



NISA Suite of FEA Software

Release Notes

Version 20.0

NISA Software

September 2020

***Worldwide Copyright (C) NISA Software.
All rights reserved.***

World Headquarters:

NISA Software
2635 N 1st St, Ste 131
San Jose CA 95134, USA

- Web URL : <http://www.nisasoftware.com>
- FTP : <ftp://nisasoftware.com>
- E-mail : Sales@nisasoftware.com

NEW FEATURES

NISA2

- Coupled displacement face feature is added to connect nodal displacements on two connected surfaces where nodes are not one to one exactly match on these two surfaces.
- Shell nodal thickness can be defined in element as an alternative to use RCTABEL data group to defined nodal thickness.
- Node offset is added to composite shell element to allow element nodes offset to the top, bottom or any distance from the shell mid-surface.
- Average nodal strains are calculated as long as nodal stresses are requested. Average strains can be printed in output file and saved for post-processing.
- Strain ratio failure criterion is introduced as an additional failure criterion for composite element. Either stress ratio or strain ratio can be select as failure for output and post-processing .
- Element nodal temperature input is introduced as a alternative to define temperature at nodes using *NDTEMPER and *NDTEMDIF data group. Temperature can be specified for each node of an element. A Element can have different temperature from other elements connected to the same node.
- Recalculate element stiffness as requested for each load case in multiple load cases with the same boundary conditions. This feature allows different element stiffness between load cases due to different temperature in temperature dependent material.

DYNM2

- Add *PRTPRESTORED data group to export values of selected pre-stored spectrums in output file if it is requested.
- Expand pre-stored spectrum to include India pre-stored spectrum year 2016.

HEAT2

- Shell element thickness can be defined in element instead of using RCTABEL. This eliminates unnecessary large number of RCTABLEs from metal forming model in which each element has different thickness.

DISPLAY III

Pre-processing

- New Contact element : Coupled Displacement Face
This facilitates to create contact faces on elements whose mesh do not match
- Facility to request History data for Average nodal strain, Top surface nodal strain, Bottom surface nodal strain - for Linear and Non-Linear Transient analyses.
- Facility to modify thickness of each node to a different value in an element.
- Facility to print pre-stored data used in Shock analysis in output file for the user's convenience.
- New boundary condition - Element temperature - This data group may be used to specify temperature and temperature difference at nodes for an element - Applicable to Static, Buckling and Nonlinear static analyses.
- Composite offset – Offset option for composites NKTP = 32,33 is provided in property table (*RCTABLE)
- Cyclic symmetric bc – For a cyclic symmetric model DISPLAY identifies the nodes on radial edges with same displacement and rotation values and creates coupled displacement data group for the identified node pairs

Post-processing

- History data plot for Average nodal strain, Top surface nodal strain, Bottom surface nodal strain - for Linear and Non-Linear Transient analyses
- Contour plot of Unaveraged results(elemental results) for composite element NKTP 32 for NLStatic analysis – Nodal stresses, Nodal stress resultants, Nodal strain and Failure criteria results.

- Contour plot of average nodal strain for linear static and non-linear static analysis.
- Contour plot of average nodal strain for composites
- Composites – Survey plot feature enhanced to plot maximum and minimum values

DISPLAY IV

Pre-processing

- Facility to request History data for Average nodal strain, Top surface nodal strain, Bottom surface nodal strain - for Linear and Non-Linear Transient analyses.
- Facility to modify thickness of each node to a different value in an element.
- Facility to print pre-stored data used in Shock analysis in output file for the user's convenience
- Composite offset – Offset option for composites NKTP = 32,33 is provided in property table (*RCTABLE)
- Cyclic symmetric bc – For a cyclic symmetric model DISPLAY identifies the nodes on radial edges with same displacement and rotation values and creates coupled displacement data group for the identified node pairs

Post-processing

- History data plot for Average nodal strain, Top surface nodal strain, Bottom surface nodal strain - for Linear and Non-Linear Transient analyses.
- Contour plot of Unaveraged results(elemental results) for composite element NKTP 32 for NLStatic analysis – Nodal stresses, Nodal stress resultants, Noda strain and Failure criteria results
- Post-processing of Floor Response Spectra
 - Plotting floor response spectrum, acceleration versus frequency, for a given set of damping ratios
 - Peak Broadening and Lowering the Secondary Response Spectra

