



SIEMENS



Siemens PLM Software

# NX for Automotive Suppliers

Flexibility and power to work with all your customers

## Benefits

- Boosts product design productivity
- Accelerates automotive design processes
- Promotes collaboration
- Reduces design process waste
- Improves design quality

## Features

- Automotive design with supporting tools in a complete product design solution from design to production
- High-performance modeling, drafting and comprehensive assembly design powered by synchronous technology
- Preconfigured solutions tailored for the full range of automotive design applications

## Summary

The NX Automotive Supplier bundles provide a full set of automotive design tools using NX™ software, the world's leading design solution for automotive OEMs. These bundles are prepackaged and competitively priced in three different performance tiers. They include tools and applications for general design, documentation, and validation, as well as automotive-specific tools developed with our partners in the automotive industry. NX Automotive Supplier bundles provide value-added upgrades to move customers up from one tier to the next. Each tier provides a complete solution package capable of production automotive work.

NX Automotive Supplier bundles include engineering process management capabilities for improved design team collaboration. Optional extended engineering process management tools provide scalable collaboration and advanced management.

All of the NX Automotive Supplier bundles share a common denominator of standard design and drafting, as well as data management tools. Each of the solutions

builds upon the other, providing increasingly more sophisticated and advanced design capabilities.

## Bundles

### [Automotive Supplier Entry Bundle](#)

This entry-level solution provides capabilities for creating and editing designs of typical automotive components and assemblies, with solid modeling and drafting, basic freeform modeling and sheet metal design. It includes tools for design review, rapid prototyping, web publishing, validation checking, high-definition 3D (HD3D) reporting tools and custom program execution. It also adds curve and surface-quality analysis tools and a full set of translators for taking advantage of non-native data.

### [Automotive Supplier Engineering Bundle](#)

This bundle includes all the capabilities in the NX Automotive Supplier Entry Bundle, and it adds specialized tools such as advanced freeform shape modeling, shape visualization and analysis and advanced sheet metal design. It also includes the ability to use user-defined features, and WAVE control for interpart relationships in assemblies.

# NX for Automotive suppliers

## Features *continued*

- Power and flexibility that supports virtually any design methodology
- A cornerstone of a complete automotive product development system
- Foundation for product lifecycle management
- Extendible with a selection of add-on application modules

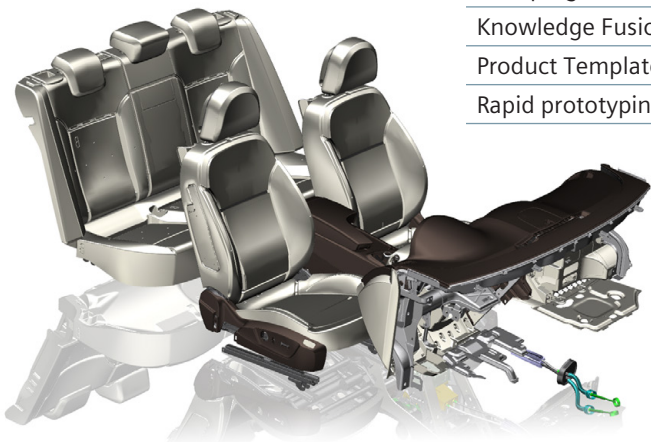
This bundle also adds enhanced validation capabilities, with tools such as Optimization Wizard and validation tools specifically designed for molded and sheet metal parts.

## Automotive Supplier Advanced Engineering Bundle

The Advanced Engineering Bundle is the highest-performance solution, containing a wide variety of NX design tools. In addition

to those in the other bundles, the NX Automotive Supplier Advanced Engineering Bundle adds a specialized automotive body design module, as well as additional packaging capabilities, weld creation and analysis, and electrical and mechanical routing. It also includes the NX Advanced Simulation software package for high-performance CAE capabilities.

