



# **NISA Suite of FEA Software**

## **Release Notes**

**Version 17.0**

**Cranes Software, Inc.**

**July 2009**

***Worldwide Copyright (C) Cranes Software, Inc.  
All rights reserved.***

**World Headquarters:**

Cranes Software, Inc.  
1133 E Maple Road, Suite 103  
Troy, MI 48083  
U.S.A.  
TEL: (248) 689-0077  
FAX: (248) 689-7479

- Web URL : <http://www.nisasoftware.com>
- ftp : <ftp://nisasoftware.com>
- E-mail : [info@nisasoftware.com](mailto:info@nisasoftware.com)

**TABLE OF CONTENTS**

Page No.

**NEW FEATURES..... 5**

**PRE & POST PROCESSORS ..... 5**

    Pre-Processor in DISPLAY III & DISPLAY IV .....5

    Features specific to DISPLAY IV .....6

*Meshing* .....6

*Performance and Improvements*.....7

*Report Generation*.....7

*User Interface* .....8

*Pre-processing in DISPLAY IV*.....9

*Post-processing in DISPLAY IV*.....12

**CAD TRANSLATORS.....15**

    Sat2Neu, CATIA V4/V5, Pro/E, Inventor, UG, STEP, Adv. IGES to DISPLAY Neutral.....15

**FEA TRANSLATORS .....16**

    FE data translation from NISA to ANSYS.....16

**PROCESSORS (SOLVERS) .....17**

    NISA II.....17

    ADVANCED DYNAMICS .....20

    ROTOR DYNAMICS .....20

    ENDURE .....20

    EMAG .....21

**NISA/CIVIL.....22**

*Modeling (Pre-processing)*.....22

*Analysis*.....23

*Design* .....23

*Post-processing*.....24

*Drawing*.....25

*User Interface* .....27

**ERROR CORRECTIONS .....29**

**PRE & POST PROCESSORS .....29**

    DISPLAY III .....29

DISPLAY IV.....30

**CAD TRANSLATORS.....33**

**FEA TRANSLATORS .....33**

    FE data translation from NISA to ANSYS.....33

**PROCESSORS (SOLVERS) .....34**

    NISA II.....34

    ADVANCED DYNAMICS.....34

    ENDURE .....34

**NISA/CIVIL.....35**

**LICENSING & PACKAGING ENHANCEMENTS.....38**

**O/S SUPPORT.....38**

## NEW FEATURES

In this section new features and enhancements to the NISA suite are presented.

### PRE & POST PROCESSORS

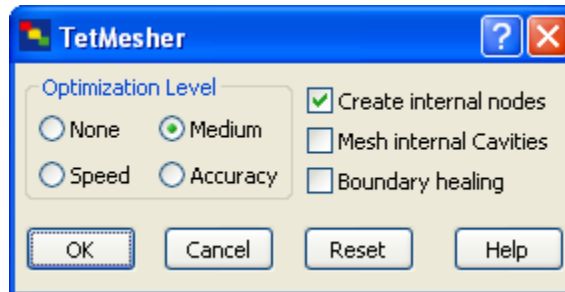
New features in DISPLAY III and DISPLAY IV modules are explained in the following sections. These developments are enhancements to the pre and post processing capabilities of solver modules such as NISA II, NISA/DYNAMICS, FLUID, HEAT III and EMAG. This also covers the DISPLAY IV Graphical User Interface enhancements.

#### Pre-Processor in DISPLAY III & DISPLAY IV

1. Advanced Tetrahedral Mesher (ATM):

This feature has the ability to create tetrahedral mesh from an existing linear Tri/Quad surface mesh. Surface mesh can be generated using Advanced Surface Mesher (ASM) facility in DISPLAY, or imported from any other third party software.

FEM→Mesh→Adv Tet Mesher



2. Large input Format:

Restriction of one million nodes and elements is removed. This feature enables the user to have nodes and elements ID number beyond 1 million (I6) and up to 100 million (I8). This feature is available for NISA II solver.

3. Neutral File Format:

Neutral file format is extended to support all Direct Frequency Response Analysis data groups and MATDIR1, MATDIR2 data groups.

