

NX for Mechanical Design

Benefits

- Facilitates design control, speeds the design process, increases designer and design team productivity and improves design throughput
- Improves design team performance, especially for handling large, complex models
- Raises product quality by minimizing design errors
- Produces faster, more accurate and complete product documentation
- Produces significant time, effort and cost savings by facilitating design re-use
- Facilitates better integration and coordination between multiple design disciplines, design teams and their related CAD systems

Features

- Comprehensive 3D design capabilities, including wireframe, surface, solid and direct modeling
- Synchronous technology for uniting parametric and history-free modeling in the same design environment
- Assembly modeling with full-context, multi-CAD digital mockup and validation tools

Summary

The NX™ software for mechanical design provides a comprehensive set of leading-edge CAD modeling tools that enable companies to design higher quality products faster and less expensively. The NX comprehensive mechanical design solution lets you choose the tools and methodologies that best suit your design challenge. Innovative technologies deliver breakthrough mechanical design capabilities that set new standards for speed, performance and ease-of-use.

Transforming product development by delivering greater power, speed, quality, productivity and efficiency for mechanical design

NX mechanical design capabilities are unmatched in terms of the power, versatility, flexibility and productivity they deliver to your digital product development environment. NX enables you to establish a complete design solution for your environment, including leading-edge tools and methodologies for:

- Comprehensive high-performance modeling, which enables you to seamlessly use the most productive modeling approaches – from explicit solid and surface modeling to parametric, process-specific and history-free direct modeling that works with models from any CAD system.
- Active mockup and assembly design, which enables you to work interactively with massive multi-CAD assemblies while leveraging leading assembly management and engineering tools.
- Standards-compliant drafting and 3D annotation, which streamlines the creation of product documentation by directly leveraging your 3D master model.

Today's mechanical design challenges

Manufacturing companies are driven by an increased pressure to develop more innovative products in a shorter time frame while continuing to deliver high levels of quality. Even with mature 3D CAD technology, many companies fail to significantly reduce process waste, improve product quality or deliver breakthrough product innovations that stir the imagination of the marketplace.

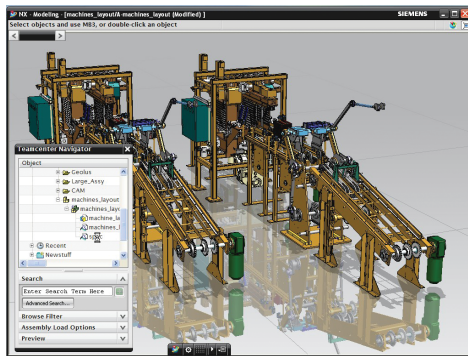
These challenges require a radical rethinking of the business model that pertains to product development. This rethinking should begin with transforming the design process. Product design fuels the entire development

NX

NX for Mechanical Design

Features *continued*

- Interactive design of massive assemblies that improves the performance and capacity of your design environment
- Process-specific, streamlined modeling tools for sheet metal routed systems and other applications
- Configurable, intuitive interface that facilitates ease-of-use, user learning and accessibility to powerful modeling capabilities
- Associative integration with all NX product development solutions, including NX industrial design, electromechanical, simulation, tooling and machining solutions
- Automated, real-time design validation checking to monitor functional requirements
- Knowledge capture and automation tools
- Seamlessly integrated, transparent engineering data and process management



NX seamlessly integrates Teamcenter-managed engineering and process data, enabling designers to quickly search a single source of knowledge and locate the information they need to use.

effort. Today's design processes involve increasingly complex products comprised of design elements created by multiple teams, disciplines and suppliers using independent CAD systems. These complex processes require product makers to coordinate the activities of team members dispersed across different geographies while retaining design intent from the start of the design project to its completion.

Companies need design processes that compress the design cycle by eliminating value-added tasks, maximizing knowledge re-use and proactively addressing manufacturability issues before they reach the factory floor. Product developers need to "design-in" rather than "inspect-in" product quality.

NX next-generation design solutions

NX delivers next-generation design solutions that transform the entire product development cycle. NX represents a radical departure from conventional CAD systems. NX improves speed and efficiency while eliminating wasted work by providing unique technologies and methodologies, including:

Knowledge-enabled design

NX automates and simplifies design by enabling you to leverage the product and process knowledge that your company has gained from its experiences as well as from industry best practices. NX tools enable designers to capture knowledge in the form of high-level product structure, templates, frequently used design features, engineering rules, formulae and validation checks. Knowledge-enabled design helps your company reduce design costs, compress the design cycle and improve design quality.