	Automotive Supplier Entry Bundle	Automotive Supplier Engineering Bundle	Automotive Supplier Advanced Engineering Bundle
<b>Design modeling</b>			
Solid/feature modeling	•	•	•
Synchronous modeling	•	•	•
Basic freeform modeling	•	•	•
Freeform shape modeling		•	•
NX Realize Shape		•	•
Rendering	•	•	•
Visualize shape		•	•
Analyze shape		•	•
Assembly design	•	•	•
Advanced assembly design		•	•
User-defined features	•	•	•
WAVE control		•	•
Body design			•
General packaging			•
Packaging for review	•	•	•
GRIP program execution	•	•	•
Knowledge Fusion sharing	•	•	•
Product Template Studio	•	•	•
Rapid prototyping	•	•	•



NX provides a variety of tools for automotive interior design

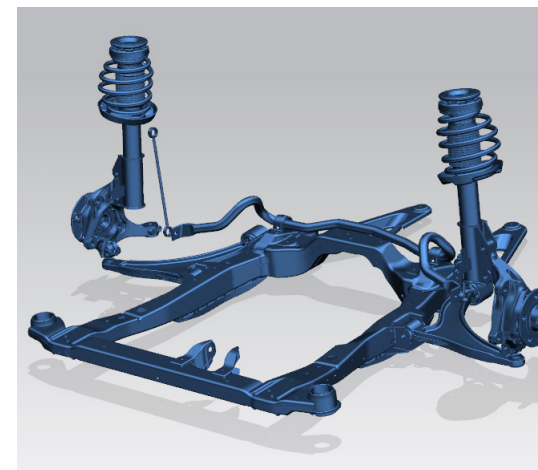
	Automotive Supplier Entry Bundle	Automotive Supplier Engineering Bundle	Automotive Supplier Advanced Engineering Bundle
<b>Process-specific modeling tools</b>			
Sheet metal design	•	•	•
Advanced sheet metal design		•	•
Weld creation and analysis			•
Basic routing		•	•
Electrical and mechanical routing			•
<b>Drafting and annotation</b>			
Drafting	•	•	•
Product and Manufacturing Information (PMI)	•	•	•
HTML publishing	•	•	•
<b>Product validation</b>			
Optimization Wizard		•	•
Molded part validation		•	•
Basic model validation	•	•	•
Sheet metal formability analysis		•	•
Curve and surface quality analysis	•	•	•
HD3D Visual Reporting	•	•	•
HD3D validation checking	•	•	•
<b>Translators</b>			
IGES	•	•	•
DXF/DWG	•	•	•
STEP AP203/AP214	•	•	•
2D Exchange	•	•	•
CATIA V5	•	•	•
<b>Simulation</b>			
Stress and Vibration Wizards	•	•	•
Advanced Simulation			•
<b>Data management</b>			
Teamcenter NX CAD data management	•	•	•
NX Embedded Client	•	•	•

## Automotive design tools

### Design modeling

#### *Solid/feature-based modeling*

provides the core modeling capability to support the creation of 2D and 3D wireframe models, swept and revolved bodies, Boolean operations and basic associative editing. NX uses a unique approach to 3D design that combines parametric, feature-based modeling with the speed and efficiency of direct modeling with synchronous technology. These tools support the creation and associative editing of standard design features such as holes, slots and pockets, as well as automotive-specific features. With a full range of parametric modeling operations, NX enables you to locate features relative to any other feature or object and to instance features to establish associative sets. NX also supports advanced modeling techniques for blending, tapering and hollowing to create thin-wall components. Synchronous modeling works with both native and imported geometry, enabling you to directly modify design data from other CAD systems, enabling easy collaboration with OEMs and suppliers.



*Assembly design is easier and faster with NX advanced tools.*

**Assembly design** 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 any component of the design model while working in the context of the assembly. NX Automotive Supplier bundles include tools for building and manipulating assembly structures. The use of interpart relationships enables the creation of parametric assemblies which capture and preserve the design intent whenever changes are made.

