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NX IS AN INTEGRATED SOLUTION FOR PRODUCT DESIGN, DEVELOPMENT AND MANUFACTURING

Abstract: *Siemens NX software is an integrated design, engineering and manufacturing solution that helps you deliver quality products faster and more efficiently. Integrated CAD / CAM / CAE: Intelligent Solutions, Better ProductsNX offers key capabilities for fast, efficient and flexible product manufacturing. This article discusses the advantages and disadvantages of NX.*

Key words: *Software, Siemens NX, design, engineering, manufacturing, quality products, CAD, CAM, CAE, Better ProductsNX.*

Language: *English*

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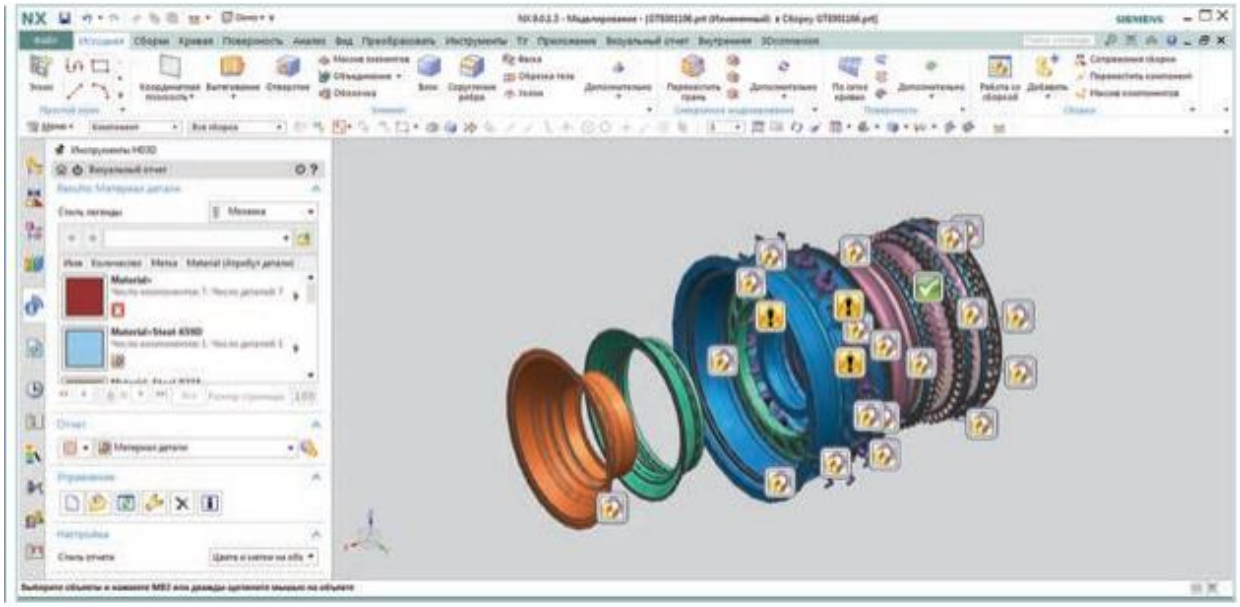
Introduction

The NX ™ software product is an integrated CAD system that offers a set of solutions for design and production preparation tasks and contains engineering analysis tools. NX is the result of the merger in 2002 of Unigraphics and I-deas, which were already the leading CAD systems in many industries at the time. The origins of Unigraphics, the first release in 1973, were from the aerospace industry. The I-deas package, released in 1982, was developed under the leadership of companies in the automotive industry. Currently, the NX system is implemented in most industries and in many companies it has become a standardized solution for automating the processes of design and technological preparation of production. The NX system is based on the Parasolid geometric kernel from Siemens PLM Software and is a set of applications divided into the following areas: NX CAD - tools for 2D and 3D design of parts and assembly units, as well as preparation and release of design technological documentation. NX CAM - tools for automating the creation of programs for CNC machines, managing tool libraries, setting up postprocessors and simulating machining based on the created program. NX CAE is a set of applications for the automation of engineering calculations and simulation of physical processes based on electronic models of assemblies and parts of a product being developed. The suite of applications included in NX CAE is based on the NX Nastran finite element solver and offers advanced tools for preparing simulation models and processing the results obtained. The use of a single platform for applications from various fields allows you to significantly optimize the data flows transmitted between specialists, and to avoid unnecessary translation processes from one system to another. The model developed in NX CAD applications is used as a basis for work in NX CAE