You can manage your entire design process with Siemens' Teamcenter® software, which lets you establish and seamlessly integrate a single source of product and process knowledge into your design environment. This enables you to coordinate your design chain, standardize your design processes and accelerate decision making throughout the design cycle.

Process innovation NX enables you to establish an interactive environment where everyday design work can be streamlined through the implementation of task-oriented workflows that improve designer productivity. NX design environments facilitate high-performance modeling techniques that provide design teams with the flexibility and power to handle design of virtually any size or complexity. NX allows you to dynamically integrate your CAD processes with planning, simulation, tooling, manufacturing and other lifecycle processes and make informed design decisions by recognizing the requirements of all design stakeholders and coordinating their activity.

More specifically, NX addresses the mechanical design process directly through its capabilities for:

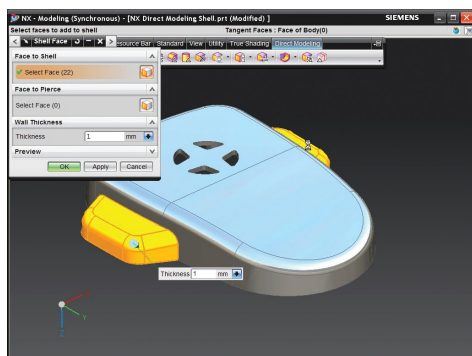
- Comprehensive high-performance modeling
- Active mockup and assembly design
- Standards-compliant drafting and 3D annotation

Comprehensive high-performance modeling

NX delivers the most powerful and flexible modeling solutions available – solutions that enable you to freely use any modeling technique that fits your design challenge. All NX modeling tools are built on Siemens' Parasolid® geometry modeling kernel, the world's most powerful, robust and widely used modeling foundation.

Design Freedom NX supports Design Freedom powered by Siemens' groundbreaking synchronous technology. This unique approach enables you to unite feature-based parametric and history-free modeling in the same design environment.

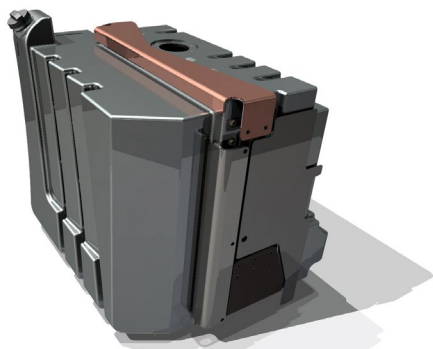
Design Freedom means that your designers can use NX synchronous modeling tools to modify design geometry initially created on other CAD systems or by other modeling techniques. It does not matter whether the data in question was imported from another CAD system or whether is native parametric or nonparametric. NX



NX synchronous technology enables designers to perform history-free editing on imported CAD models.

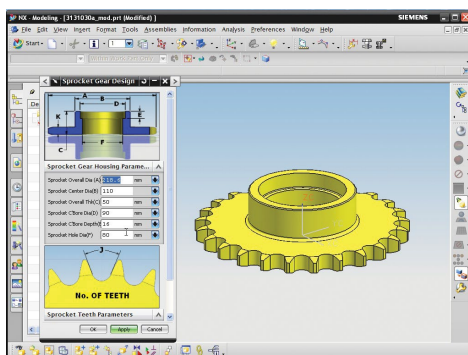
synchronous tools enable designers to work directly with any geometry without the need to rebuild data. These synchronous tools let designers use parametric features without the limitations of a feature history.

Process-specific modeling Conventional CAD systems leave it up to the designer to figure out how to apply a system's various tools. In contrast, NX logically structures commands in workflows oriented to accomplishing specific design tasks. NX is prepackaged with industry roles that tailor the user interface and incorporate best practice guidelines. NX also delivers process-based tools that build in domain expertise for meeting specific challenges (such as dealing with sheet metal components) or for performing industry-specific processes (such as handling automotive body structures and general packaging). Process-specific design aids enable designers to work faster than general purpose CAD tools.



Sheet metal design is one of several streamlined process-specific modeling tools provided by NX.

Design re-use Design re-use becomes increasingly more important as today's manufacturers try to expand their market share by establishing product platforms. NX lets you use any part or assembly as a template for new designs, building product knowledge and best practices into your development processes. This capability supports your commonization, modularization and re-use initiatives by enabling you to reduce design time by up to 80 percent.

