

NX Mach Series Industrial Design

fact sheet

Siemens PLM Software

www.siemens.com/plm

► Summary

NX® Mach Series Industrial Design software is a diverse and comprehensive set of tools that spans the product development process from concept to production surfacing, including Class A surfaces. NX Mach Series Industrial Design creates a finished product, not just a surface model.

Features

Provides tools for creating, modifying and analyzing free form and aesthetically pleasing surfaces and shapes

Allows for conceptual models as well as for producing precisely defined surfaces, including Class A surfaces

Supports the designer's need to control surface curvature and surface transitions

Facilitates mathematically well-defined and controlled manufacturable design solutions

Provides dynamic displays of changes

Delivers easy to use functionality for sophisticated designs

Provides comprehensive toolkit for design visualization

The NX Mach Series Industrial Design suite offers advanced surface design tools, analysis tools and high-end visualization capabilities specifically tailored to the needs of industrial and automotive designers, as well as automotive stylists. These tools include all of the basic options for the initial concept stages, such as the creation and visualization of proposed designs, as well as additional tools for the production of primary and secondary surfaces. NX Mach Series Industrial Design combines surface and solid modeling tools and puts powerful traditional engineering tools on the designer's desk.

Because of its level of integration with the entire NX Mach Series of products, NX Mach Series Industrial Design users can take full advantage of modeling, assemblies, simulation, manufacturing and product data management functionality. The combined strength of all of these products delivers the most complete industrial design and advanced surfacing solution available today.

Free form modeling

NX Mach Series Industrial Design facilitates the creation of curves and surfaces that are mathematically precise by using tools that eliminate the complexity inherent in the results. The NX Mach Series Industrial Design suite lets users create Class-A surfaces and provides surface-to-surface continuity controls that are suitable for the most demanding requirements, including automotive and consumer product design, guaranteeing manufacturability.

Basic free form modeling. This complex-shape modeling application supports the creation of complex surface and solid models. Basic free form modeling capabilities include:

- Create solids from sheets
- Basic sweeping along curves
- Proportionally developed shapes using 1, 2 and 3 rail methods
- Lofting – ruled, curve mesh, lofted shapes using standard conic methods, and meshes of points and curves
- Special surface creation – surface extension and n-sided, bounded plane, offset
- Surface manipulation tools: surface extension and surface normal control
- Body-based trimming
- Surface trimming using curves

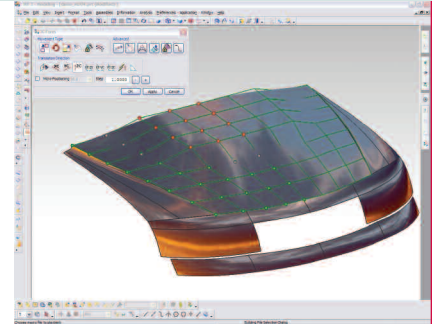
Advanced free form modeling. Extends capabilities to offer the types of surface creation and manipulation techniques often used when creating complex, organic forms in the conceptual design stage of product development. An example would be the direct surface manipulation tools that provide real-time graphical feedback. Also included is a suite of analysis tools that assist in the analytical and visual evaluation of surface quality. Advanced free form modeling capabilities include:

- Guided assistance when generating solids from sheets
- Advanced filleting capabilities – circular and/or conic (constant, linear, S-shaped, variable radius)
- Advanced surface creation tools – blend, bridge and transition surface creation from external data – through points, from poles, from point clouds
- General-use design and manufacturing sweeps, and flanges – geometric law extension, silhouette flange, ribbon builder, sectional sweep
- Approximated offsetting of complex areas
- Surface shaping via pole and control point manipulation
- Surface redefinition via boundary, degree and stiffness controls
- Associative, global model deformation – alter surfaces to explore design alternatives
- Modify surfaces to account for effects of springback and metal forming
- Simplify models by combining several surfaces into a single surface
- Extend and enlarge sheet bodies
- Isoparametric trim and divide
- Abstract modeling for finite element analysis using mid-surface

