NX now includes Capital® electrical systems engineering software collaboration enhancements that enable users to browse Capital projects and diagrams from within NX via an embedded Capital web client. You can now create 3D bundles directly from Capital® Harness™, and this release also supports access to electrical design content from Capital without a full client license of Capital.



New splice enhancements help reduce design time. Splice placement options include allowing splices to be located on paths without defining the path; the splices move with changes to the path. Splices placed in Capital can automatically be placed in NX and can be moved in NX and updated in Capital.

### Mechatronics Concept Designer

Enhancements to Mechatronics Concept Designer include faster simulation and validation of handling robots in machines. Users can now leverage offline inverse kinematics to validate machine designs including handling robots and to validate teach points of robots during virtual machine commissioning. These tools enable you to exchange project data faster, saving time and improving collaboration.



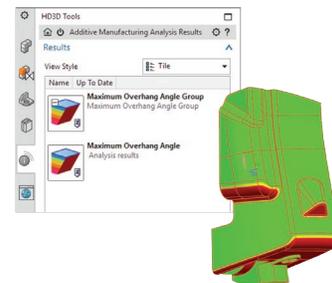
### Generative engineering and integrated validation

Disconnected systems for design and validation make it more difficult to verify the function and quality of your designs. In NX, designers can harness the power of a combination of technologies to develop the next generation of products.

### Validation

Validation in NX has been enhanced to provide a more productive user

experience. Expanded checking framework capabilities now include color mesh analysis and persistency. These can be applied to the additive manufacturing overhang check and are available for inclusion in Check-Mate checkers. These enhancements provide immediate feedback on the impact of design changes and reduce evaluation time.

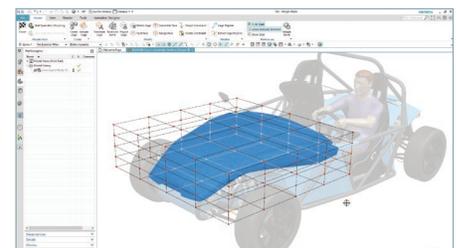


### Most productive modeling environment

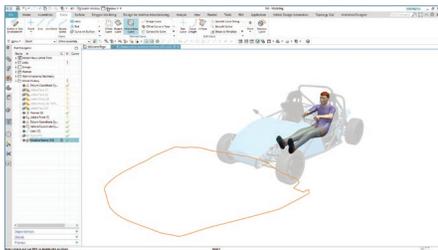
In the past, you may have used multiple disconnected solutions to complete the entire design, validation and manufacturing process. NX enables a flexible approach to design allowing parametric, surface and facet geometry to work seamlessly in a single model.

### Design

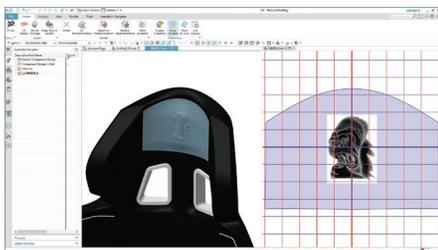
One of the major enhancements in NX includes new and improved capabilities for faster reconstruction of traditional CAD geometry from convergent meshes. The new Convergent Modeling™ morph mesh feature enables complex shape modifications that are not possible via other means. You can select one or more convergent bodies for editing using multiple morph cages and make edits to either the entire body or local areas.



A new feature called shadow curve supports vision system design workflows. With the shadow curve feature you can create silhouette curves on the design body and create shadow outline curves on a plane, cylinder, sphere or sheet body. The silhouette and shadow curve features provide better visibility and faster analysis of your products.



Another new feature within design called the texture modeling command enables you to easily convert 2D graphics and textures into 3D geometry for use in molding and 3D printing. Users can transform, scale, pattern and emboss or punch through modeled textures.