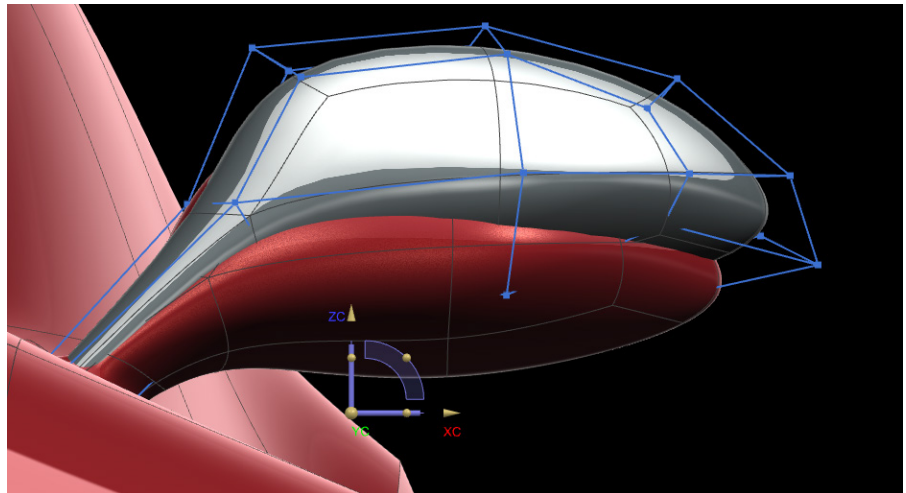
Assemblies can be easily traversed using the assembly navigator in conjunction with intelligent component searching capabilities. NX also supports creation and use of flexible assemblies and parts enabling different sizes and configurations of components, such as springs and pistons.

#### **Advanced assembly modeling**

The advanced assembly modeling capabilities in the NX Automotive Supplier Engineering bundles enable you to simplify components or subassemblies into a single lightweight solid, to enclose assembly geometry in an envelope of planar faces, to partition assemblies into meaningful regions, and to manage weight and other mass properties of components and assemblies.

**Basic freeform modeling** provides the shape modeling application that supports the creation of complex surface and solid models. Basic freeform modeling capabilities include:

- Creating 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



*NX Realize Shape makes it easy to explore complex shapes at the concept stage.*

- 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 freeform modeling** extends surface modeling to include complex filleting, blends and transition surfaces. Guided assistance accelerates the creation of solids from sheets, and the software also helps create surfaces from external point, poles and point cloud data. It includes general-purpose design and manufacturing sweeps and flanges. Designers have full control over surface shape with tools such as direct point and pole manipulation, as well as boundary, degree and stiffness controls. Surfaces can be trimmed, extended, offset, combined, divided or enlarged. Advanced surface analysis tools provide real-time graphical feedback that assists in analytical and visual evaluation of surface quality.

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

**NX Realize Shape™** software is an exciting concept design method that uses advanced subdivision shape creation. It is intuitive: you can create a shape based on a cage that can be extruded, swept, lofted, revolved, copied, and more. The cage faces can be created from curves or polylines and subdivided as much as you want to give you greater control with smooth transitions. The end product is high-quality B surfaces in an editable NX feature. This allows for rapid conceptualization of ideas without the need for expert knowledge. NX Realize Shape can be used in combination with or alongside other surfacing and design tools. NX Realize Shape is a complete set of tools for subdivision modeling.



**Rapid prototyping** enables users to automatically output/export model data in the 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.

**Rendering** provides users with rendering tools to communicate designs clearly and to create 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.



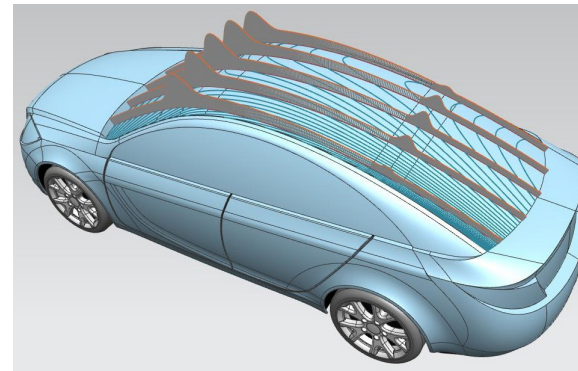
*High-quality rendering lets you see what the product will really look like.*

**Visualize shape** provides dynamic, real-time rendered display with materials, lighting and shadows, and environments. This capability enables you to see what models will actually look like in the vehicle environment in real time, with no delays for rendering, enabling better decision-making early in the design process.