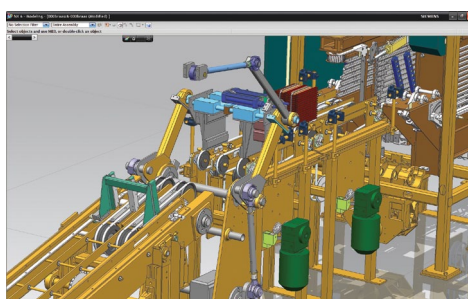


NX knowledge-enabled design tools let companies re-use existing designs as customized templates for new designs.

Active mockup and assembly design

NX powerful mockup and assembly capabilities provide your design environment with a variety of interactive capabilities that expedite your assembly design and engineering processes.

Active Mockup NX Active Mockup enables your designers to easily navigate large assemblies and establish an appropriate environment for detailed work on

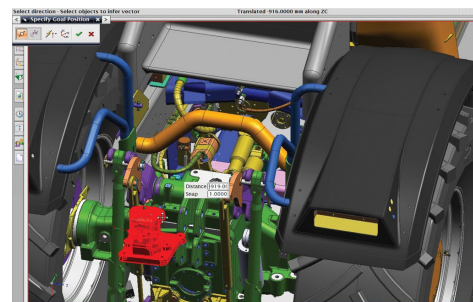


NX Active Mockup and assembly management enables collaborative design to be performed in the full context of complex assemblies.

subassemblies and components. Active Mockup utilizes industry-standard JT™ technology to enable designers to load thousands of components from multiple CAD systems in just seconds. JT is a highly flexible CAD neutral format that allows designers to fully represent all relevant model information. The JT format can be created from most major CAD applications. Depending on your type of business process, JT data can be very lightweight or very rich. JT models are able to hold precise model geometry, product structures, attributes and PMI, including geometric data, translations and annotations. JT enables you to load assemblies with up to a million parts with exceptional interactive performance.

As a result, Active Mockup provides designers with a true interactive design capability that lets them rapidly display, section and rotate very large assemblies. Active Mockup enables design teams to collaborate by viewing, modifying and evaluating complete digital mockups. Teams can view as much of the product design as required to evaluate its parts in the context of a particular task. These design-in-context capabilities facilitate rapid problem resolution during the physical assembly process.

Assembly design validation NX provides validation tools that enable your designers to identify and resolve assembly design and process problems in the early stages of development – without resorting to physical prototyping. NX enables designers to perform interactive clearance checking and interference checking to detect and



NX assembly validation tools include extraction path planning capabilities that optimize designs so that a product can be easily assembled, disassembled and maintained.

eliminate fit problems. Designers can interactively simulate assembly motion to check and optimize moving components.

With automated assembly path planning and motion envelopes, designers can optimize products for assembly, disassembly, maintenance and service. Designers can record and play back assembly and motion sequences as movies that can be shared as assembly instructions with the factory floor. These simple to use tools can be leveraged to quickly validate design changes as the product evolves.

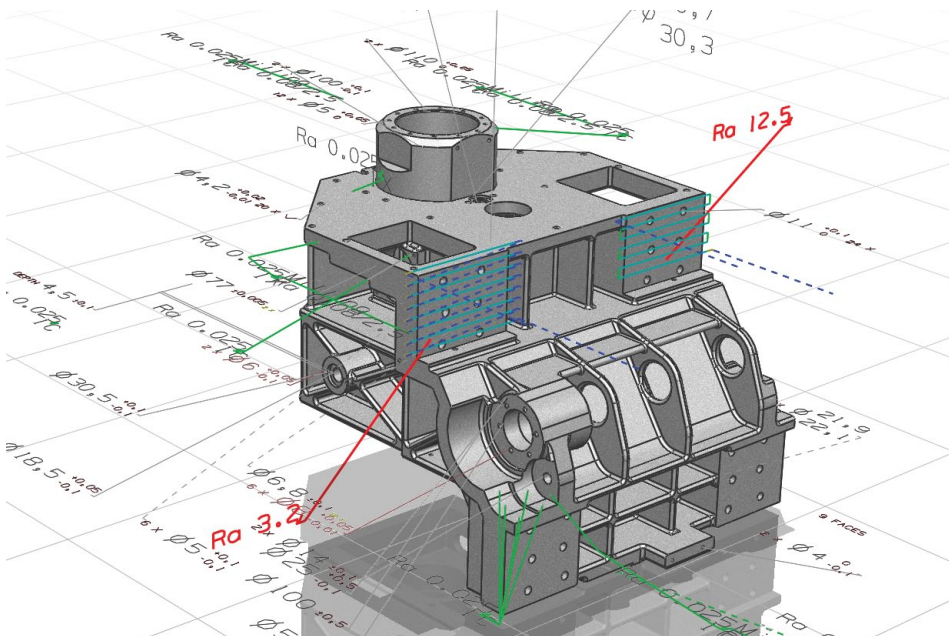
Routed systems design NX provides tools that enable designers to create and validate routed subsystems for both mechanical and electrical designs. Mechanical routing tools and libraries are available for tubing, piping and steelwork. Electrical routing tools enable designers to place wiring, conduit and raceways while standard component libraries are available for electrical systems. NX electrical routing includes wire harness design and manufacturing support, eliminating the need for physical prototype and reducing product development time by allowing designers to perform interface checks, validate design rules, visualize the routing pattern in 3D and trace the location of specific wires and connections.

