GENERAL

# autodesk nastran in cad 2017 and autodesk inventor

Published: September 19, 2026 | Index ID: #-

## Introduction

In today's fast paced engineering world, the ability to seamlessly transition from conceptual design to detailed analysis is no longer a luxury—it's a necessity. Engineers and designers rely on robust Computer-Aided Design (CAD) tools to create intricate models, while simulation platforms such as Autodesk Nastran provide the analytical rigor required to validate performance, safety, and compliance. For users of **Autodesk Nastran in CAD 2017** and **Autodesk Inventor**, mastering the integration between these two powerful suites can dramatically accelerate product development cycles, reduce costly iterations, and unlock deeper insight into mechanical behavior.

Whether you are a seasoned engineer looking to refine your workflow or a newcomer exploring the possibilities of simulation driven design, this article offers a detailed, step by step guide to harnessing the full potential of Autodesk Nastran within the CAD 2017 environment and leveraging Autodesk Inventor's intuitive modeling capabilities. We'll cover everything from installation and licensing to advanced simulation techniques, best practices, and real world case studies that illustrate the tangible benefits of this integration.

## Understanding the Core Players

### Autodesk Nastran in CAD 2017

Autodesk Nastran is a high performance finite element analysis (FEA) solver that has evolved from its origins in NASA's structural analysis software. Within the CAD 2017 ecosystem, Nastran is embedded as a comprehensive simulation engine that supports a wide array of analysis types—including static, dynamic, thermal, and nonlinear studies. By integrating directly into the CAD workflow, Nastran eliminates the need for external data conversion, preserving geometry fidelity and ensuring that model updates propagate instantly to the analysis model.

### Autodesk Inventor

Inventor is a parametric 3D CAD application designed for mechanical design, assembly, and documentation. Its user friendly interface, robust feature set, and strong interoperability with other Autodesk products make it a go to choice for engineers across industries. Inventor's native support for Nastran simulation allows designers to perform analysis directly within the modeling environment, fostering a tighter feedback loop between design and validation.

### Why Combine CAD 2017 and Inventor with Nastran?

- Streamlined Workflow – Eliminates manual file exports, reducing errors and saving time.
- Parametric Consistency – Design changes automatically update the analysis model.
- Real Time Feedback – Engineers can see simulation results in the context of the 3 D model.
- Scalable Complexity – Supports large, complex assemblies while maintaining performance.
- Cost Efficiency – Reduces the need for multiple software licenses and training.

## Getting Started: Installation and Licensing

### System Requirements

Before installing, verify that your workstation meets the minimum hardware and software specifications:

- Operating System: Windows 10/11 (64 bit)

- CPU: Multi core processor (Intel i5/i7 or equivalent)
- RAM: Minimum 16 /GB (32 /GB recommended for large assemblies)
- GPU: DirectX 11 compatible graphics card with at least 4 /GB VRAM
- Hard Drive: SSD with 20 /GB of free space for installation, plus additional space for simulation data

### Installing CAD 2017 and Autodesk Inventor

1. Download the Autodesk Desktop App from the official website and sign in with your Autodesk account.
2. Select CAD 2017 from the list of available products and click Install.
3. Once CAD 2017 is installed, repeat the process for Autodesk Inventor.
4. During installation, ensure that the Nastran Simulation Module is selected. This option is typically bundled under the Simulation category.
5. After installation, launch Autodesk Inventor and verify that the Nastran tab appears in the ribbon interface.

### Licensing Options

Autodesk offers several licensing models to accommodate different organizational needs:

- Subscription – Pay monthly or annually for the latest updates.
- Perpetual License – One time purchase with optional maintenance for upgrades.
- Academic License – Discounted rates for educational institutions.

For simulation heavy projects, consider the **Autodesk Nastran Professional** add on, which unlocks advanced analysis features such as nonlinear material models, contact analysis, and multi physics coupling.

## Building Your First Simulation Model

### Step 1: Create a Parametric Part in Inventor

Begin by designing a simple component—such as a bracket or a gear—using Inventor's parametric features. Define dimensions with constraints and parameters so that any future modifications automatically update the geometry.

### Step 2: Assemble Components (If Needed)

For multi piece assemblies, use the **Assembly** environment to position parts with mates. Ensure that all components are fully defined to avoid simulation errors.

### Step 3: Launch the Nastran Simulation Workflow

From the Inventor ribbon, click the **Nastran** tab, then select **New Simulation**. Choose the analysis type that matches your design goals (e.g., *Static Structural*, *Thermal*, or *Modal*).

### Step 4: Define Materials

Assign material properties directly from Inventor's material library. If your design uses custom alloys or composites, create a new material definition by specifying Young's modulus, Poisson's ratio, density, and temperature coefficients.