**User-Defined Features** provide an interactive means to capture and store families of parts for easy retrieval and editing using the concept of a user-defined feature (UDF). Users can take an existing parametric solid model and establish the relationships between parameters, define the feature variables, set default values and decide the general form that the feature will take when invoked. Once they have been created, these UDFs reside in a library that can be accessed by any designer. After a UDF is added to a design model, any of its parameters can be edited and its behavior will conform to the design intent specified at creation. UDFs save time by reducing rework and reduce the chance of error by making it easier to re-use existing, proven designs.

**WAVE control** makes it easier to model assemblies of complex systems. It enables automatic propagation of change throughout the engineering process, including conceptual design, design for manufacturing, and manufacturing "part in process." WAVE enables users to identify critical design variables that drive product design, and then capture them in an associative control structure. Changes to the key design variables automatically update the top-level system and all subassemblies and components. NX WAVE control facilitates a high-level, systems-oriented design process that enables concurrent engineering and promotes design re-use and standardization of the design process.

**NX Body Design** is specifically developed for the automotive industry. It is a collection of assistants and advisors tailored to the specialized role of automotive Body-In-White (BIW) design. NX Body Design provides designers and engineers with out-of-the-box processes that are customizable and allow quick and simple creation and validation of BIW parts.



*NX includes advanced curve and surface analysis tools.*

**NX General Packaging** is a collection of automotive-specific tools that automate many of the tasks associated with the mechanical and occupant packaging of a vehicle. General Packaging provides tools for checking a vehicle for SAE standards compliance and local country regulations. It also includes SpaceFinder, which measures the inner volume enclosed by a complex shape or assembly, such as an automotive interior.

**Packaging for review** with XpresReview is an electronic design review process that enables NX users to package together all the files needed for a design review. These files can include the model, drawings, and any other documents needed for collaboration, such as text documents, spreadsheets, or graphics files. XpresReview makes communicating with OEMs faster and less error-prone.

**GRIP program execution** provides the capability to run previously compiled Graphics Interactive Programming (GRIP) applications. Users can run GRIP applications from interactive NX or as a GRIP batch program. GRIP programs enable users to customize NX to suit their particular needs.

**Knowledge Fusion sharing** is a fully integrated knowledge-based engineering (KBE) tool that permits knowledge-based extension of NX by the end user. Knowledge Fusion sharing enables designers to share and re-use powerful applications that take advantage of engineering knowledge. These applications support the capture and re-use of design intent and user intelligence to increase design speed and productivity while intelligently controlling change propagation.

**NX Product Template Studio** is a codeless tool that enables users to wrap a design for re-use. Using simple drag-and-drop methods, an NX designer can greatly simplify interaction with useful parametric models by quickly and easily wrapping a descriptive user interface around the design. This packaged template can then be easily re-used by subsequent designers. With the ability to include product manufacturing information, drawings, validation checks, kinematics or dynamics analyses, finite element analyses and more as part of a product template package, Product Template Studio is a powerful way to capture robust, self-validating modular packages of product design in an easily re-usable form.

### Process-specific modeling tools

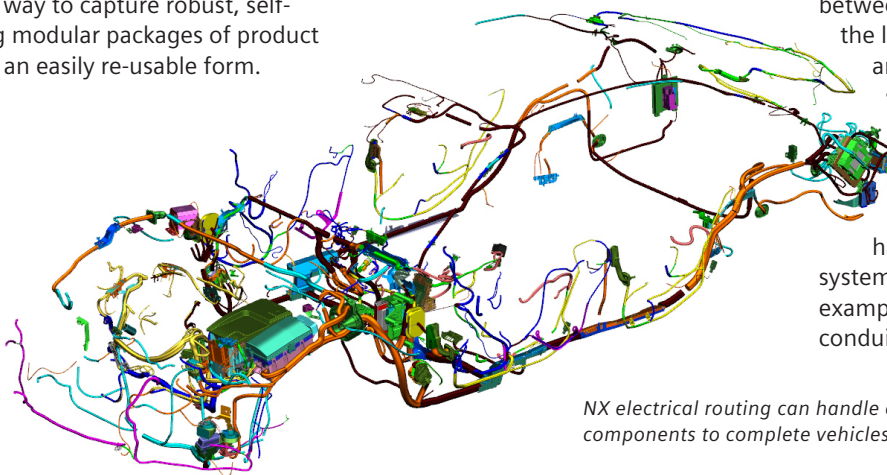
**Sheet metal design** is a solids-based application that is focused on design for manufacturing of sheet metal parts. Users can create sheet metal component models using feature-based design tools for tabs, flanges and other typical features. They can define forming tables and bend sequence tables, 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.

