ENGINEERING SIMULATION

The Comprehensive Guide to Autodesk Nastran In-CAD 2017: Technical Implementation, FEA Principles, and Engineering Strategy

Published: January 29, 2026 | Index ID: #694

The evolution of digital prototyping has fundamentally shifted from simple geometric modeling to the complex simulation of physical behavior. Within this transformation, **Autodesk Nastran In-CAD 2017** stands as a pivotal milestone. By embedding the industry-standard Nastran solver directly into the **Autodesk Inventor** environment, engineers gained the ability to perform high-fidelity **Finite Element Analysis (FEA)**without the friction of data translation between disparate software platforms. This integration ensures that simulation is no longer a post-design validation step but a core component of the iterative design process.

The Architecture of Nastran In-CAD

To understand the power of Nastran In-CAD 2017, one must first understand its lineage. **Nastran (NASA Structural Analysis)** was originally developed for the aerospace industry in the late 1960s. Its transition into the commercial sector led to various iterations, including MSC Nastran and NEi Nastran. Autodesk's acquisition of the NEi Nastran solver paved the way for **In-CAD**, a tool designed to bridge the gap between pure CAD users and dedicated simulation analysts.

The 2017 version of the software, specifically build **2017.0.0.27**, introduced significant enhancements in CAD-to-FEA associativity. Unlike standalone solvers, Nastran In-CAD operates as an "Add-In" within Inventor. This means that any modification to the CAD geometry—such as changing the thickness of a plate or the diameter of a fastener—is immediately reflected in the simulation model, preserving constraints, loads, and mesh settings.

Core Mechanics of Finite Element Analysis (FEA)

At its technical core, Nastran In-CAD 2017 utilizes the **displacement-based finite element method**. The software discretizes a continuous physical structure into a finite number of elements (tetrahedral, hexahedral, or shell). Each element is governed by a set of algebraic equations derived from the principle of virtual work. The fundamental equation solved by the Nastran engine is:

[K]{u} = {F}

- [K]: The Global Stiffness Matrix, representing the material properties and geometric configuration of the assembly.
- {u}: The nodal displacement vector (the primary unknown).
- {F}: The externally applied load vector (forces, pressures, moments).

By solving this system of linear or non-linear equations, the software determines the displacement at every node. From these displacements, **strains** and **stresses**(such as Von Mises or Principal stresses) are calculated using the material's constitutive laws.

Technical Specifications and System Requirements

Deploying Nastran In-CAD 2017 requires careful attention to hardware and software compatibility. A common point of confusion for users is the versioning alignment between the CAD host and the simulation add-in. As documented in technical support data, **Nastran In-CAD is strictly linked to the Inventor version.** To run Nastran In-CAD 2017, the host machine must have **Autodesk Inventor 2017**installed. While there were historical workarounds for running older versions of In-CAD with newer Inventor builds, the 2017 release established a rigid synchronization for stability and API compatibility.

Hardware Recommendations for Simulation

While the minimum requirements often mirror those of standard CAD software, FEA is highly computationally intensive, particularly during the **matrix inversion** (solving) and **meshing** phases.

Component	Minimum Requirement	Recommended for Professional FEA
CPU	Intel Core i5 (Dual Core)	Intel Xeon or Core i9 (8+ Cores)
RAM	8 GB	32 GB to 64 GB (Crucial for large matrices)
GPU	DirectX 11 compliant	NVIDIA Quadro or AMD Radeon Pro (4GB+ VRAM)
Storage	HDD (100GB Free Space)	NVMe SSD (For fast I/O during temporary file writing)

Installation and Licensing Workflow

One of the critical operational tasks for a CAD administrator is the activation of the software. For the 2017 version, the process follows a specific sequence. After the primary installation of Inventor 2017, the Nastran In-CAD installer is executed. Upon the first launch, the user must navigate the licensing dialogue to ensure the solver is recognized.