### Step 5: Apply Boundary Conditions

- Supports – Define fixed, pinned, or roller constraints where the part attaches to the environment.
- Loads – Apply forces, pressure distributions, or thermal loads as required.
- Symmetry – Use symmetry planes to reduce model size without sacrificing accuracy.

### Step 6: Mesh the Geometry

Invoking the Nastran mesh editor, select appropriate element types (e.g., solid, shell, or beam). For complex surfaces, use adaptive meshing to refine critical regions while keeping the overall mesh size manageable.

### Step 7: Run the Simulation

Click **Run** to submit the job to the Nastran solver. Monitor progress in the **Job Monitor** window. Once complete, review the results directly within Inventor.

### Step 8: Visualize and Interpret Results

Use the built in post processing tools to plot stress, displacement, or temperature contours. Employ **Animation** features for dynamic studies, and export data to Excel or PDF for reporting.

## Advanced Simulation Techniques

### Nonlinear Material Models

When dealing with metals that exhibit plasticity or composites with anisotropic behavior, enable the **Nonlinear Material** option. Define yield criteria, hardening laws, and damage models to capture real world responses.

### Contact Analysis

For assemblies involving moving parts, set up contact pairs to model friction, clearance, and pressure distribution. Use **Penalty** or **Lagrange Multiplier** methods depending on the required accuracy.

### Multi Physics Coupling

Combine structural, thermal, and fluid analyses by linking Nastran with other Autodesk simulation tools. For example, use **Autodesk CFD** to compute airflow around a component and feed temperature data into Nastran for thermal structural coupling.

### Optimization Workflows

Leverage the **Design Space Exploration (DSE)** feature to perform topology optimization or parameter sweeps. Define objective functions (e.g., weight reduction, stress minimization) and constraints, then let Nastran iteratively refine the geometry.

## Best Practices for Efficient Integration

### Maintain a Clean Geometry

Remove unnecessary small features, fillets, or weld beads that can inflate mesh size without adding analytical value. Use Inventor's **Simplify** tools to streamline the model.

### Use Parametric Design to Your Advantage

By embedding design variables into the CAD model, you can automatically propagate changes to the simulation. This reduces the risk of mismatched models and speeds up iteration cycles.

### Set Up a Consistent Naming Convention

Label parts, assemblies, and simulation components systematically. For example, prefix material definitions with *MAT_* and boundary conditions with *BC_* to improve clarity.

### Validate Mesh Quality Early

Before running full analyses, check element aspect ratios, skewness, and Jacobian values. Poor mesh quality can lead to solver convergence issues and inaccurate results.

### Document Every Step

Maintain a simulation log that records parameter values, material data, boundary conditions, and solver settings.

This documentation is invaluable for troubleshooting and for future project handovers.

## Troubleshooting Common Issues

### Solver Convergence Problems

- Check for unrealistic loads or boundary conditions.
- Refine the mesh in high stress regions.
- Switch from Penalty to Lagrange Multiplier contact methods if contact stiffness is causing divergence.

### Unexpected Displacements or Stress Results

- Verify material property assignments.
- Ensure that constraints are properly applied and not over constrained.
- Inspect the mesh for element distortion.

### File Compatibility Errors

- Confirm that the CAD file is saved in a compatible format (e.g., .ipt, .iam).
- Use the Rebuild command in Inventor to refresh the model before importing into Nastran.
- Check for missing or corrupted components.

## Real World Case Studies

### Automotive Suspension Component

An automotive manufacturer used Autodesk Nastran in CAD 2017 and Autodesk Inventor to analyze a new suspension arm. By integrating the simulation directly into the design process, the team reduced the component weight by 12 % while maintaining structural integrity. The parametric design allowed rapid iteration of cross sectional shapes, and the optimization workflow identified a topology optimized geometry that delivered the same performance with fewer material resources.

### Medical Device Implant

A medical device company developed a titanium hip implant using Inventor's parametric modeling. The Nastran solver performed a nonlinear, contact aware analysis to predict implant wear under cyclic loading. The simulation revealed critical stress concentrations that were mitigated by adjusting the fillet radius, resulting in a design that passed regulatory fatigue tests in half the time.

### Aerospace Landing Gear


## Baca Juga (Artikel Terkait)

- **advanced engineering electromagnetics solution manual edition hardcover by balanis constantine a published by wiley**
  URL: http://djjsearch.com/advanced-engineering-electromagnetics-solution-manual-edition-hardcover-by-balanis-constantine-a-published-by-wiley.pdf
- **advanced engineering electromagnetics solutions manual**
  URL: http://djjsearch.com/advanced-engineering-electromagnetics-solutions-manual.pdf
- **advanced engineering electromagnetics wiley 1989 grading**
  URL: http://djjsearch.com/advanced-engineering-electromagnetics-wiley-1989-grading.pdf
- **advanced engineering fluid mechanics by biswas**
  URL: http://djjsearch.com/advanced-engineering-fluid-mechanics-by-biswas.pdf