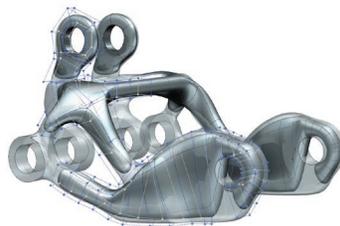
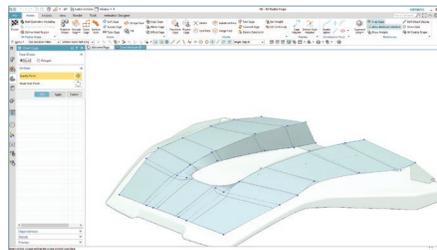
### NX Aerospace Design

Aerospace structures design has been enhanced to deliver greater feature capability that supports a broader range of use cases. Aerospace shelf modeling now has improved material removal that automatically finds intersecting ribs. An update to freeform chamfers includes using ramps with zero length. Aerospace flange modeling has been enhanced with improved preview behavior and intuitive and consistent length and direction values. Aerospace step modeling enhancements include intuitive and consistent offset direction

values and improved “extend to next wall” results. With these enhancements, designing aerospace components with NX is ten times faster than with traditional methods.

### NX Realize Shape

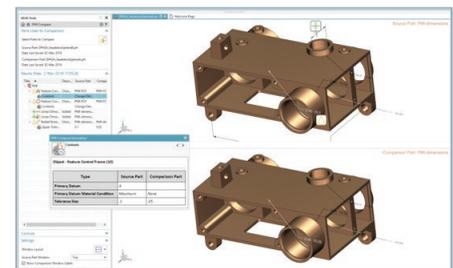
The introduction of a draw cage capability in NX Realize Shape™ software significantly accelerates creation of subdivision surfaces using scanned or other geometry as a reference. A new section tube command uses subdivision resurfacing methods to deliver faster results for refining topology than previous methods.



### Model-based definition

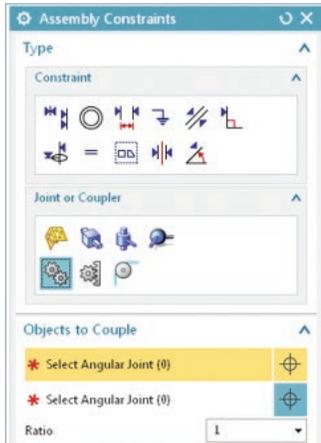
Workflow and functional improvements for model-based definition are focused on design efficiency and productivity. One of the major enhancements is expanded support and automation for publishing technical data packages (TDP) to Teamcenter using a new TDP dataset. There is now automation via batch process publishing as well as optional automatic generation and attachment of STEP242 for 3D PDF, JT™ format for 3D PDF and JT plus PDF. This enhancement supports effortless publishing of technical data packages. TDP enhancements also include several UI improvements that enable you to create and publish configurable templates to provide a smoother experience.

Product and manufacturing information (PMI) annotation plane definition has been enhanced to improve ease of use and discoverability. Defining and orienting PMI on a model is now more natural, easier to interpret and more productive. Face selection for PMI dimensions now features simplified dimension creation on spherical and toroidal faces, leading to significantly faster and more intuitive authoring. Updates to PMI compare for hole callouts and PMI display dramatically reduce PMI and model view checking time and easily identify changes. NX PMI now supports workflows based on smart topology table (STT) enabled JT data when opened in NX with minimally load lightweight display. Minimal loading of JT data with STT is now consistent with minimal loading of PRT files, improving ease of use. Together, these enhancements provide measurable gains in productivity, increased efficiency and simplification.



### NX Animation Designer

NX Animation Designer enhancements provide you with the best tools to share designs in the highest quality, optimizing the evaluation process with faster simulation setup and validation. Users can leverage joints and couplers created in assemblies and validate joint definitions to achieve realistic articulations and to toggle through key motion positions such as full open and closed. You can now export high-definition (HD) videos to share your designs across teams to evaluate designs.



### Assemblies

Because you need to visualize larger amounts of data than ever before, NX meets this requirement with best-in-class assembly loading performance. Performance enhancements include improvements to the minimally load lightweight display option.

In addition, JT format assemblies and components can now be loaded minimally, and NX smart lightweight capabilities allow you to work with facet data and when required exact data is loaded automatically.

### P&ID Designer

P&ID Designer now features easy-to-use authoring for the layout of piping and instrumentation systems. You can now drag and drop to place equipment and connect components with common pipe stocks. Updates to integrated change management enable you to manage all aspects of a design process from systems, sheets and runs to equipment. This capability reduces errors and design time by managing and communicating P&ID designs and progress.