**Advanced sheet metal design** contains tools for designers who model complex parts, including flanges along curved or complex faces that cannot be formed without material deformation. NX Advanced Sheet Metal includes features for designing both straight brake and complex formed parts. Some capabilities of Advanced Sheet Metal Design include the ability to create flanges to match an existing complex surface with options to infer length from the reference surface. Users can also unform these advanced flanges to add cutouts across bends or other features. Users can also unform complex geometries, including non-sheet metal parts, to an alternate shape and quickly join two separate sheet metal bodies with a bridge using different bridge types.

**Weld creation and analysis** with NX Weld Assistant enables users to model material joining through fusion welds, mechanical connections, and solid-state connections. This includes edge, groove, fillet, plug, spot, and seam welds as well as beads, tape, dollops, and clinches. NX Weld Assistant also provides information on the welds and connections to help perform Finite Element Analysis of the assembled product as well as validation checking with NX Check-Mate. NX Weld Assistant facilitates downstream re-use by automatically creating appropriate 2D drafting documentation and annotation based upon the 3D weld feature.

**Basic routing** enables designers to create and edit pipe, tube and cable runs. The routing capabilities also enable intelligent part selection and placement of standard components such as elbows and tees. The routing capabilities can be extended by adding discipline-specific functionality for advanced applications including cabling, piping and tubing and heating, ventilation and air conditioning (HVAC).

**Electrical and mechanical routing** is an integrated suite of tools that facilitate the entire design process for routed systems, including wire harnesses, cables, piping, tubing, conduit and raceways. These process-specific tools reduce detailed design time, improve product quality and transfer product information seamlessly between the logical design, between the logical design, physical design, analysis, manufacturing and service sectors. Electrical routing tools provide electrically smart features and functions to automate the design, modification and analysis of wire harnesses. Mechanical routed system design includes tools and example libraries for tubing, piping, conduit and raceways.



*NX electrical routing can handle everything from the smallest components to complete vehicles.*

### Drafting and Annotation

**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 dimensioning, symbols, tabular notes, sheet layout and placement of standard orthographic and auxiliary views, automatic view creation from the 3D model, hidden line processing and automatic parts list generation. Templates that include drawing borders and view layouts can be dragged and dropped into models to automate much of the work associated with manually created drawings. Drafting tools can be configured to comply with the drawing standard selected by the user – ANSI, ISO, JIS, DIN, GB and ESKD.

**Product and Manufacturing Information (PMI)** allows you to store geometry, tolerance and dimensioning information directly within the 3D model, rather than on a 2D drawing. NX offers a 3D annotation tool that captures and associates PMI to the 3D model, and complies with all the major concepts and requirements defined in the standard for 3D product definition

(ASME Y14.41 and ISO 16792 TC 10). PMI supports the creation of dimensions, tolerance features, weld and surface symbols, material specification notes, part identification labels and a number of other manufacturing and process-related annotations. Because the PMI is created in the 3D CAD model and directly associated to objects in the part, the information can be easily re-used by a number of downstream processes, from the 2D drawing to final article inspection and buyoff. Incorporating PMI during the design process can enhance and shorten the design cycle through better communication, fewer errors, streamlined design and manufacturing processes and faster change management.

**HTML publishing** enables users to publish design data in web-ready HTML format to create 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.