4. NISA Parametric Language (NPL):

MACRO is extended to support array parameters declarations and storage (1D array).

```
DIMENSION XYZ[3]
```

```
GRD, ADD, 3, XYZ[1] / XYZ[2] / XYZ[3] / 0
```

MACRO is renamed to NISA Parametric Language (NPL) and the documentation is revamped for easy syntax reference.

## 5. Session Command:

Session command length is extended up to 512 characters and is no longer limited to 74 characters. This feature enables the user to group string of commands to perform various tasks in sequence.

Session command, "DIAG MESH" is added. This feature is useful to print the assigned values of the variables, user defined strings, and evaluator output. RCR needs to be set as ON to see this data.

```
DIAG, MESH, ***** (NDCOUNT) *****
```

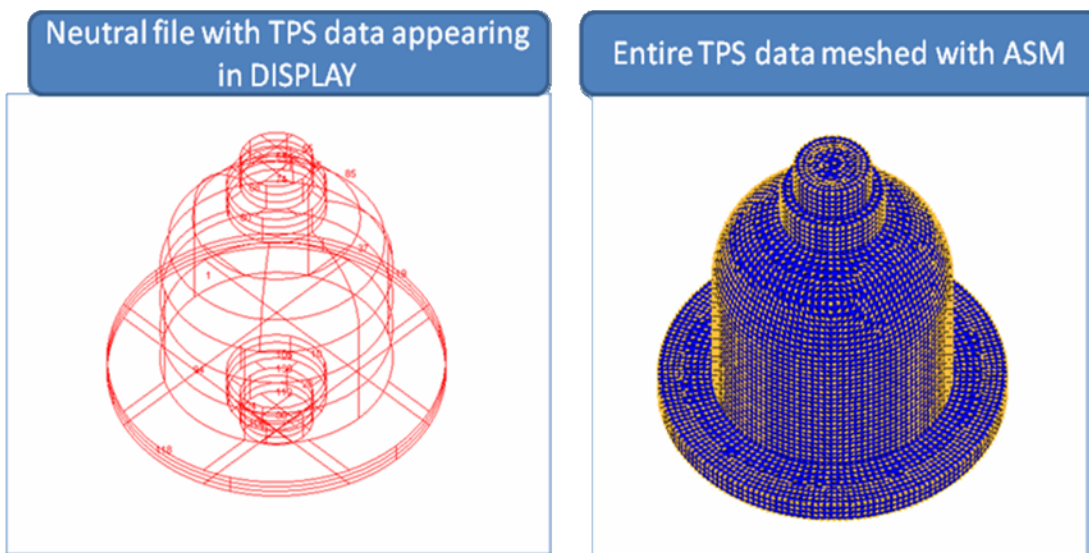
## Features specific to DISPLAY IV

### Meshing

#### 1. Advanced Surface Mesher (ASM):

Advanced Surface Mesher (ASM) is extended to work with Trimmed Parametric Surface (TPS) data. Using this feature, the user can mesh individual or entire TPS data with the required meshing options. With individual TPS Meshing, the connectivity between 2 adjacent TPS is not automatic. The user needs to ensure the connectivity. If all the TPS are meshed together, the connectivity is maintained.