### Mold, tool and die design

Enhancements to mold, tool and die design are focused on improving overall tool design productivity. They include

tool design workflow that eliminates many manual modeling steps, report generation from completed tool designs and usability improvements for standard parts accessibility from the re-use library.

### Ship structure design

Ship design enhancements include an improved steel structure design workflow for creating profile cutouts that saves time throughout the workflow. The ship navigator has undergone several usability enhancements that make it easier for you to find and center selected objects in the navigator from the graphics window, edit parameters of ship structure objects, and automatically assign parts to respective nodes as part of basic to detail design transition. Additionally, the ship navigator will now be available in the NX Ship Structure Detail Design application.

### Collaborative design management

Increased supplier collaboration creates challenges with integrating systems from different sources. Increasing complexity causes more packaging challenges. Lack of traceability of requirements from design to validation and manufacturing leads to quality and performance issues. The unmatched breadth and depth of the Teamcenter portfolio means that you can solve more of the tough challenges required to develop highly successful products.

### Teamcenter Integration for NX

Updates within Teamcenter Integration for NX enable you to launch NX and load descendants upon opening from Active Workspace. This contributes to faster launches, time savings and improved ease of use.

### Immersive visualization

NX enables users to review their designs from an entirely new perspective, with interactive modification, measurement,

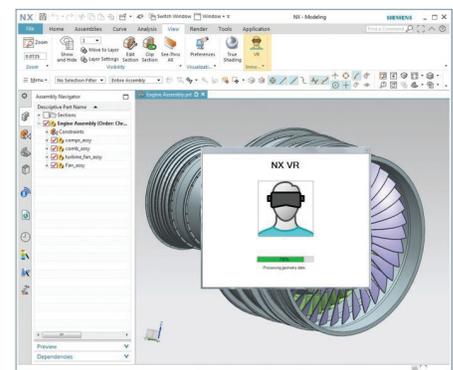
and sectioning of parts in the digital twin at full scale.

### Visualization

Visualization has been enhanced to include a new update display setting to improve display quality when faces change during design, new options for enhanced selection, and new fly-through options to provide even better control for directing and stopping navigation. These updates streamline design workflows with smoother view navigation.

### NX Virtual Reality

The breakthrough integrated virtual reality (VR) tools of NX have been enhanced with new capabilities. These include an intuitive UI specifically designed for 3D that offers navigation and viewing controls, inspection and review tools, rulers, sections, snapshots and 3D paint markup. These updates yield better understanding and communication of the digital twin, fewer analysis cycles and reduced design time.



### NX for manufacturing

New and enhanced capabilities in NX for manufacturing allow companies to stay competitive using advanced manufacturing methods, cutting tools and innovative technologies, including:

- Automated numerical control (NC)

programming, enhanced machining process visualization, and G-code-driven robotic simulation for efficient programming and error-free production

- High-speed cutting methods combined with a new generation of barrel-shaped tools for significantly shorter machining times
- Programming multiple parts on a line using NX Machining Line Planner's expanded capabilities for higher productivity

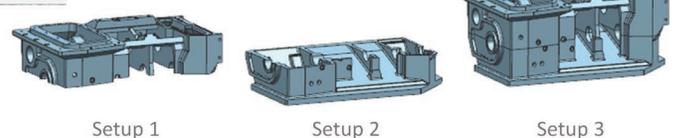
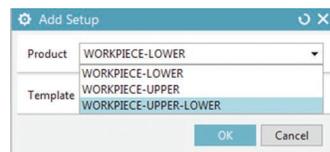
### Prismatic parts machining

Automatic tool creation using part features can significantly accelerate NC programming of parts with complex multi-step holes that require special drilling tools. Using conventional methods to manually create drills and program these holes is time-consuming. NX CAM automates the process: it recognizes the hole geometry and then finds an existing step drill or automatically creates a new step drill to machine the hole. In the process, you can define the machined sections and specify material stock for subsequent machining. This automated process can significantly reduce the tool creation time.