Step-by-Step Activation Procedure:

1. Launch Inventor 2017: Navigate to the 'Environments' tab on the ribbon.
2. Access Nastran In-CAD: If the tab is missing, check the 'Add-Ins' manager to ensure the software is loaded and set to 'Load Automatically'.
3. License Selection: Upon clicking the Nastran environment, a dialog box will appear. Select 'Product Information'.
4. Network vs. Standalone: Specify whether the license is being pulled from an Autodesk Multi-User License server or a Single-User serial number.
5. Verification: Ensure that the 'Analysis' tree appears on the left side of the screen, confirming the solver is ready for input.

Types of Analysis in Nastran In-CAD 2017

Unlike basic stress analysis tools included in standard CAD packages, Nastran In-CAD 2017 offers a comprehensive suite of analysis types. This allows engineers to simulate not just static loads but dynamic, thermal, and buckling scenarios.

1. Linear Static Analysis

The most common form of simulation, used to determine displacements and stresses under constant loads. It assumes the material remains within the **elastic region**(Hooke’s Law applies) and that displacements are small enough that the stiffness matrix [K] does not change during the calculation.

2. Modal Analysis (Normal Modes)

This is used to determine the natural frequencies and mode shapes of a structure. Identifying **resonance frequencies**is critical in preventing mechanical failure in vibrating environments, such as automotive engine components or rotating machinery.

3. Linear and Non-Linear Buckling

Buckling analysis predicts the critical load at which a slender structure will suddenly lose stability. Linear buckling

(Eigenvalue) provides a theoretical safety factor, while non-linear buckling accounts for initial imperfections and material yielding.

4. Thermal Stress Analysis

This analysis calculates the distribution of temperature and the resulting thermal expansion or contraction. It is vital for components like heat exchangers or electronic enclosures where constrained thermal expansion can lead to massive internal stresses.

5. Non-Linear Static and Transient Analysis

This handles scenarios where the material yields (Plasticity), large deformations occur, or parts come into contact. The 2017 version significantly improved the **Contact Manager**, allowing for more robust handling of 'Sliding' and 'Separation' contact types between assembly parts.

Feature Comparison: Inventor Stress Analysis vs. Nastran In-CAD

Many users wonder why they should upgrade from the basic stress analysis tool built into Inventor to the full Nastran In-CAD suite. The following table highlights the technical disparities.

Feature	Inventor Stress Analysis	Nastran In-CAD 2017
Solver Engine	Proprietary Linear Solver	Industry-Standard Nastran Solver
Non-Linear Materials	No	Yes (Plasticity, Hyperelasticity)
Fatigue Analysis	No	Yes (Cycle-based life prediction)
Element Control	Automatic Tetrahedral	Full control (Tetra, Shell, Beam, Rigids)

Advanced Meshing Strategies

The accuracy of an FEA result is directly proportional to the quality of the mesh. In Nastran In-CAD 2017, users have access to sophisticated meshing tools. **Mesh Convergence** is a critical concept here; an engineer must ensure that the results (like max stress) do not significantly change when the mesh is refined.

Parabolic vs. Linear Elements

Nastran In-CAD defaults to **parabolic elements** (10-node tetrahedrals for solids). These elements include mid-side nodes, allowing the element edges to curve. This is essential for accurately capturing the geometry of fillets and holes. Linear elements (4-node tetrahedrals) are computationally “cheaper” but are prone to **shear locking**, leading to artificially stiff results.

Local Mesh Controls

Instead of refining the mesh for the entire model—which increases solve time exponentially—engineers should apply **Mesh Controls** to specific edges or faces. For example, applying a refinement to a bolt hole where stress concentrations are expected while maintaining a coarse mesh on the bulk of the part is an industry best practice.

Troubleshooting and Common Failure Modes

Working with Nastran In-CAD 2017 can occasionally present challenges, particularly during the solving phase. Here are the most frequent issues and their technical resolutions.

1. Fatal Error 9050 (Insufficient Memory)

This occurs when the solver attempts to store the stiffness matrix in RAM but exceeds available capacity. **Solution:** Increase the Windows Virtual Memory (Pagefile) or switch the solver setting to 'Sparse' or 'Out-of-Core' to utilize disk space.