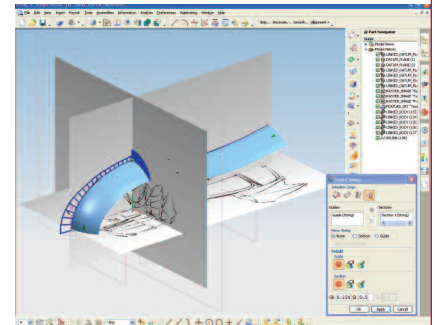
Free form shape design. Enables designers to create conceptual surfaces for quickly capturing initial design intent, as well as creating and editing curves directly on surfaces. It has direct surface modeling capabilities that maintain associative surface boundary controls ranging from G0 to G3. Styled sweeps associatively sweep profile curves along multiple guides, and advanced surface trimming creates cut surfaces independent of original surfaces.

Advanced surface analysis

Integral to the ability to create or modify curves and surfaces is the ability to dynamically analyze the effects of user input. NX Mach Series Industrial Design includes an extensive set of tools for analyzing and visualizing the shape of resulting surfaces. Advanced surface analysis functions include a deviation gauge, section analysis, adjacent edge analysis, draft analysis and grid section analysis – all important capabilities for validating surfaces employed in designs.



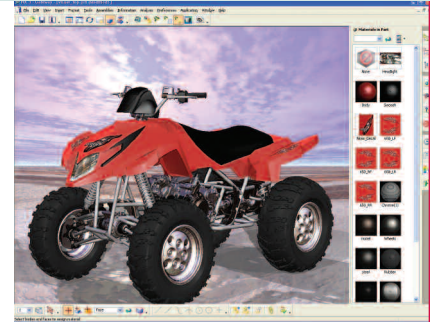
Advanced surface design tools include direct pole manipulation, surface continuity controls, and reflectivity analysis aids.



NX Mach Series Industrial Design includes specialized tools for creating organic, styled shapes in precision geometry, using hand sketches as a foundation for modeling.

Dynamic and photorealistic rendering

Provides users with rendering tools to communicate designs clearly and to create accurate images that can be used throughout the design and manufacturing processes. Users can accurately visualize designs to reduce costs and shorten design cycles, and to specify real-world materials that will be used when their products are manufactured.



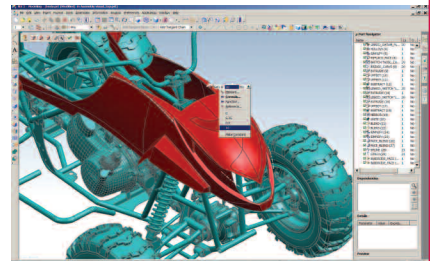
NX Mach Series Industrial Design includes surface textures, lighting effects, and advanced rendering for photorealistic image creation.

Solid and feature-based modeling

This general modeling capability supports the creation of 2D and 3D wireframe models, swept and revolved bodies, Boolean operations and basic associative editing. Feature-based modeling supports the creation and associative editing of standard design features such as holes, slots and pockets. It lets users hollow out solid models and create thin walled objects. A feature can be located relative to any other feature or object and can be instanced to establish associative sets of features.

Assembly modeling

Assembly design. This capability supports “top-down” and “bottom-up” assembly modeling. It provides for rapid navigation of the assembly hierarchy and allows direct access to the design model of any component or sub-assembly. It supports the “design in context” approach where changes can be made to a design model of any component while working in the context of the assembly.



With solid modeling, feature modeling, assembly design, and high-performance surfacing, NX Mach Series Industrial Design is a comprehensive product design and styling solution.

NX includes tools for building and manipulating assembly structures. The use of inter-part relationships, such as mating conditions and basic WAVE links, enables the creation of parametric assemblies which capture and preserve the design intent whenever changes are made.

Assemblies can be easily navigated using the assembly navigator in conjunction with intelligent component searching capabilities. NX assemblies also includes support for modeling and switching between different states of flexible assemblies and parts.

Advanced assembly modeling. Extends the toolset for building, editing and evaluating assembly models. It is especially useful for users working with large assemblies, but it also improves the productivity of most users who work regularly in an assembly context.