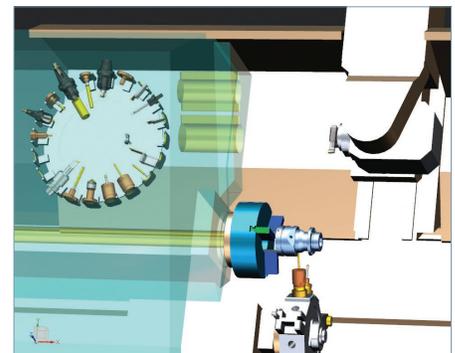
Tool magazine visualization helps you program CNC machine tools equipped with tool magazines faster. You can accurately display the machines' tool magazines, along with all the stored tools. During NC programming, you can

easily assign or change the target pockets. The accurate digital display of the tool magazine helps you avoid any possible collisions between the tools stored in the magazine.

NX Machining Line Planner enhancements enable manufacturing of multiple products on a line, which is important for automotive manufacturers. For example, after machining the upper and lower parts of a transmission case, the two parts are assembled and machined together. For each setup, you can export a solid in-process workpiece (IPW) that tracks the state of the machined part for accurate visualization. The new capability enables you to create drawings associative to the IPW after each step, ensuring that you use the latest version of the shop documentation even after design or process changes.



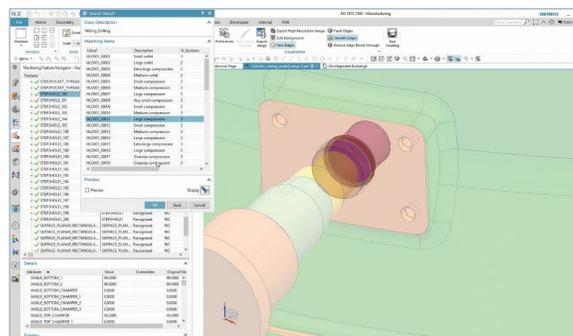
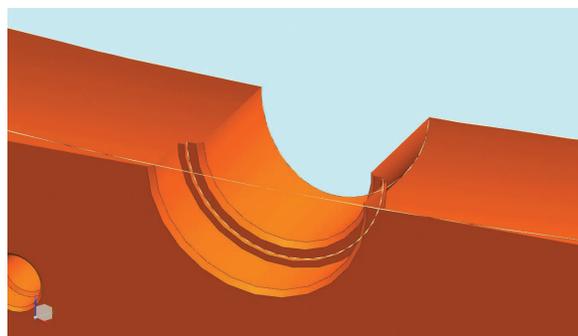
Transmission case example: the upper and lower parts of the case are machined in setups 1 and 2, respectively. The two parts are assembled and machined together in setup 3.



Programming of CNC machines with automatic tool magazines is streamlined with NX CAM enhanced visualization capabilities.

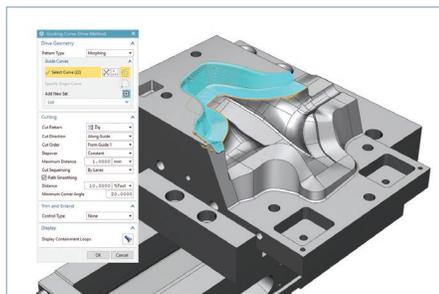
### Mold and die machining

Adaptive milling, NX CAM's high-speed roughing method, can reduce machining time by up to 60 percent. With new enhancements you can generate toolpaths with smooth engage and retract moves. The sharp corners of the non-cutting transfer moves can be rounded as well. These new controls help you further reduce machining time, minimize machine wear and extend tool life.



NX CAM uses the geometry of multi-step holes to automatically create custom step tools for machining complex holes.

Guiding curves finishing, an advanced machining method, uses parts' edges to generate toolpaths that follow the natural shape of the machined area. New enhancements help you efficiently handle imported CAD part models that often have imperfect geometry, such as disjointed edges. Previously, manual repair of these curves was necessary. Now, NX identifies the gaps and automatically connects disjointed curves, saving programming time and creating smooth finishing operations that result in excellent surface finish.