NX maintains associative links between the P&ID layout and the 3D systems to ensure that system logic is maintained. Designers can quickly apply logical design changes to the routed system. Routed systems are fully associative to NX assemblies to facilitate design changes. Automated bill of material and other reporting provide information for subsystem manufacturing.

Standards-compliant drafting and 3D annotation

NX delivers production-driven tools for documenting your designs either as standards-compliant 2D drawings or 3D product and manufacturing information (PMI) annotations. You can use these capabilities to ensure that design intent is properly communicated throughout your development organization. They improve product quality by removing potential sources of interpretation error from your design environment and by speeding the process you use to take your designs to manufacturing.

NX drafting capabilities enable designers to rapidly lay out their drawings, as well as to create drawing views, detailing and dimensioning and geometric dimensioning and tolerancing (GD&T). Since NX adheres to the industry standards for PMI 3D model annotation, the dimensions, symbols and tolerances can be automatically inherited on drawing views and directly used by downstream analysis and manufacturing applications. This improves productivity by eliminating the need to re-enter this information.



Designers can use NX to directly include complete PMI on a 3D design model, thereby improving communications and accelerating downstream applications.

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NX Mach Series Industrial Design

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

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.

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.

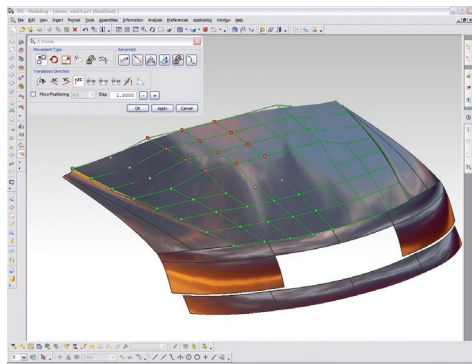
Basic free form modeling This complex-shape modeling application supports the creation of complex surface and solid models. Basic free form modeling capabilities include:

- 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

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NX Mach Series Industrial Design

- 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



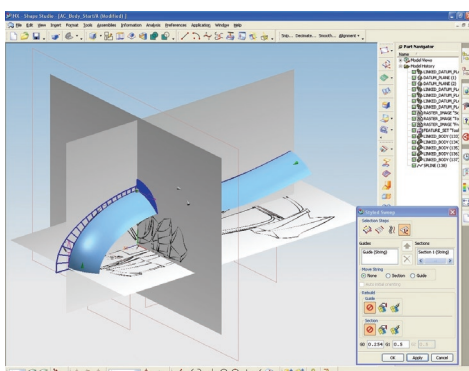
Advanced surface design tools include direct pole manipulation, surface continuity controls, and reflectivity analysis aids.

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.



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

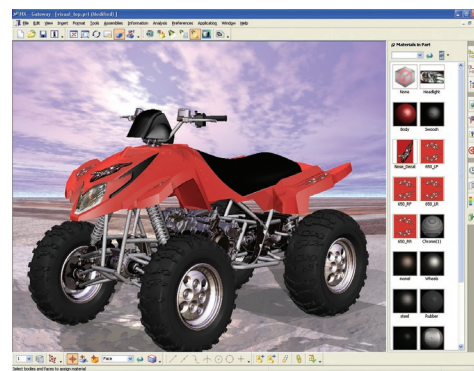
Advanced surface analysis

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.

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.