Component filtering techniques allow users to quickly identify and load the components of relevance to their current task, avoiding unnecessary delays and screen clutter caused by loading irrelevant components.

Advanced assembly modeling also provides extra flexibility with faceted representations to further improve the performance and memory efficiency of loading large assemblies. Assembly enveloping techniques enable the user to represent major subassemblies as simplified abstractions to further reduce assembly load times and avoid the display of unwanted or proprietary interior detail.

The sophisticated clearance analysis and weight management tools provided in advanced assemblies are also optimized for analyzing potential problems with fit, clearance and mass properties.

Rapid prototyping

NX can automatically output/export model data in faceted STL format used with rapid prototyping technologies, such as stereolithography and fused deposition. This capability reduces turnaround time regardless of the specific rapid prototyping technology employed.

User-defined features (UDFs)

User-defined features provide an interactive means to capture and store families of parts for easy retrieval and editing. They let users take an existing associative solid model, created using standard NX modeling tools and establish relationships between parameters, define feature variables, set default values and decide the general form the feature will take when started. Existing UDFs reside in a library that can be accessed by anyone using NX feature modeling tools.

Straight brake sheet metal

This solids-based application is focused on design for manufacturing of sheet metal parts. Users can create sheet metal component models using feature-based design tools for tabs, flanges and other typical features. They can define forming tables and bend sequence tables, and reform the solid model taking into account material deformation properties. The sheet metal tools generate accurate flat pattern data for downstream applications from solids, sheets and wireframe geometry.

Product validation

The NX Mach Series Industrial Design suite validates product designs with a quality assurance checking utility. The product validation capabilities evaluate parts, assemblies and drawings to check that:

- Files are compliant to corporate standards and best practices
- Models are parametrically correct
- Model quality problems do not occur (for example, mismatched edges, tiny gaps in faces and non-manifold conditions)
- Assemblies are complete

The product validation capabilities allow for different levels of checking. For example, a company can use the results of a check to determine product quality metrics. By using product validation, users can reduce design rework time by detecting and eliminating quality issues during the design stage in the product life cycle.

Optimization Wizard

Helps customers understand which design parameters are really key to their design objectives. Users identify candidate variable design parameters and a design goal. The wizard then applies sensitivity and filtering tools together with engineering constraints to identify the more critical design parameters and then optimize them. The step-by-step Wizard provides design engineers with a method to ensure that their product designs are fully optimized to best meet their design goals. It also lets designers and engineers capture engineering requirements, automate the exploration of design alternatives and automatically identify optimized solutions.

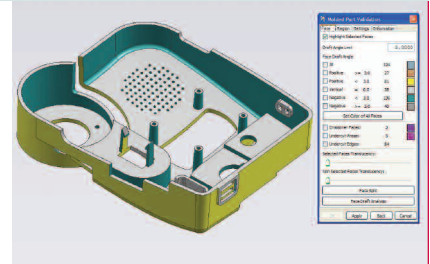
Molded part validation

With NX molded part validation, designers can check the draft of a part, even if they know virtually nothing about mold design. NX analyzes parts and automatically provides designers with information about draft angles, undercut areas, sharp corners, small radiuses, etc. It provides designers with an easy visual check of core and cavity sides that doesn't require any special analysis or knowledge of mold design.

By detecting errors early, designers can avoid the time-wasted back-and-forth that occurs when the tooling department or supplier discovers that a part can't be molded as designed.

Validation capabilities include:

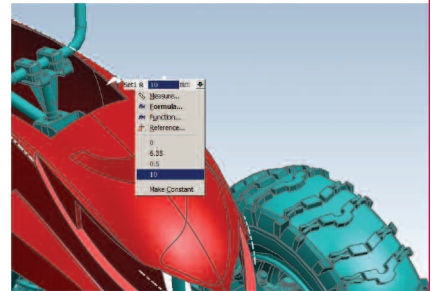
- Examining face properties with the ability to color faces for several conditions and types of analysis
- Additional validation information for optimizing mold production and part manufacturability



Molded part validation automatically checks plastic part designs for manufacturability.