Enhancements

NISAII

- Remove preset limit on slave node on rigid links, slave nodes on coupled displacements, and master node on Rig2link data. The limitation of allowable slave/master nodes in one of these data group is the program memory (MEMO) available for computation.
- NISA2 input data is expanded to 128 characters per line instead of 80 characters per line
- When centroidal stresses or resultants are requested in *POSTCNTL and KSTR is not set, the program sets it to KSTR=7

Dynm2

- Expand length of input file data line from 80 characters to 128 characters for Dynam2 input data.
- Dynm2 works with Presys license scheme.

Heat2

- Speed up reading data group *ELEMENT and SPTEMP. For large model, reading *ELEMENT and *SPTEMP took extremely long time.
- Input file data line has been expanded to 128 characters long instead of 80 characters long.

DISPLAY III

- Stress Linearisation feature is improved.
- Reaction force values can be plotted on nodes

DISPLAY IV

- Reaction force values can be plotted on nodes

Error Corrections

NISAI

- When the RIG2LINK is very long, the coefficients of MPC become very small number. They may be smaller than the internal tolerance of the small number check. Those components of the MPCs are cut out from the MPC.
- Missing mass calculation (or influence matrix for ground excitation) is activated in *EIGOUT card but there was no calculation done if *ECHO=OFF (no data echo printing in the output file). KPSED=3 did not activate missing mass calculation.
- Run time error for KSED, TOLDUm, CFREQ due to undefined variable.
- Hardware network security may crash if there was no version 18 installed on the system
- Sandwich element should have zero value in Sxz and Syz on face layers, which was not the case in the analysis result.
- Number of Gauss points was not defined properly for *PRESTN calculation.
- Eigenvalue 'check run' ran into error of 0 record encountered in file 26.
- Radial return method in material plasticity cannot converge in 2d plane stress element using very fine user strain-stress curve.
- The size of requested memory for sparse solver printing *s instead of value.
- Averaging composite stress resultants are incorrect due wrong subroutines are used to read back data.
- In large format *RIG2LINK data group, it might have error in reading data.
- Reaction forces might be incorrect in nonlinear transient dynamic.
- Using large format, program issued error due to 'duplicate event ID'.
- Buckling analysis job stopped due to stack overflow without error message.
- The request memory size in sparse matrix solver was printed incorrectly.
- Material principal direction might not correct for composite element with nonzero stack angle.

- Pressure load or mass matrix might be not correct when shell element has node offset.
- Interlaminare shear might not be correct due to material angle conversion.
- ETA network license update. Client machine may not find eta license.
- Incorrect failure criteria value for NKTP=33 using temperature dependent failure coefficients such as FXT,FXC etc.
- Unreasonable result might be given from the analysis with CFORCE loading due to program compiler bug.
- In nonlinear analysis, the program might issued address for composite stress resultant was not defined. It did not create any problem because this component was not saved in nonlinear analysis.
- Incorrect displacement as well as stress in NKTP=39 with non-standard sections for nonlinear analysis.
- Incorrect element generation for 1st level element generation with more than one element in the base set.

Dynm2

- When FRSOUT was specified in the input file, HISOUT data was available in post process.
- Incorrect Node ID might be printed in FRSOUT printout if more one damping ratio was specified.
- 64bit version dynm2 might crash when center stress is requested for shock spectrum analysis.
- Log window disappeared when the job was done. It is better to wait for the user to type in a key to close the log window.
- Program encountered 0 record in reading file 27 if VMS was listed in the RSET used by SNAPSHOT
- Time history input data issued unnecessary warning message for TIME1 and TIME2 when they are equal to TSTART and TEND time.
- Correct the message of 'incorrect record number to retrieve data for FRSOUT printout to file 18'.

Heat2

- Incorrect label DISPLACEMENT instead of "TEMPERATURE" was printed in output file.
- Model get negative pivot in stiffness or diverge solution with CONVBC.
- Program crashed when using large format and local coordinate system.

