

Inventor Nastran 2020 list of changes

Inventor Nastran 2020.0 is the new name for Nastran In-CAD.

Inventor Nastran is now available to install. From manage.autodesk.com, locate the Product Design & Manufacturing Collection. Click "View items" and locate Inventor Nastran. (Of course, you need to have Inventor 2020 installed before installing Inventor Nastran.)

Here is my list of changes, gathered from the What's New, Release Notes, and issues addressed by development (which includes changes to "In-CAD" and to the Nastran solver that are relevant.)

1	Added a MESH CONVERGENCE tool to automatically refine the mesh until the change in result is below a specified value. For more information, see Mesh Convergence .
2	Added a HYDROSTATIC LOAD to simulate the hydrostatic pressure due to a fluid. This load type should be used instead of the variable load definition. For more information, see Hydrostatic Load . Note the following known issues: <ul style="list-style-type: none">• If an edge is selected for the hydrostatic pressure Fluid Depth Direction, and then if the vector direction is updated manually, the manual vector direction is not shown when the load is edited.• Use a unit vector for the fluid depth direction. Otherwise, the load is scaled by the magnitude of the vector.
3	Added ability to place the simulation results in Vault to store and manage. Due to the potential large size of the simulation files, the user must manually indicate when an analysis is to be placed in the Vault. See the What's New in V2020 for details.
4	<p>The input "Bottom Fiber Distance" did not work properly for shell elements. If the value was changed from the default value, the shell was offset the wrong amount, so the results were for a different arrangement than intended. This has been corrected in 2020, but now the input is the offset of the midplane. You now enter the distance from the mesh to the mid-thickness of the shell.</p> <p>When using a shell model created in a previous version that includes a bottom fiber distance, the input must be corrected when using version 2020. See Shell thickness visualization when using Bottom Fiber Distance in Nastran In-Cad is not what Nastran... for details.</p>
5	Corrected a problem in large models when using solver contact. Some models would end with a T2104 warning as described in Warning T2104 during an analysis with Nastran In-CAD .
6	The "Results > Probe" now shows the correct values when the units are changed. The conversion was applied incorrectly as described in Probe tool not giving correct values in Nastran In-CAD
7	Improvements when working with a Frame Generator model, including: <ul style="list-style-type: none">• Speed improvement to save the file.• The solid parts are hidden so that the frame geometry (lines, end points) can be selected to apply the load and constraints.

	<ul style="list-style-type: none"> The original sketch from which the frame was generated can no longer be selected. Previously, the sketch could be selected which was causing problems described in Load not applied at valid FE mesh location when using Frame Generator interoperability with Nastran
8	The contour legend is now shown even when the maximum and minimum results are the same value.
9	Correct results given when a bolt connector has a preload and the analysis is solved with the PCGLSS solver. See Bolt preload gives wrong results with Nastran for details.
10	Input for material properties, loads, constraints, and so on, now use the unit system set in the Units branch of the model tree. Previously, dialogs would use the Inventor Document units depending on how the dialog was activated as described in Load units are not the same as analysis units in Nastran In-CAD .
11	Input for shell element thickness works when using a comma for the decimal separator. Previously, the value was not saved correctly as described in Shell thickness input does not work in Nastran In-CAD .
12	The default list XY Plots (created for some analyses, such as Normal Modes) will be listed when a previous analysis is activated. See List of XY Plots is missing in Nastran In-CAD for details.
13	Orthotropic materials allow Poissons ratio outside the range of 0.0 to 0.5 for orthotropic material.
14	Stress linearization now shows the results for the current subcase. Version 2019 only showed the results for the last subcase.
15	Fixed E5004 error (stiffness matrix singular) or E5051 warning (excessive solution) that occurred with orthotropic materials when using the PCGLSS solver.
16	Temperatures read from a heat transfer analysis to a stress analysis are now converted to the current unit system. See The results of a thermal stress analysis in Nastran In-CAD are inaccurate when reading temperatures ... for details.
17	The input table for temperature dependent modulus of elasticity (E) is now identified as load scale factor versus temperature. The input is a multiplier for the entered modulus. See Displacement wrong when using temperature dependent modulus in Nastran In-CAD for details.
18	When loading a material from a user-defined library, the stress-strain data for a nonlinear material is shown properly. See Loading non-linear material database fails to correctly display stress-strain curve in Nastran In-CA... for details.
19	The XY Plot of heat rate through elements now includes all faces. See Heat rate through element is from one face only in Nastran In-CAD for details.
20	The input table for temperature dependent thermal conductivity (k) is now identified as load scale factor versus temperature. The input table is a multiplier for the entered conductivity. See Thermal results are wrong when using temperature dependent conductivity in Nastran In-CAD for details.
21	Correct velocity results are shown at midside nodes on bonded faces. Previously, the results were shown as 0. See Velocity results are zero at points on a contact face in Nastran In-CAD for details.
22	Analysis fails with Fatal Error E5073 if normal modes analysis is run after the analysis is changed from shock/spectral response analysis to normal modes analysis.

23	Corrected problem with reading pressure loads from Autodesk CFD analysis.
24	Improved temperature calculation in a heat transfer analysis when using parabolic elements to minimize temperatures above or below the ambient.
25	New iLogic commands to create or deleted analyses and subcases.
26	The files in the InCAD\FEA folder associated with an analysis are deleted when the analysis is deleted.
27	The MFLUID command now works with the Lanczos extraction method. See How to include fluid in a modal analysis in Nastran for details.
28	The CONSTRAINTS command on the ribbon provides direct access to Pin and Frictionless constraint types. (They can still be accessed from the dialog as before.)
29	The acceptable input on the Parameters dialog have been updated for some variables, such as CONTACTSTAB.
30	Multi-axial fatigue analysis with multiple loads gave incorrect results in version 2019. See the forum post Multi axial Fatigue Analysis not displaying the dynamic area results . (added August 1, 2019)