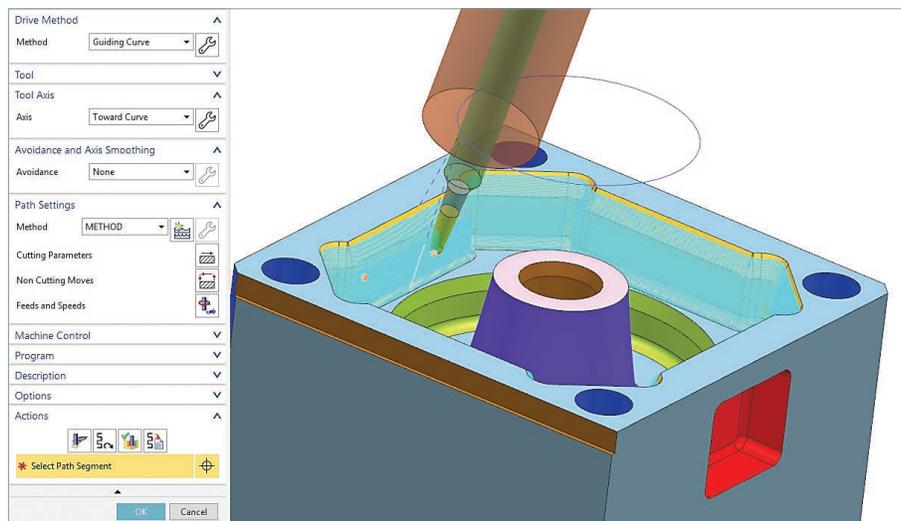


Guiding curve finishing automatically connects disjointed edges, saving programming time and ensuring high-quality surface finish.

### Complex parts machining

Smart toolpath regeneration slashes the time required to recalculate 5-axis toolpaths after adjusting the tool axis, which can be critical to successfully programming complex parts. NX CAM software almost instantaneously recalculates the updated toolpath while maintaining the toolpath smoothing and trimming settings. You can achieve 10 times faster toolpath regeneration. This rapid toolpath recalculation allows you to virtually try different tool axis settings to define the best machining strategy for parts with challenging geometry.

Taper barrel tools are being increasingly used for 5-axis machining because of the unique advantages they provide: machining higher surface quality with fewer passes. NX CAM's support for these tools has been expanded to include taper barrel tools. The large barrel radius makes the shape of these

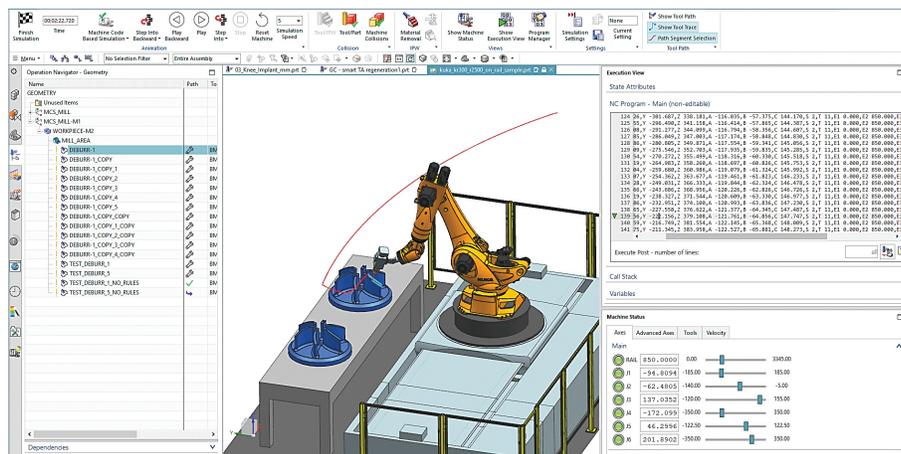


Smart toolpath regeneration enables 10 times faster toolpath recalculation after adjusting the tool axis, helping NC programmers find the optimal way to machine complex parts.

tools almost conical, allowing you to position the tool against a flat wall and prevent holder collisions. You can quickly model taper barrel tools based on data from tooling vendor catalogs. Using taper barrel tools for 5-axis machining operations helps improve the surface finish and reduce machining time.

G-code driven robot simulation enables accurate validation of the robotic tasks prior to production. With NX CAM

software, you can program robots to perform machining tasks as well as material handling operations. The new simulation enhancement enables you to emulate fly-by motions using the zone values and simulate external NC programs and subprograms. In this release, digital models for KUKA robots are provided. Using the G-code driven simulation, you can validate robotics operations in NX CAM, helping you reduce setup times and eliminate errors.