and NX CAM applications. This provides a concept of work using a master model. This means that the original model serves as a data source for consumers, but at the same time they work with its associated copy. On the one hand, this makes it possible to guarantee the author of the model its safety, and on the other hand, it allows consumers to track all the changes that are made to the model.

Another undoubted advantage of a single platform is the unification of the interface and the availability of the same tools for specialists in various fields. This greatly simplifies the learning process and avoids duplication of the same toolkit for specialists in different fields. NX CAE and NX CAM users have access to advanced modeling tools for NX CAD applications. This makes it possible to quickly and optimally adapt the resulting associative model to the needs of a particular specialist and at the same time not lose connection with the original data source. As a top-level CAD system, NX is focused on automating the design and manufacturing of the final product. This distinguishes it from systems focused on intermediate stages, such as issuing paper documents or preparing programs for CNC machines. The ideology of work in the NX system is based on an electronic model of a product, and all tools for automating development processes are based on an electronic model of a part or assembly unit. NX makes it possible to implement a complete description of the product being developed in an electronic model and use this description at all stages of the development process. The model serves as a data source for the creation and release of documentation, calculations, tooling development, preparation of marketing materials and other main and auxiliary processes.

The NX system implements the concept of an adaptive interface that can adapt to the needs of any user, depending on the tasks performed and the level of system development. For easier mastering of the system and to differentiate the functionality for the needs of different categories of specialists, NX offers a role mechanism. Each role includes functionality in relation to a task or corresponds to a certain level of mastering the system functionality by the user. For novice users, the system offers a role with a minimal set of commands and options, which allows you to quickly start working with the system. And for more advanced users, the corresponding role gives access to advanced command options and opens additional functions for work. The system interface itself can be presented in the form of a standard Windows architecture with floating toolbars, as well as in a more modern form, based on a ribbon menu with advanced customization functions. For more flexible work, NX provides a two-way logic of work - the user can call a command and select geometric objects, and vice versa - when selecting an object, the system offers the user a set of commands that are applicable to this type of object. This significantly speeds up the user's work and allows him to think about the immediate task, and not about the system interface. For quicker mastering of the NX functionality by users moving from other systems, the interface has an interactive command search function. By entering a keyword, the user immediately sees where this or that command associated with the entered word is located. Key words can be either a desired action or a geometric object, or the name of a command from other CAD systems - in this case, NX will show its closest analogue.



Picture 1

From an architectural point of view, the NX system consists of a set of applications and modules that are focused on solving problems in a specific area from the CAD / CAM / CAE directions. The user himself chooses the application for work in accordance with what he needs and what tools he needs. The availability of an application or tool is subject to license availability. For optimal configuration of workplaces, NX has a flexible licensing system. There are several basic configurations of applications and modules, which are typical workstations, to which additional modules can be connected to expand the functionality. A large set of built-in and plug-in translators allows for two-way data exchange with other CAD systems, with different levels of data transfer.