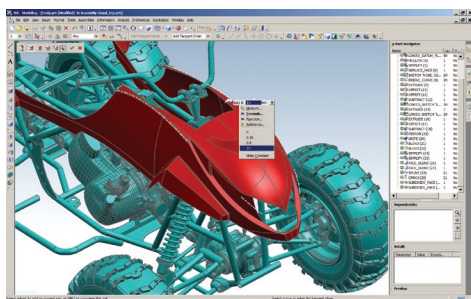
NX Mach Series Industrial Design includes surface textures, lighting effects, and advanced rendering for photorealistic image creation.

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 subassembly. 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 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:

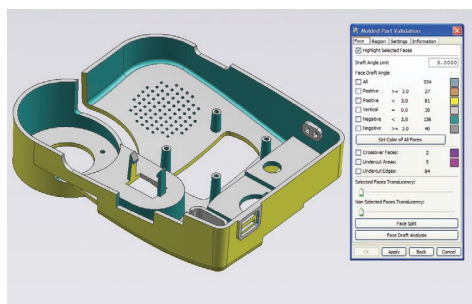
- Files are compliant to corporate standards and best practices
- Models are parametrically correct
- Model quality problems do not occur (for example, mismatched edges, tiny gaps in faces and nonmanifold conditions)
- Assemblies are complete

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 automatically checks plastic part designs for manufacturability.

Molded part validation

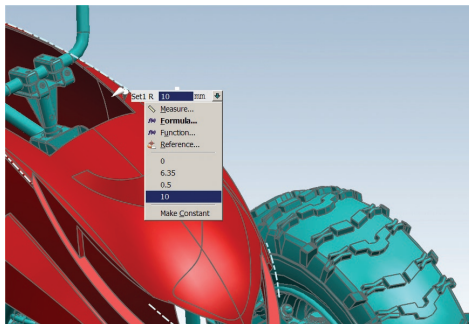
With NX molded part validation, designers can check the draft of a part, even if they know virtually nothing about mold design. NX analyzes parts and automatically provides designers with information about draft angles, undercut areas, sharp corners, small radii, etc. It provides designers with an easy 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



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

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.

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.

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NX General Packaging

Intelligent tools for automotive mechanical and occupancy packaging

Benefits

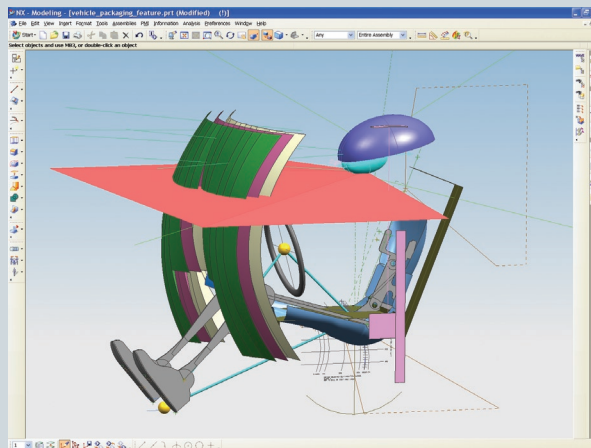
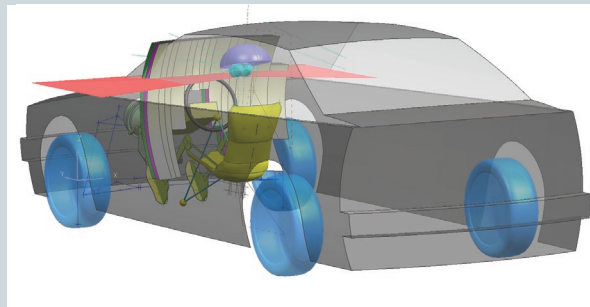
- Industry standards and process knowledge are stored for repeatability and standardization
- All tools are ready to go out of the box with minimal customization costs
- Frees up highly skilled packaging engineers to concentrate on complex packaging problems
- Enables rapid evaluation of alternative designs in a unified design environment
- Leverages integrated analysis to provide fast feedback on design's compliance with established standards

Features

- Automatic geometry creation and integrated compliance checks
- Permits early occupancy and mechanical packaging studies
- Helps establish key vehicle reference points for configuring and measuring aspects of the interior automobile compartment
- Estimates horizontal seat travel for accommodation studies

Summary

NX™ General Packaging is a suite of tools that automates many of the tasks associated with the mechanical and occupancy packaging of an automobile. NX General Packaging provides assistance to certify the vehicle for compliance to SAE standards and a variety of governmental regulations, including those in the US, Europe and Asia. The software helps users assess the accommodations and ergonomic performance of the vehicle.