G-code driven robot simulation enables digital validation in NX CAM, which minimizes setup time and eliminates errors in production.

### Simulation and verification

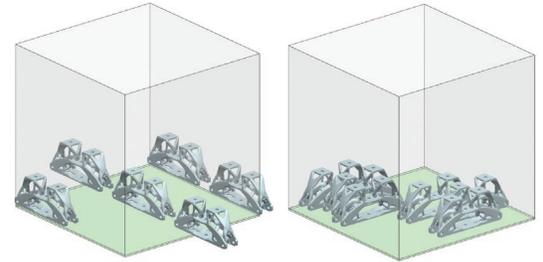
Multiple in-process workpieces can be used to create a composite IPW that serves as an input blank for subsequent machining. In the past, it was only possible to transfer a single IPW source to another setup, which can limit your manufacturing capabilities. For example, it is more effective to manufacture complex castings by machining different parts sections in multiple setups and then assemble them together for final machining in a subsequent setup. With this new enhancement, you can streamline the NC programming and improve your production efficiency.

### Additive manufacturing

In previous versions of NX, designers could specify a print coordinate system (CSYS) that relayed information about the preferred print orientation to the

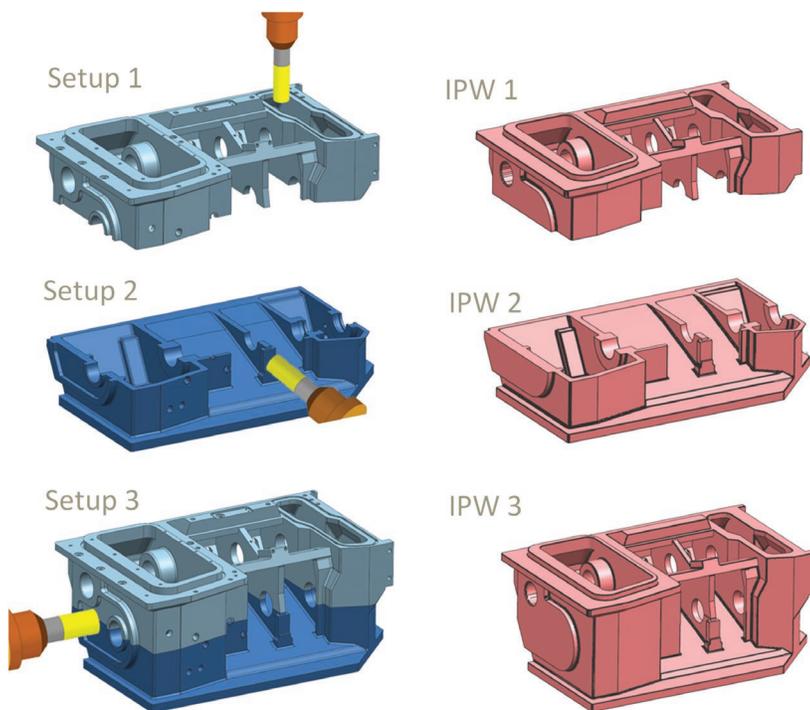
printer operator. The print CSYS information is saved with the part, and with the latest release of NX, the printer operator now has the option to add parts to the build tray using the print CSYS to define the initial orientation. The operator can adjust the orientation using the normal build tray setup tools in NX. As print orientation can affect part function and finish, this functionality facilitates communication about the print orientation between the designer and the machine operator.

3D nesting has been part of the NX additive manufacturing functionality for quite some time, but 3D nesting of parts in a build tray is primarily used for plastic printing. For printing with metal materials, parts are almost always arrayed in a single layer on the build tray. 2D nesting is now an option that can be selected when nesting parts in



Parts in the build tray before and after using the new 2D nesting functionality in NX additive manufacturing.

the build tray in NX additive manufacturing. This option automatically places as many loaded parts onto the build tray as possible. When parts are loaded using an existing print CSYS for orientation, those orientations can be honored using the constraints in the automatic nesting dialog.



NX CAM enables programming of disassembled parts in individual setups and creating the resulting IPW (setup 1 and 2). Multiple IPWs are assembled into a composite IPW, which is used as an input blank for subsequent machining (setup 3).

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