Electronic mock-up of a product is one of the key links in the construction of an end-to-end design and production process from the technical specification to the finished product. It depends on how fully and accurately the electronic layout will describe the developed layout, what tasks can be solved based on the electronic description of the product, and how deeply it will turn out to integrate all development participants into a single environment. If earlier the use of CAD systems was limited only to the creation of 3D models for linking and creating drawings, now the main goal of introducing and using top-level CAD systems is to provide all participants in the development with the necessary information about the product to solve their problems. Naturally, this task is solved not only by the CAD system in which the data is created, but also by the life cycle management system. In the Siemens product line, this is the Teamcenter® PLM system, which has an integration module with the NX system. Realizing that product design is not limited to 3D modeling, Siemens PLM Software offers an integrated suite of applications in NX and Teamcenter systems that cover all major phases of product design and manufacturing preparation: Requirement Development. The combination of Teamcenter PLM's requirements management capabilities and parametric modeling tools is transforming the way you define and control requirements. The terms of reference is not just a set of documents, but related parametric and textual requirements that are integrated into an electronic layout. This provides tracing between the parts of the electronic layout in the NX system and all the requirements that apply to these parts. When the technical specifications or regulatory documents change, the project participants see which specific unit or part these changes apply to. The feedback also works - the characteristics of the developed electronic layout of the unit or part are compared with the parametric constraints defined in the terms of reference. If the permissible limits are exceeded, the system signals this. Conceptual study of the product. For this stage, NX offers a full range of 2D and 3D modeling tools, with the help of which the main design options can be worked out and documented. In addition, the NX system has a special application, Mechatronics Concept Designer, designed to conceptually work out the components of the product, determine the relationships between them and the logic of the mechanisms included in the product.

Product layout and decomposition. To solve layout problems, the user is offered basic and advanced capabilities of the Assembly module in combination with modeling tools. It is also possible to use optimization and control tools. Decomposition of the product is supported by the functionality of the WAVE inter-model relationship management module, which allows you to define parametric and geometric dependencies between all components. Detailed study. To detail the components of a product, the NX system provides the user with a toolbox of several applications with the help of which an accurate model of a part or assembly is built. In this case, the model can be based on external source data, determined at the stage of decomposition, and meet the requirements specified at the stage of forming the technical task. Engineering analysis. For carrying out engineering analysis tasks, NX offers two levels of solutions - for applied calculations for designers and constructors, as well as for specialists in calculation departments. In the first case, simple wizards are used that allow you to carry out a preliminary simple calculation of the product being developed in a step-by-step mode. In the second case, we are talking about a set of applications included in the NX CAE direction. In addition to these solutions, Teamcenter provides the Teamcenter Simulation module, which closes design data management tasks and provides a link between design data and engineering calculations. Assembly and analysis. Both NX and Teamcenter applications provide tools for creating assemblies and analyzing collectability. In addition to the formation of the assembly structure and its geometric representation, the user has tools at his disposal that allow you to work out the assembly sequence, analyze assembly and conduct a number of other analyzes. Eliminating errors in the electronic layout helps to avoid corrections and modifications of the product when it has already been put into production and when the cost of corrections is already orders of magnitude higher than at the development stages.

Tooling development. The development of technological equipment is greatly simplified in the presence of an electronic model of the product.

When designing tooling elements, associative links to electronic models of the corresponding units and parts are used. This makes it possible to develop tooling already in the early stages of the readiness of the electronic layout and to quickly track the changes taking place in it. And with the effective use of associative links, it becomes possible to quickly adapt

existing tooling elements for products that have a certain degree of similarity with those already developed. For tooling professionals, NX offers a full suite of modeling tools as well as a range of specialized die and mold design applications.

Preparation of programs for CNC machines. The tasks of this stage are closed by a set of applications included in NX CAM. At the same time, the possibilities for using NX for production preparation are not limited only to the nomenclature of parts manufactured on CNC machines. The use of modeling tools and synchronous technology allows you to create operational sketches for parts manufactured on universal equipment, as well as prepare models for individual operations on CNC machines. Preparation of documentation. The NX system offers several ways to create design and engineering documentation using the Drafting and Specifications applications. The user can create both classic drawings based on electronic models of parts and assemblies, and annotate directly 3D models, linking dimensions and annotations directly to geometry. Quality control. For quality control, NX offers a number of tools that allow you to check the electronic layout and documentation for various kinds of inconsistencies. This can be control of design, control of collection, control of the possibility of manufacturing a part with reference to a specific technological process, and many other types of control that accompany the development process. In addition, NX allows you to link information to the electronic model, which is subsequently used in Teamcenter and Technomatix applications for collectability analysis or dimension chain calculation.

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