2. Rigid Body Motion (Mechanism Error)

If a model is not fully constrained in 3D space (X, Y, Z translations and rotations), the solver cannot find a static equilibrium. **Solution:** Use the 'Check for Unconstrained Components' tool or apply weak springs to identify which part is 'flying away'.

3. Mesh Failure on Complex Geometry

Often caused by 'Sliver Faces' or 'Zero-length Edges' in the CAD model. **Solution:** Use Inventor’s 'Repair Environment' to simplify the geometry or remove tiny features like embossed text and extremely small fillets that do not contribute to the structural integrity.

Field Guide: Executing a Professional Simulation Workflow

To ensure repeatable and defensible results, a standardized workflow should be followed for every simulation project using Nastran In-CAD.

Step 1: Idealization

Do not simulate the raw CAD model if it is overly complex. Simplify the geometry. Replace thin-walled solids with **Shell Elements** and long, slender components with **Beam Elements**. This reduces the number of Degrees of Freedom (DOF) and improves accuracy.

Step 2: Material Assignment

Assign accurate physical properties. For linear analysis, **Young’s Modulus (E)** and **Poisson’s Ratio (;Ò** are mandatory. If performing thermal analysis, the **Coefficient of Thermal Expansion** and **Thermal Conductivity** must be defined.

Step 3: Boundary Conditions (Constraints and Loads)

Constraints represent how the part is held. Loads represent the environment. A common mistake is applying a 'Fixed' constraint to a face that should be allowed to rotate. Use **Pin Constraints** or **Frictionless Constraints** where appropriate to avoid over-stiffening the model.

Step 4: Post-Processing and Validation

Once the solver finishes, do not just look at the 'Red' spots in the stress plot. Check the **Displacement Plot** first. Does the deformation look physically intuitive? If a beam is bending the wrong way, your constraints or load directions are likely incorrect. Finally, compare the peak stress against the material's **Yield Strength** to calculate the Factor of Safety (FoS).

The Broader Impact of Integrated Simulation

The transition of Nastran In-CAD 2017 into what is now known as **Autodesk Inventor Nastran** reflects a broader trend in engineering: the democratization of high-end simulation. By moving the solver into the CAD interface, the barrier to entry for performing complex FEA has been lowered. However, this ease of use comes with a responsibility for the engineer to understand the underlying physics. A tool as powerful as Nastran can produce precise-looking results that are entirely inaccurate if the boundary conditions or material models are flawed—a phenomenon known in the industry as "Garbage In, Garbage Out."

The 2017 version remains a robust choice for organizations maintaining legacy workflows or those operating on permanent licenses from that era. Its ability to handle everything from simple static stress to complex non-linear contact makes it a versatile asset in any mechanical engineer’s toolkit. As the industry moves toward generative design and AI-driven optimization, the fundamental principles of FEA established and refined in Nastran In-CAD 2017 continue to serve as the bedrock of structural validation and innovation.

Baca Juga (Artikel Terkait)

- **Comprehensive Guide to Finite Element Analysis (FEA) for Free Fall Drop Testing: Engineering Principles and Applications**
  URL: http://alghafgolden.com/finite-element-analysis-drop-test-simulation-guide.pdf
- **The Comprehensive Guide to Abaqus Documentation: Technical Architecture, FEA Methodologies, and Implementation Strategies**
  URL: http://alghafgolden.com/comprehensive-guide-abaqus-documentation-technical-fea-strategies.pdf
- **Mastering Autodesk Nastran In-CAD 2015: A Technical Deep Dive into FEA Integration and Structural Analysis**
  URL: http://alghafgolden.com/mastering-autodesk-nastran-in-cad-2015-technical-guide.pdf
- **The Engineering of Fluid Dynamics: A Comprehensive Technical Guide to Bladeless Technology and CFD Simulation in Ansys**
  URL: http://alghafgolden.com/ansys-fluent-bladeless-technology-cfd-guide.pdf

Tags: #Autodesk Nastran #Inventor 2017 #FEA Analysis #Finite Element Method #Mechanical Engineering #CAD Integration