Vehicle Packaging.

All of the tools in NX General Packaging are written in the style of wizards, and offer step-by-step instructions to complete complex vehicle packaging tasks. In addition, NX General Packaging embeds industry-specific knowledge of the process of vehicle design and certification. Associativity of the output geometry automatically updates analysis results as the design of the vehicle evolves from art to final product.

NX

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NX General Packaging

Features *continued*

- Determines the anchorage location for vehicle seat belt restraint systems
- Determines maximum reach of drivers in a simulated driving situation
- Represents location of driver's eyes
- Determines visible and nonvisible areas of instrument panel
- Verifies windshield and wiper system performance against established standards
- Analyzes the field of vision of outside and inside rearview mirrors
- Permits early studies of driver's vision angles
- Determines how interior components reflect on the windshield
- Assists in understanding the effect of engine movement on adjacent components
- Creates an envelope representing the full range of a tire's motion

NX General Packaging addresses a broad range of packaging tasks, including:

Vehicle Packaging To define overall parameters, NX General Packaging provides wizards for vehicle and occupant packaging. With the Vehicle Packaging wizard, users specify vehicle type (Class A or Class B), transmission type, driver side location and for Class B vehicles, the number of rows and maximum passengers per row. The user can then create occupants for the selected seating positions using the Vehicle Packaging wizard. The wizard guides the user through the specification of seat movement, male/female ratio and type of restraint system. These packaging parameters are associatively linked to the other tools included in NX General Packaging, eliminating the need to re-enter data for specific packaging tasks and providing a single context for all occupancy packaging-related studies for a vehicle.

Hip Point Design The Hip Point Design (HPD) wizard helps establish key vehicle reference points for configuring and measuring aspects of the interior automobile compartment. These points include seating reference points for each occupant position and heel points (accelerator heel point for the driver and floor reference points for passengers).

The wizard creates the reference points according to the Driver Designated Seating Position Design Procedures and Design Procedures for second or third Row outboard seating positions in the SAE J4002 Feb 2004 Standard. The HPD wizard meets the recommendations of SAE J4002 Feb 2004 and applies to Class A vehicles only.

2D Manikin The 2D Manikin wizard in NX General Packaging permits occupancy studies early in vehicle development. Leveraging actual human statistical data, the manikin can be sized to represent a wide range of driver and passenger populations on both Class A and Class B vehicles.

Represented as contours in a side view, the 2D manikin is created in accordance with the recommendations of the SAE J833 and J826 standard. In addition to occupancy

considerations, the manikin can be used for performing basic posture checking with an integrated posture checker.

Seat Lines The Seat Lines Assistant is used during early stages of vehicle development to create seat position lines and to determine the length and position of horizontal seat travel. As a design tool, the assistant makes it easy to determine the level of accommodation provided by proposed seat tracks. As a checking tool, it implements SAE J1516 and J1517 standards describing where drivers position their adjustable seats.

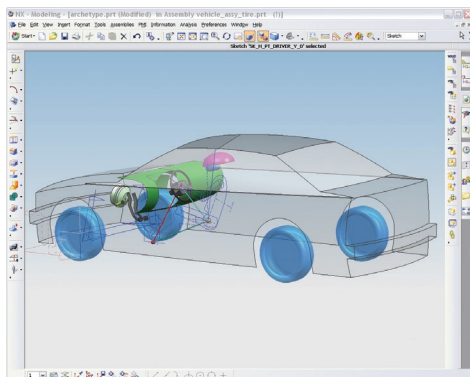
Seat Belt Anchorage This tool provides design recommendations for determining the anchorage location for vehicle seat belt restraint systems. The option is useful when performing occupant packaging studies to determine anchorage locations for restraint systems that conform to the guidelines of SAE.

Output includes the acceptable anchorage zone for the pelvic restraint system, the upper torso restraint system and the top tether for child restraint devices. The anchorage point for the pelvic restraint can be located on the vehicle structure or on the seat assembly.

This function follows the recommendations of SAE J383 (June 1995) and SAE J1369 (October 1997). For SAE J383, it creates a closed curve representing the acceptable zone for the anchorage point of the pelvic restraint system. A closed curve is also created for the acceptable zone of the upper torso restraint system. For SAE J1369, it creates a sheet body representing the enclosed volume in which to locate the anchorage point for the top tether strap of a child restraint system.