FEM→Mesh→Adv Surface Mesher



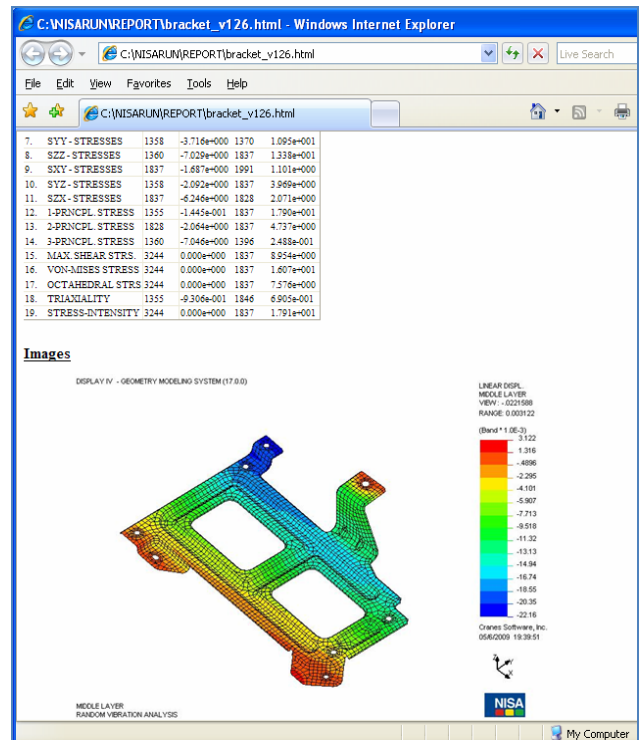
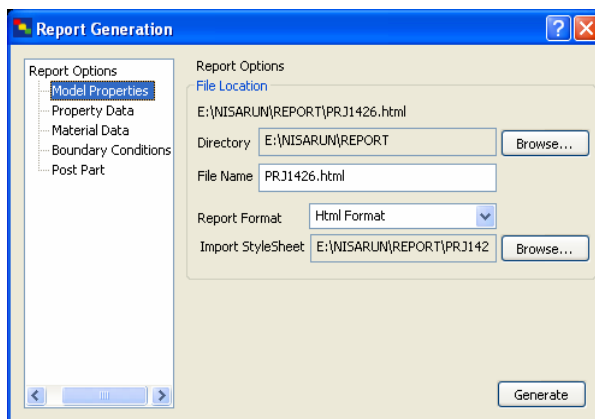
### Performance and Improvements

2. Lighting and shading parameters are fine-tuned to get realistic view of the model.
3. Enhanced the performance of key operations such as node merge and all delete operations in the range of 60 % to 90 %. Speed of hidden line plotting and shaded view for elements is improved in the order of 60 %.
4. Post processing of contours: Continuous color contour is made as default where as the conventional 14 color band plotting is made optional.
5. While plotting contours, graphs and history plots the view area is optimized.
6. Graphical window and the Entity/Workspace window are improved to work independently.
7. Gradual rendering of the entities is implemented.

### Report Generation

8. Report generation for both pre and post processing is added to automatically generate HTML and Word document reports.

Tools→Generate Report



### User Interface

9. The tree structured menu in the workspace area is enhanced for quick and easy access of the menu items.
10. Session commands are displayed in the dialog boxes.
11. Nodes and Elements data is presented in a text format. The user can further utilize this data.

View→Entity Information

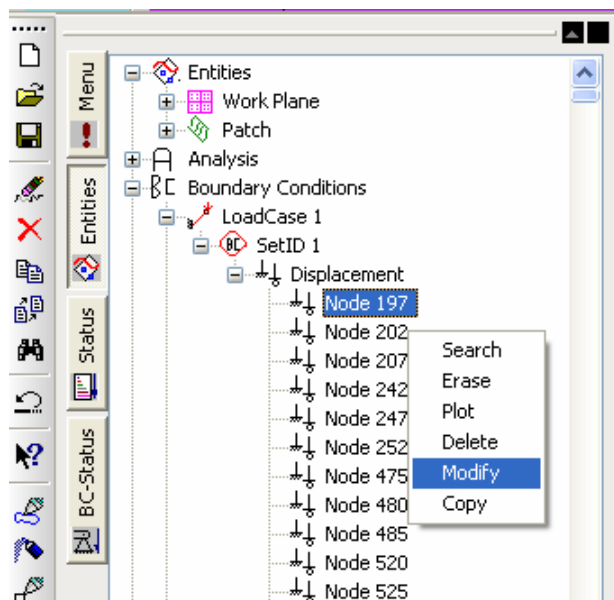
12. Post Toolbar is appended with forward and backward arrows to change the load case or time step number. This helps user to post process the results without using post status dialogue. This feature is available in Linear and nonlinear static analysis.
13. Command Syntax help is provided as user types in the session command window. After 2 levels, the entire command syntax is shown as a tool tip.



14. A “Measure” toolbar is provided to measure linear length between Grids and Nodes. Also, the linear length is measured for the selected line.



15. Entity tree options are enhanced to list Boundary Conditions and Analysis data. Operations such as modify, search, add IDs can be executed from the entity tree (Workspace window).