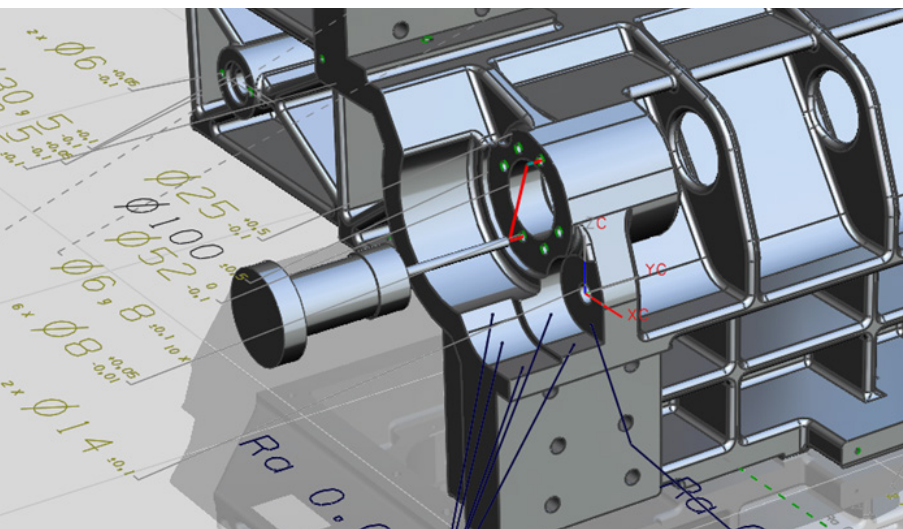
### Product Validation

**Optimization wizard** helps users understand which design parameters are most important to meeting 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 meet their design goals. It also enables designers and engineers to capture engineering requirements, automate the exploration of design alternatives and automatically identify optimized solutions.

**Basic model validation** validates product designs with a model quality assurance checking utility, NX Check-Mate, which evaluates parts, assemblies and drawings to check that:

- Files comply with corporate data quality standards
- Best practices for modeling and assemblies have been employed
- Drawings comply with international standards and corporate documentation best practices
- Poor quality geometric data imported from other systems is quickly uncovered and remedied before it becomes a larger problem (for example, mismatched edges, tiny gaps in faces and non-manifold conditions common in lower-precision modeling tools).

Companies can use the validation checking results to establish product quality metrics and pre-emptively head off quality issues in the earliest stages of the development process.



With PMI you can embed manufacturing information directly in the 3D model.



Validation checks embedded in a design can also help notify a designer if the value of a model expression ever strays from within appropriate limits. These validation checks can be created on-the-fly or can be linked to external sources of design requirements, such as Excel® spreadsheets.

**Molded part validation** enables designers to check the moldability 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, and other factors that can compromise molding. It provides designers with an easy visual check of core and cavity sides that does not 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 cannot 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

**Sheet metal formability** analysis with NX One-step Formability Analysis is a state-of-the-art finite element method-based unforming, flattening and analysis solution for tool and die makers. By combining industry knowledge, best practices and automation, the formability analysis enables the rapid creation of flattened blanks and preforms from complicated freeform sheet metal part geometry. With the capability for unforming and flattening the most complex and challenging sheet metal components, One-step Formability Analysis helps tool and die makers save time by effectively validating designs for thinning, stress/strain and springback.

**Curve and surface quality analysis** checks the quality of curves and surfaces according to the criteria described in the VDA 4955 standard.

**HD3D Visual Reporting** software blends information of interest from the company's data sources directly into the 3D product design environment. This helps users make easier unambiguous assessments, interpret information more accurately and synthesize product and process data rapidly into correct design decisions. It correlates information from multiple disjointed data sources into a single visual interactive environment that delivers product analytics directly to design teams