Nis2ext

- When material properties EX equals EY, NIS2NAS assumed that was an isotropic material not orthotropic material. MAT1 card instead of MAT8 card was written out.
- Hill's failure criterion is always written to FT field of PCOMP card for composite element because NISA always calculates it. In case it is not needed or incompatible with other input data, users should put this field in blank.
- Since failure criterion is used in FT field of PCOMP card, value in Xt, Yt, and S fields are expected in MAT8 card for orthotropic material. NASTRAN does not accept zero value for those fields in this case. A warning message will be issued in NASTRAN input file. Users should provide these values or turn off failure theory (FT) in PCOMP.
- For core material in sandwich plate, users usually provide value in S value for core material but not Xt and Yt. In this case, the NIS2NAS will put a small value to Xt and Yt to allow NASTRAN job to go forward.

DISPLAY III / IV

- Realistic plot of beam elements is extremely slow - Fixed
- Layer number is not displayed in contour plot in case of composite elements - Fixed
- Writing multiple external results – Results are not written for Top and Bottom layers – Fixed
- In beam contour plot, error message is displayed when loadcase id is changed in POST STATUS form-Fixed
- Error message is displayed if RAM size not given for iterative solver – Fixed
- History plot not working for Linear transient and non-linear transient dynamic analysis – Fixed

- *SPDISP and *PRESSURE datacards are not written back in nisa file for those nisa files which are created by NISA-Presys –Corrected
- Post-processing of beams - Axial stress contour plot – One of the beam element shows incorrect value- Fixed

DISPLAY III

- History plot not working for Non-Linear analysis if the nodes ids are large (more than 6 digits) _ Corrected.

DISPLAY IV

- History plot not working for Linear transient and non-linear transient dynamic analysis – Fixed
- End release code in not displayed in message window – Fixed

NISA/CIVIL

New Features in Translation Interface

- Import Structural Data from ETABS : includes geometry, properties, Loads & BC Data (nearly seamless translation)

Enhancements in Translation Interface

- Import Structural Data from STAAD PRO: includes Dynamic Load Cases data import (nearly seamless translation)
- AutoCAD DXF interface : Introduced 3D frames importing, Surface entities translated as Plate/Shell elements

New Features in Pre-processing

- RC Shear Walls : Auto-Identification from shell model
- Generate Beam elements for Curved profile: in any of the standard planes, with user control for number of divisions
- Temperature Load : introduced option for specifying Element Loads (corresponding update in NISA FE Solver)
- Wind Loads as per IS:875-2015

- Seismic Loads as per IS:1893-2016
 - Option to generate only Accidental Torsion forces for the Seismic Load case
 - Notional Load (with Load %)
- Cardinal point setting : option for auto-generation of offset to Beam type elements

Enhancements in Pre-processing

- Auto-Generation of Foundation Springs : Option to scale stiffness Factor
- Selective Renumbering of Elements : Feature extended for shell elements
- Beam Loads - Line Mode option : extended the feature to include multiple sets of continuous beams association with single selection (for similar load data)
- Panel load transfer : Retain association after editing
- Load case dependant Material/Property change : Feature extended for Dynamic Load cases
- Dynamic Loads input storage :
 - Ground motion data directional factors made easily editable (for both Pre-Stored & User-Defined Spectra data)
 - Material Damping spectral data association

New Features in Post-processing

- Post-Processing Plots : enabled for Dynamic Load Combinations (envelop)
- Post-Processing Plots : enabled for Moving-Vehicle Load Combinations (envelop)
- Tapered Elements : enabled Graphical Post-processing & Analysis Report processing
- On-screen display of RC Beams and Columns Designs : Selective display of Flexure & Shear reinforcement
- Mass Center, Stiffness Center Reporting : introduced in NISA File under Seismic Load Case with Accidental Torsion
- Dynamic analysis results : enabled option to print results into a file for reference (CSV format)
- Final Re-bar arrangement – RC Beams and Columns : option introduced to print data into a file for reference (CSV format)
- SRSS Combination of Forces : Static and dynamic
- Strip BMD and SFD Plots : for Shell models in any plane (.CSV file generated)
- Selective Range display of results : RC Columns Design
- Display of Total Reaction, Moments & CG for actively displayed supports
- Transfer of support reactions to a Raft Model : auto-generation, to analyse separately
- Beam & Columns Design Forces : Envelop of forces for groups (in *.FRC output)