Driver Control Reach This tool can be used to estimate the extent of a driver's hand control reach. Hand control reach is the maximum reach of drivers in a simulated driving situation, with the nonreaching hand on the steering wheel and the right foot on the accelerator pedal.

The software creates 3D surfaces forming envelopes representing the hand reach capability for a specified proportion of the



2D Manikin.

driver population and torso restraint system. The function follows the recommendations of SAE J287 (June 1988).

Eyellipse The Eyellipse wizard helps certify vehicles for compliance with various regulatory standards. The wizard graphically depicts the location of the driver's eyes for specified population percentiles, creating an elliptical model representing driver eye locations in the vehicle and also creates features representing head contours and EEC vision points. These are used in other NX General Packaging tools, including Instrument Panel Visibility Assistant, Windshield Vision Zones Assistant and Mirror Certification Assistant. The eyellipse is also integrated within the Hip Point Design and Vehicle Packaging tools.

The Eyellipse wizard creates features that comply with a selection of standards, including SAE J941 (September 2002, June 1997 or October 1985) for eyellipses, SAE J1052 (August 2002 or April 1997) for head position contours and the EEC Vision Points 77/649 standard.

Instrument Panel Visibility The Instrument Panel Visibility wizard lets you calculate the visible and nonvisible areas of an automotive instrument panel which can be limited by the steering wheel and smart switch levers. Designers can use this wizard to determine early in the vehicle development process if the shape of the steering wheel with its smart switch levers meets the recommendations of SAE J1050a, which describes the driver's view of the instrument panel.

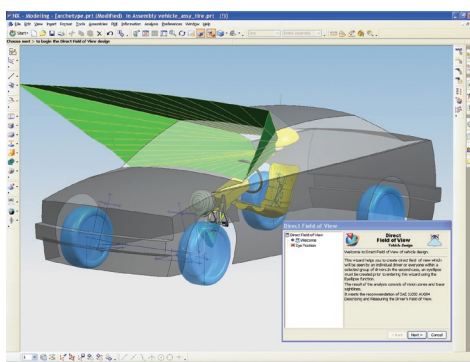
Once the designer creates or selects the geometry representing the instrument panel location, steering wheel rim and hub and smart switches, the wizard can analyze and report steering wheel obstruction, steering wheel vision faces, smart switches/levers obstruction and smart switches/levers vision faces.

Windshield Vision Zones The Windshield Vision Zones wizard lets you verify the conformance of a windshield and wiper system design to established vehicle standards. You can generate test areas on a windshield based on SAE and ECE

(European) standards. The program creates the actual windshield wiped area and calculates the percentile of the actual wiped area and checks it against the standard.

The designer specifies the vehicle class, eyellipse, hip point, windshield and wiper system details with step-by-step guidance from the wizard. With this input, the wizard calculates the wiped area and test areas A, B and C and checks whether the results meet the requirements of the standard.

Direct Field of View The Direct Field of View function can be used early in the vehicle development process to calculate the ambinocular vision regions of an individual driver or a group of drivers and to evaluate whether the vision angles are within an acceptable range.



Direct Field of View.

The user can limit the vision region by either defining the eye/head rotation or by specifying a window boundary. The regions are dependent on the eye points or an eyellipse (the eyellipse takes into account the vehicle type and the driver population percentile). The software creates a geometric feature representing the direct field of view and can also perform quick checks on the up, down, left and right angle values of that field of view.

A-Pillar Obscuration American and European automotive standards define the limits by which a driver's field of view may be obscured by the A-pillars of a vehicle. The A-Pillar Obscuration wizard measures this obscuration using the methods recommended by these standards, and reports whether the vehicle adheres to the

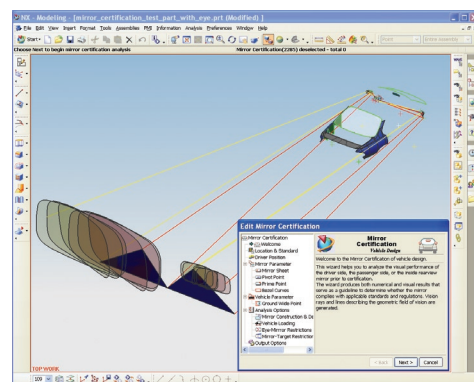
standard or violates it. You can also work in reverse and create criteria geometry (such as sight lines) for an A-pillar, given a target obstruction angle.