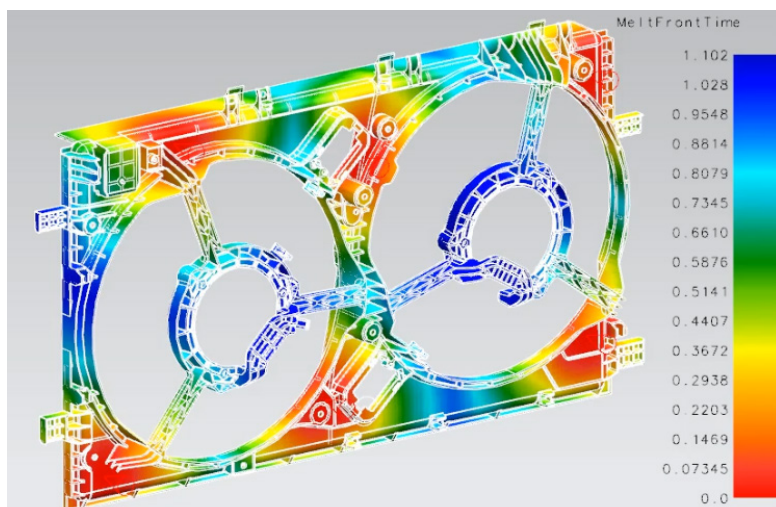
**HD3D validation checking** provides an automated, customizable tool that helps companies proactively improve product quality. Using NX Check-Mate, this capability provides intuitive, visually engaging HD3D tools to assist in identifying and resolving issues in designs. By automatically and continuously ensuring that CAD data and product designs adhere to industry, customer or company standards, HD3D validation checking helps engineering professionals produce their deliverables right the first time.

#### Translators

**Data exchange** provides capabilities for translating data into and out of NX in standard exchange formats, including IGES, STEP AP203, STEP AP214, DXF/DWG and 2D exchange. These translators include geometry repair and simplification capabilities to ensure the most useful data possible. All of the translators can be run externally from NX or directly inside NX from "File Import/Export" and "File Open/Save As" or from the command line, making them customizable to any workflow.

#### Simulation

**NX Stress and Vibration Wizards** are designed to be fast and simple to use, and bring simulation capabilities to all designers using NX. Developed with non-expert CAE users in mind, the wizards provide clear and concise guidance throughout the process, up to and including the creation of web-based reports. Designers benefit from the experience of advanced analysts, while ensuring that work does not need to be repeated and can be leveraged and extended throughout the organization.



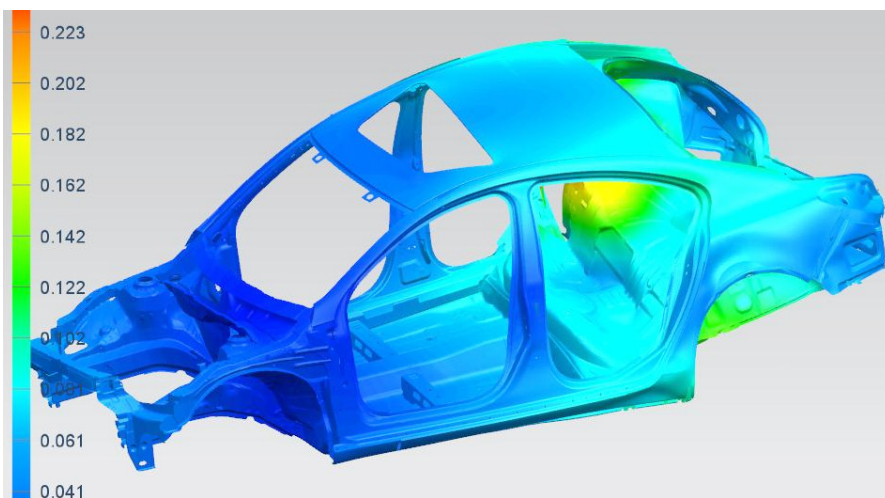
Mold analysis enables you to quickly identify problems with your molded part designs.



NX Advanced Simulation software is a modern, multidiscipline CAE environment for advanced analysts, workgroups and designers that need to deliver high-quality performance insights in a timely fashion to drive product decisions. NX Advanced Simulation integrates best-in-class analysis modeling with the power of an integrated NX Nastran® software solver for basic structural analysis. NX Advanced Simulation also forms a foundation for performing additional solutions including advanced structural, thermal, flow, engineering optimization and multiphysics analyses.

#### Data Management

Teamcenter NX CAD data management and NX Embedded Client give designers the ability to manage data from inside the NX environment. They support access to Teamcenter via the NX user interface, enabling functionality such as vaulting, check-in/check-out, revision management, attribute synchronization, and searching. They also include translation capabilities for generating visualization files. With Teamcenter NX CAD data management, users can manage their in-house data easily from within NX.



A key advantage of NX is fully integrated CAE capabilities.



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