Enhancements in Post-processing

- On-screen display of RC Beams and Column Designs : Enhanced with reinforcement percentage (in parenthesis)
- Analysis report (improvements & New Features):
 - option included to account Dynamic Load results & Load Combinations
 - option included to account Moving/Vehicle Load results & Load Combinations
 - Influence Lines (for Unit Moving Load)
 - Combined Foundation
 - Base Pressure detailed & Summary min.-max. values; min.-max. values for all load cases/combinations selected
- Force Diagram (BMD-Frame): Improved & New options
 - Read Results from File(to transfer results from 3rd party FE Analysis Software)
 - Generate Report File
 - Contours Plot by Range
 - Reference Node for displacement
 - Display of Stress Contours for Shells (including option for Un-Averaged stresses, Layers, Resultant Displacements, Integrated Forces)
 - Display of Base Pressure Contours for Shells (Also written to CSV file with Min. & Max. Values summary); one load case/combination at a time

New Features & Enhancements in Designing

- Design of RC Footings for spring supports :Column above the spring is auto-identified
- RC Shell Design:
 - Corrections for Shear Capacity computations
 - Corrections to Wood-Armor moments accounting
 - Corrections to moment signs
 - Corrections to dynamic Load combinations forces accounting & sub-combinations
- Pile Foundation Design: Revised & improved
- RC Beam Design : Design for maximum span moments enabled (Design from Support to Support; auto identification of supports)
- RC Beam Design : Option to ignore Shear Capacity of Concrete (as per IS:13920-2016)
- RC Column Design : Option for Strong Column Weak Beam considerations, as per IS:13920-2016)

- Auto-Search feature : Improved performance of Design of beams and columns
- Raft Design: Improved
- Structural Steel Design : revision of IS:800-2007 design implementation
- Shear Walls : Revised Design for junction forces extraction/accounting
- Shear Walls : implemented Accidental Torsion checks
- Footing Design : Introduced options for checks against sliding, overturning

- Display of individual results for combined footings Report(Restored)
- AERB Codes for design (Thoroughly revised & Improved)
 - option for Abnormal Load condition (design factors)
 - RCC: Analysis mode for user provided steel (as per NPCIL & BARC requirements)
 - For RC Beams, Columns & Shells
 - Detailed interaction checks for capacity computations
 - “Liquid Retaining Structures” Design method for Shells (in addition to “Limit State”)
 - RCC Beam & Column design forces & capacities written into .CSV format file
- General & specific bug-fixes & stabilization of features

Utility Features

- Analysis Post-Data extraction (Speeded up): Considerable optimised memory management for large number of modes extractions for dynamic load cases & combinations
- Analysis Report: Speeded up
- Design Execution (Speeded up): RC Beams, Columns, Shells
- A range load Cases/combinations can be associated using “Shift” key
- Graphic View quick selection Right Mouse click menu items are appended with ‘Erase Loads’, ‘Erase/Plot Realistic Plot’, ‘erase Design Information’
- Shortcut keys for convenient view option for selected entities:
 - “Ctrl” key + ‘Erase|Section’ in Top View: shows section in Front view
 - “Shift” key + ‘Erase|Section’ in Top View: shows section in Side view
- Post Processing for selected set of entities: Graphical results for Members, Shells or Nodes with min. & max. Range values
- Members of Levels Display: option to plot with connected Columns beneath
- Property Data: read data from files
- Group data copy facility for RC Shells
- Auto-Identification of One-way slab for design forms (based on load distribution pattern)
- End Release Display: Option to switch On/Off End Release Display
- Load Combination Titles & Property Titles: increased number of characters allowed from 60 to 110

- RC Shell Design Summary Report: Failed Element Status: reporting modified as “Pass/Fail” for clarity (previously reported as “1/0”)
- RC Footing display (onscreen) for Y Gravity: accounted & corrected display issues
- Units: option to directly set dual units for M-mm, KN-N
- Structural Steel Design Properties: read data from files

Error Corrections

- Beam Elements property storage (corrected):for local member offsets
- Bridge moving load auto-generation : locations of geometry transitions & curved vehicles
- Retaining Walls Design (Corrected): Key moment computation