

APPLICATION AND EFFICIENCY OF THE SIEMENS NX SIMULATION MODULE IN INDUSTRIAL ANALYSIS

<https://doi.org/10.70728/conf.v2.i02.014>

Gulchehra Tulanovna Kholmirezayeva

Lecturer, Department of Software Engineering, Andijan State University named after Zahiriddin Muhammad Babur

Abstract This thesis presents the capabilities, practical applications, and main stages of the analysis process within the Simulation (Analysis and Testing) module of Siemens NX software. In modern industry, computer-aided analysis plays a vital role in improving product reliability, creating energy-efficient designs, and reducing manufacturing errors. Using the Simulation module, mechanical stresses, deformations, thermal and dynamic characteristics of structures are modeled based on numerical methods, particularly the Finite Element Method (FEM). The research elaborates the stages of the analysis process — model preparation, adjustment of calculation parameters, solution generation, and result evaluation. Additionally, structural analysis results based on a practical example are provided, and future development directions of the Simulation module, such as multiphysics analyses and AI-based optimization, are discussed.

Keywords: Siemens NX, Simulation, analysis, testing, finite element method (FEM), mechanical analysis, thermal analysis, stress, deformation, modeling, CAE, CAD/CAM integration.

In today's modern industrial and engineering projects, pre-evaluating the reliability, safety, and operational efficiency of manufactured products is one of the most important tasks. From this perspective, computer-aided analysis, or simulation tools, play a significant role in this field.

Siemens NX is an industry-grade integrated CAD/CAM/CAE platform, within which the Simulation module enables the execution of mechanical, thermal, dynamic, and other analyses at the design stage of product development.

The NX Simulation module supports the following main analyses:

- **Static Stress Analysis** – determines stress, deformation, and displacement of a structure under applied loads.
- **Thermal/Heat Transfer Analysis** – models heat flow and temperature distribution.
- **Modal Analysis** – identifies vibration characteristics and resonance frequencies.
- **Dynamic/Transient Analysis** – studies system motion and load effects that vary over time.

- **Fatigue Analysis** – evaluates material fatigue and long-term failure risks.
- In some cases — **kinematic** and **contact analyses** are also performed.

The NX Simulation module is designed to work fully integrated with CAD data, maintaining parametric and associative relationships throughout the analysis process.

The analysis process in the simulation environment consists of the following stages:

1. **Model Preparation (Preprocessing):** This stage focuses on verifying the CAD model geometry, selecting necessary parts, and defining material properties (Elastic modulus, Poisson's ratio, thermal conductivity, etc.). Boundary conditions (loads, constraints, interactive contacts) are applied, and the mesh generation process (using elements such as tetragonal, hexagonal, tetrahedral, or refined meshes) is controlled.
2. **Adjustment of Analysis Parameters (Solver Settings):** At this stage, static, dynamic, thermal, and other parameters are selected. Iteration limits and convergence criteria are also defined during this configuration process.
3. **Computation (Solving):** Numerical solution methods — primarily the Finite Element Method (FEM) — are applied, and the analysis results are obtained using advanced computational tools.
4. **Analysis of Results (Postprocessing):** Stress distribution and deformation diagrams are analyzed. Special attention is given to visualization through vectors, isolines, and animations. Afterward, the results are verified to ensure their realism and to check the maximum allowable stress limits.
5. **Verification and Optimization:** If the results are unsatisfactory, model adjustments are performed at this stage. Material selection, cross-section modification, or adding reinforcements (ribs, gussets) are among the primary optimization tasks.

Based on the above considerations, an example can be illustrated. For instance, a **static stress analysis** of a metal frame element:

- **Geometry:** Simple section (rectangular frame);
- **Material:** Steel ($E = 210 \text{ GPa}$, $\nu = 0.3$);
- **Boundary Conditions:** The lower part is fixed, and a vertical force of 1000 N is applied to the center of the upper part;
- **Mesh:** Tetrahedral elements;

Results obtained:

- Maximum stress value: $\sim 200 \text{ MPa}$;
- Maximum deformation: $\sim 0.25 \text{ mm}$;
- Stress regions are displayed using colored isolines;

- The result is used to assess structural safety and the possibility of plastic deformation in the material.

Despite being powerful and convenient, the NX Simulation module also has certain limitations that may pose challenges for users. In cases of complex contact (sliding, separation) or large deformations, the accuracy of the solution may decrease. When the mesh is too refined, the matrix size increases, leading to higher computational loads — one of the main challenges. In addition, uncertainties in material models (plastic, viscoelastic, brittle, or neutron materials) and incorrect solver parameter selection may lead to convergence problems.

Modern analyses often require the combination of multiple physical domains (structural + thermal + acoustic), which may require external plugins or additional modules.

Future Prospects:

- Multiphysics analyses can improve the integration of structural, thermal, and electromagnetic domains;
- Generative design and topology optimization can be achieved through AI and optimization algorithms;
- Cloud-based simulations will provide faster solutions for large models;
- Real-time simulations will enable interactive analysis at the design stage.

The above analyses demonstrate that the Siemens NX Simulation module provides accurate evaluation of mechanical, thermal, and dynamic properties in modern engineering systems. The module ensures the reliability, stability, and efficiency of structures at the early design stages. As a result, manufacturing errors are reduced, material usage is optimized, and overall project quality is improved.

The main advantage of the Simulation module is its complete integration with the CAD environment, which automatically reflects design changes in analysis results. Moreover, FEM-based computational algorithms allow users to perform even complex analyses quickly and efficiently.

In the future, the Siemens NX Simulation module is expected to be further enhanced through AI-based optimization, multiphysics analysis, and cloud computing technologies. This will elevate the level of digital modeling in engineering and open new opportunities for developing high-quality, energy-efficient products.

REFERENCES:

1. Kholmiraeva, G. T. (2023). *Siemens NX is the Perfect Solution to Automated Design Problems*. IJOT. Retrieved from <https://journals.researchparks.org/index.php/IJOT/article/view/4314>

2. Siemens PLM Software. (n.d.). *NX Design Simulation Fact Sheet*. Retrieved from https://www.plm.automation.siemens.com/cz_cz/Images/nx%20design%20simulation%20fs%20W%201_tcm841-4361.pdf
3. Sadchikova, G. M., et al. (2017). *Application of NX Siemens PLM Software in Educational Institutions*. AIP Conference Proceedings. doi:10.1063/1.4972454
4. Li, X. (2018). *Modeling and Simulation of Five-Axis Virtual Machine Based on Siemens NX*. AIP Advances, 1955, 030044. doi:10.1063/1.5003432
5. Banaś, W., Herbuś, K., Kost, G., Nierychlok, A., Ociepka, P., & Reclik, D. (2013). *Simulation of the Stewart Platform Carried out Using the Siemens NX and NI LabVIEW Programs*. Advanced Materials Research, 837, 537–542. doi:10.4028/www.scientific.net/AMR.837.537
6. Okafor, C. J. (2025). *Geometric Analysis of Gutter Bracket Using Different NX Siemens Solvers for Linear and Nonlinear Investigation*. Complexity Analysis and Applications, 2(1). <https://doi.org/10.48314/caa.v2i1.35>

