

FEMAP v2020.1 MP 2 New Features and Corrections

Updates and Enhancements

Interfaces - Simcenter Nastran (formally NX Nastran)

SOL 401 and 402 Only

- Added support for reading plastic strain results at grid points for SOL401/SOL402. This includes support for solid and shell elements.

SOL 402 Only

- Added support to write request to recover Elastic Strain output at grid points for SOL402.

Output and Post-Processing

- Added support for post-processing plastic strain results at grid points for SOL401/SOL402. This includes support for solid and shell elements.

Corrections

Analysis Manager

- Corrected issue with the “Boundary Conditions” dialog box in the Master Case of Buckling Analysis for Nastran solvers which could cause FEMAP to exit unexpectedly. (PR# 9682285)

Geometry

- Corrected issue which caused some surfaces to not be offset correctly. In addition, some additional warning messages were implemented to potentially alert the user to review the offset surface(s) before meshing.

Graphics

- Corrected issue which caused a very large amount of memory to be used to display mesh size symbols on curves, which now allows a neutral file to properly imported. Please note, an issue still exists when the default point size is too small, causing a massive amounts of curve mesh evaluation and causing a degradation in performance.

Interfaces - Nastran

- Corrected issue introduced in FEMAP v2020.1 that caused the “BOLTFRC” entry to be written for versions of Nastran (i.e., Autodesk Nastran) where it is unsupported. This issue also prevented the correct entry, “BOLTFOR”, from being written for those versions of Nastran.

Interfaces - Simcenter Nastran (formally NX Nastran)

SOL 401 and 402 Only

- Corrected issue that affected the writing of the “BCSET” case control command for SOL401/ SOL402. Contact cannot be changed in a modal subcase, or set if that subcase is the first in the analysis OR not sequentially dependent, thus the “BCSET” case control command is prevented from being written in these special cases.
- Corrected issue which caused erroneous “TEMP(Load)” case control to be exported for SOL401/ SOL402 when the “From Load Set” option was selected in the Temperature drop-down, even if the load set selected did not contain any temperature loads.

SOL 402 Only

- Corrected issue which caused an invalid value to be written for the MATSYM parameter on the NLCNTL2 entry for SOL 402. The Simcenter Nastran Quick Reference Guide mentions using “YES” or “NO” for this parameter, but Nastran actually accepts only “1” or “0”.

Interfaces - ABAQUS

- Corrected issue when exporting directional pressure loads as DLOAD, type TRVEC on plate elements which would cause “TRVEC” to be written twice to the TRVEC data line (i.e., TRVECTRVEC) when analysis type is Frequency Response.
- Corrected issue when exporting directional pressure loads as DLOAD, type TRVEC on axisymmetric shells and plane strain elements where 2 TRVEC data lines would be printed for a single load for all analysis types other than Frequency Response.

Properties

- Corrected issue with Beam and Bar properties when Shape was set to NASTRAN T2 in the Cross Section Definition which could cause incorrect values to be calculated for Torsional Constant, Y Shear Area, and Z Shear Area.

Meshing

- Corrected issue which occurred when attempting to extrude elements or project nodes along a vector or using an element's normal direction. Previously, if the vector supplied by the user or the normal vector missed the edge of the surface due to numerical tolerances, the commands or API methods would fail or generate inconsistent results. Now, assuming the vector passes within numerical tolerances of the edge of the surface, the process should work consistently. In addition, corrected similar issues when attempting to project points.

API

- Corrected issue which caused deleted output vectors to be listed, one-at-a-time, to the specified List Destination(s) when using feDeleteOutputV2 or feDeleteOutput2V2.