This function follows the recommendations of SAE J1050 Jan 2003 and EEC 77/649. By using A-Pillar Obscuration during visibility studies, designers can determine the right obstruction angle that conforms to the guidelines of SAE and EEC. The software can generate sight lines given a target obstruction angle and outboard or inboard point of the A-pillar.

Glazing Shade Bands NX General Packaging includes a tool that automates the creation of curves defining the driver glazing shade band. It aids in performing visibility design studies for vehicle packaging during automobile development. The output is curves that mark the shading boundaries.

A glazing shade band is the area immediately adjacent to and below the top edge of a vehicle's glazing. Light transmission and visibility through the shade band is less than what is required for normal driving. The software provides step-by-step guidance for creating the glazing shade bands. Recommendations in the standards SAE J100 January 2005 are supported.

Mirror Certification The Mirror Certification wizard performs analyses on automotive driver and passenger side outside and inside rearview mirrors before certification, or certifies the performance of an existing mirror design. The wizard generates vision rays and lines describing



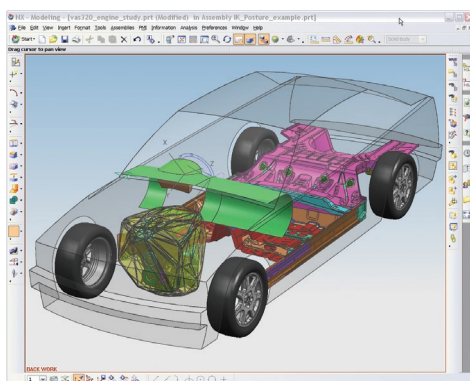
Mirror Certification.

the geometric field of vision of rearview mirrors. These vision rays and lines comply with national standards for inside and outside rearview mirrors for the following countries and regions:

- USA – Canada: FMVSS 111 – CMVSS 111
- Japan: Article 44/TRIAS 29/TRIAS 39
- Europe: 71/127/EEC (European Economic Community) – ECE R46.01 (Economic Commission for Europe)
- Saudi Arabia – Gulf States: SSA 770/771 – GS 421/422
- Australia: ADR 14/02

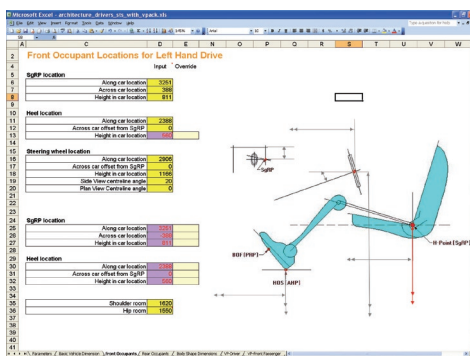
Reflection Data The Reflection Data wizard projects and creates the reflections of points or curves onto a face when viewed from a specific eye position. You can also use reflection data to create trace curves representing the path of the reflected light rays and to simulate reflections on a solid, transparent object, such as a sheet of glass (the refraction of light rays passing through the glass is considered). Automotive designers will find this tool useful to verify the reflection of interior components on the windshield of the automobile.

Engine Roll The Engine Roll wizard creates a feature envelope containing the full range of an engine's motion. Although the engine in an automobile is generally treated as a stationary object, the mounts do allow the engine to move slightly as a result of acceleration, braking and cornering of the vehicle. This movement may be sufficient to cause collisions with adjacent components.



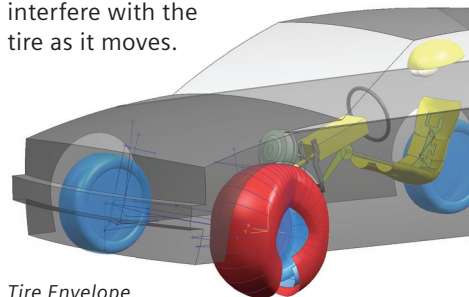
Engine Roll.

Designers can also use the Engine Roll wizard to study interferences between the engine and adjacent components and after the detailed design is complete to verify whether the engine compartment has been packaged correctly.



Functional requirements stored within Microsoft Excel spreadsheets can associatively drive the NX General Packaging tools.

Tire Envelope The Tire Envelope wizard creates a feature envelope that contains the full range of a tire's motion. This is useful during the body design process to ensure that designed parts – for example, BIW, exterior trim or ancillary drivetrain components – do not interfere with the tire as it moves.



Tire Envelope.

The wizard provides step-by-step guidance and comes with standard suspension templates for short long arm (SLA), MacPherson strut and five-link suspension and tire profile templates.

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