DesignLogic

Enables users to add design intent or knowledge on the fly in the form of formulas and expressions. For example, a designer may wish to constrain a design dimension so that it is determined by a formula or mathematical expression. DesignLogic is implemented so that right-clicking design elements allows entry of the formula or expression that controls it. Users can create features that add intelligent checking to a product's definition. The check feature is similar to other NX features. However, the check feature helps a designer to ensure a product's compliance to corporate standards throughout all phases of the design process through generating reusable rules and formulas. The tools come in the form of checks for mass, distance, size and expression. Check features display in the Part Navigator as standard features.



NX DesignLogic makes it easy to capture and control design intent in the form of expressions and formulas.

Custom program execution

Custom program execution enables users of NX Mach Series Industrial Design to run custom programs that were developed with the NX Open software development tools or the Knowledge Fusion knowledge-based engineering (KBE) package.

Web publishing

NX enables users to publish design data to the web with an HTML publisher for creating detailed documentation for component parts or assemblies based on information in NX part files. Web publishing uses template files that contain both HTML and special NX embedded commands. These commands extract information from a design file and write it to an HTML file that can be read universally.

GD&T

NX quickly and accurately associates tolerance information to geometric objects via tolerance features, including datums, feature control block and geometric dimensioning and tolerancing symbols. NX geometric tolerancing is based on a user-specified GD&T standard, and enforces the creation of

standard-compliant designs. NX also supports 3D annotation with process and manufacturing information that includes reference geometry, driving and annotation dimensions, GD&T symbols, tolerances, finishes and other product and manufacturing information directly on a 3D model. The 3D annotation complies with the ASME Y14.41 and ISO TC10 standards. Collectively, all of this data is known as product and manufacturing information and is the sum of all information needed to define and manufacture a product. The propagation of product manufacturing information (PMI) throughout a product's life cycle has proven to be more beneficial than static drawings.

Drafting

Includes tools to automate and streamline engineering drawing production. Drawings are associative to models, so model changes automatically update and are reflected in related drawings. Drafting capabilities include detailing, dimensioning, symbols, sheet layout and placement of standard orthographic and auxiliary views, automatic view creation from the 3D model, hidden line processing. Templates that include drawing borders and view layout can be dragged and dropped on models to automate much of the tedium. Drafting tools automatically comply with the drawing standard selected by the user – ANSI, ISO, JIS, DIN.

Data exchange

The NX Mach Series contains translators that can exchange the following types of design data – IGES, DXF/DWG, STEP 203 and 214, STL and 2D exchange.

Managed development environment (MDE)

The managed development environment (MDE), powered by Siemens' Teamcenter® technology, is a foundation product, process and program data management system integrated with all NX Mach Series solutions. It provides a variety of capabilities for vaulting, sharing and securing product data across distributed organizations that can be deployed to meet the challenges of today and scaled to meet those of tomorrow. An integral component of the NX Mach Series, the MDE organizes and protects all product data (including models, drawings, manufacturing and NC data, performance simulations, images, technical documents and other related information). With comprehensive search tools, the MDE saves time locating product data and helps coordinate the work of design team members. Designers can also take advantage of where used/referenced tracking and impact information.

Extended MDE option (Mach-E)

The extended managed development environment (MDE) extends product and process management capabilities as needed to include engineering change management, configuration management and process management, combining these as a platform for repeatable digital validation. By extending the environment beyond typical CAD data management, regardless of whether that data is from NX I-deas®, NX, Solid Edge®, CAM and CAE systems or other CAx products, the extended MDE supports workflow and change management. These capabilities coupled with configuration management provide a platform for realistic and sustainable portfolio management.

Add-ons

Customers can extend NX Mach Series Design solutions with a selection of optional add-on modules. These add-ons enable customers to configure their design solutions to specific requirements such as specialized design tools, standard parts applications, design-integrated simulation solutions, programming and customization toolkits and direct translators.

Consult the NX Mach Series Add-Ons Catalog for specific information on the products available.



Contact

Siemens PLM Software

Americas 800 498 5351

Europe +44 (0) 1276 702000

Asia-Pacific 852 2230 3333

www.siemens.com/plm