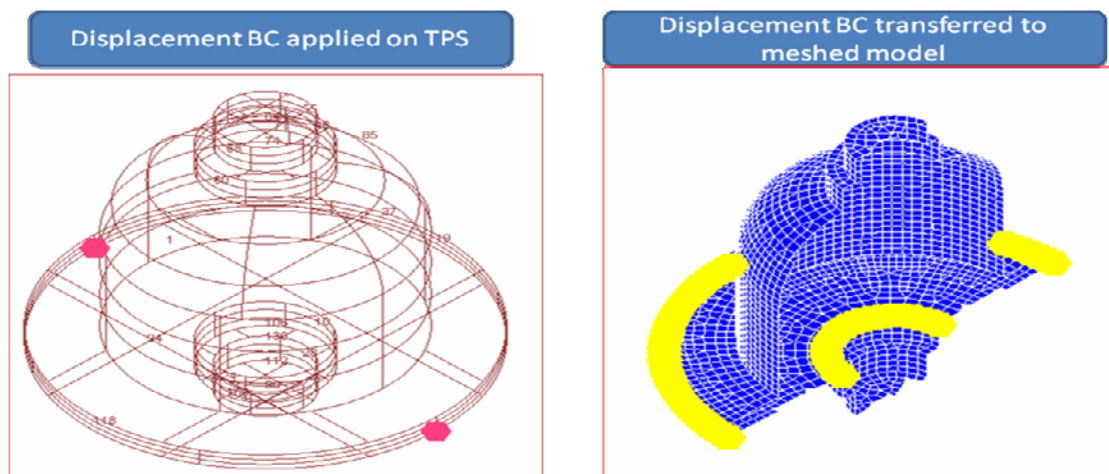


16. Context based menus and toolbars are provided. As per the context selection or the files read, the corresponding menus and toolbars will become visible. The following are the functional keys:

KEY USED	FUNCTIONALITY
F2	Geometry Toolbar
F3	FEM Toolbar
F4	Analysis Toolbar
F5	Post Toolbar
Ctrl+Left mouse Button	PAN
Shift+Left mouse Button	ROTATION
Middle button click once + Left mouse button	ZOOM IN/OUT
Left button Double click once + Left mouse button	Zoom Rectangular Box area
I	Isometric view
Z	Zoom To Fit
O	Opens Orientation Window
Q	Opens list of Recently used Dialogs
C or SPACE BAR	Sets the cursor position to Command Window.
ESC	Exits out from Full screen.
F1	Displays HTML Help

#### *Pre-processing in DISPLAY IV*

17. Applying Boundary Conditions on TPS is supported for Structural BCs such as Displacement, Pressure and Heat BCs for instance Initial temp, Specified temp and Convection. After meshing, all the BCs applied on the TPS will be automatically transferred to FE model.



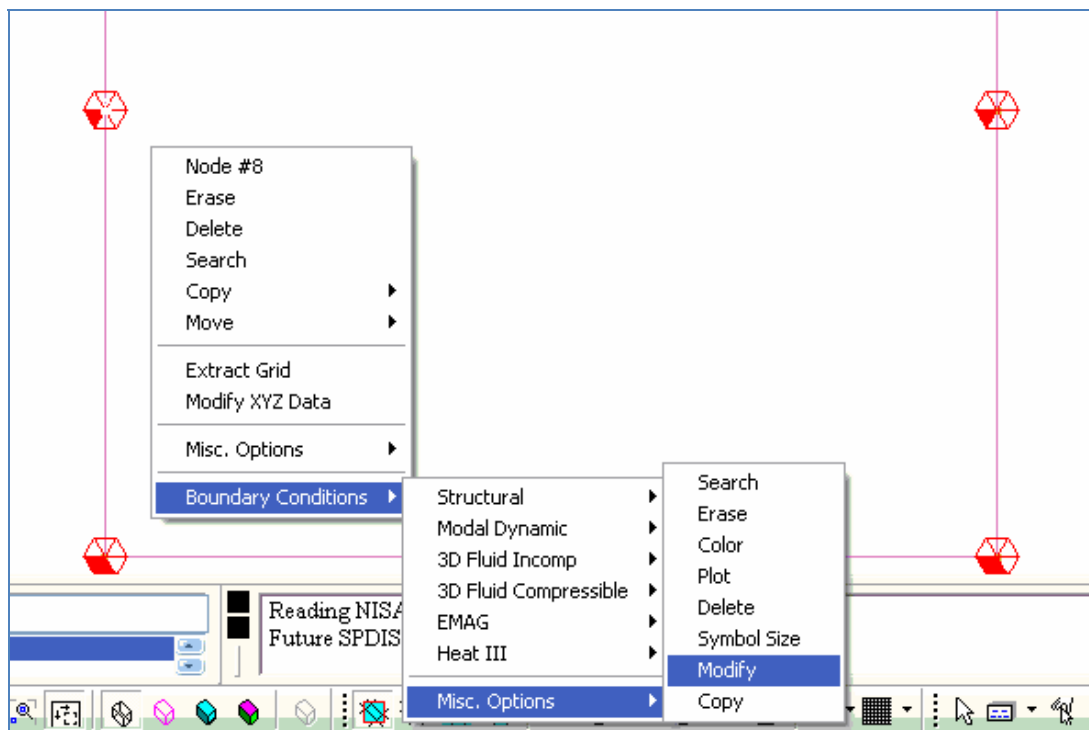
## 18. Piezoelectric Analysis:

All the data cards required for the Piezoelectric - Static/Eigen/Ltransient Analysis, are supported. Material data card \*MATPIEZO for piezo material is added. 3-D Solid Piezoelectric Element (NKTP = 6) and 3-D Membrane Piezoelectric Element (NKTP = 8) are supported. Apart from the existing supported boundary conditions, \*ECDENSITY, \*ECHARGE are supported for Piezoelectric analysis.

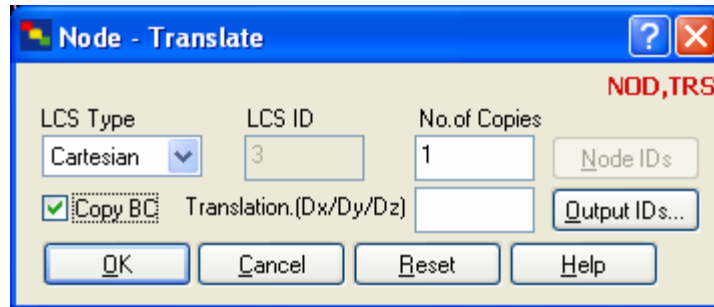
## 19. Direct Frequency Response Analysis

The data cards \*FREQCNTL, \*FREQFUNCTION are added. The boundary conditions like \*SPDISP, \*CFORCE are extended for Direct Frequency Response Analysis.

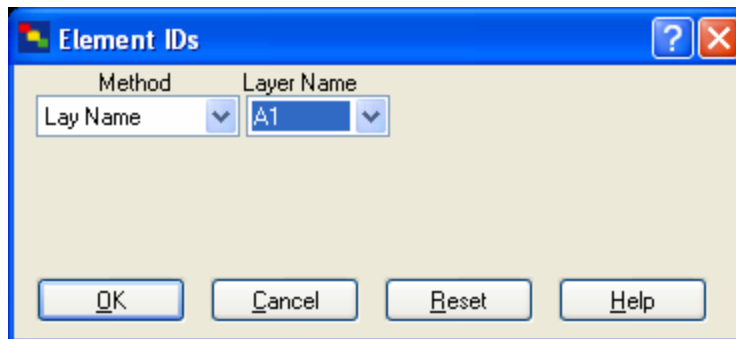
## 20. Facility to edit/modify the existing loads and boundary conditions is implemented. This is feature is available using the right mouse operation of individual BC. (Boundary Conditions → Misc. Options → Modify) as shown below:



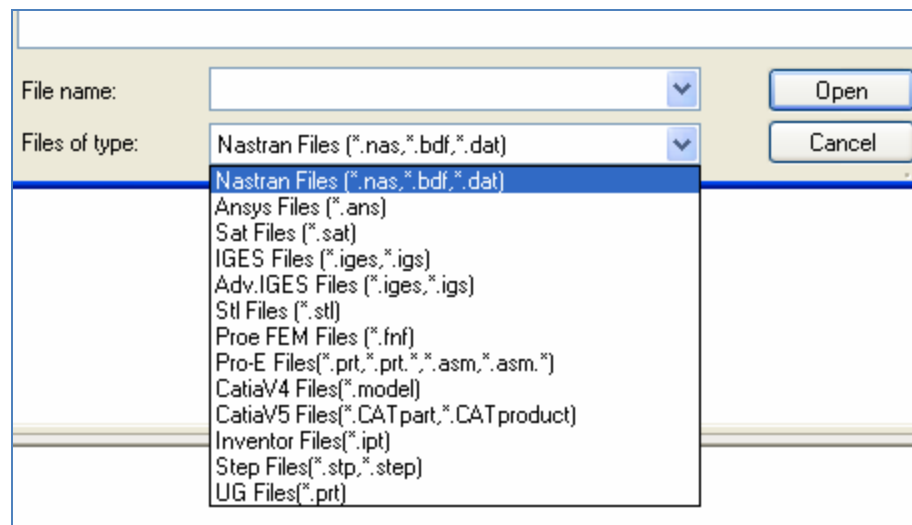
21. Facility is provided to copy the nodes and elements along with the associated boundary conditions.



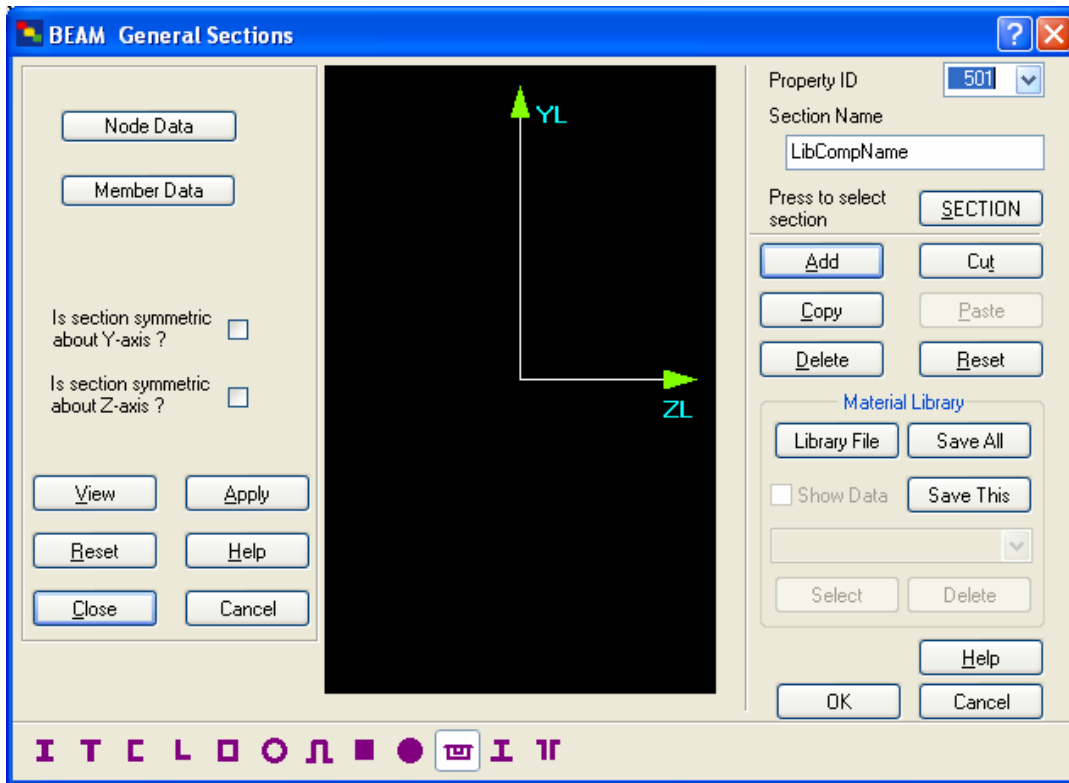
22. Using the option FEM→Data Groups→Contact Elements→Automatic, contact elements and contact surfaces are created automatically based on a user defined tolerance. Identification of the bodies is based on material or property IDs.
23. Pressure can be applied on a full plane by selecting 3 nodes.
24. Layer option is enhanced to append entities to an existing layer. Also layer names are listed in the dialog boxes for selection.



25. Import option is enhanced to support all CAD file formats allowing the various CAD file formats importation using File→Import.

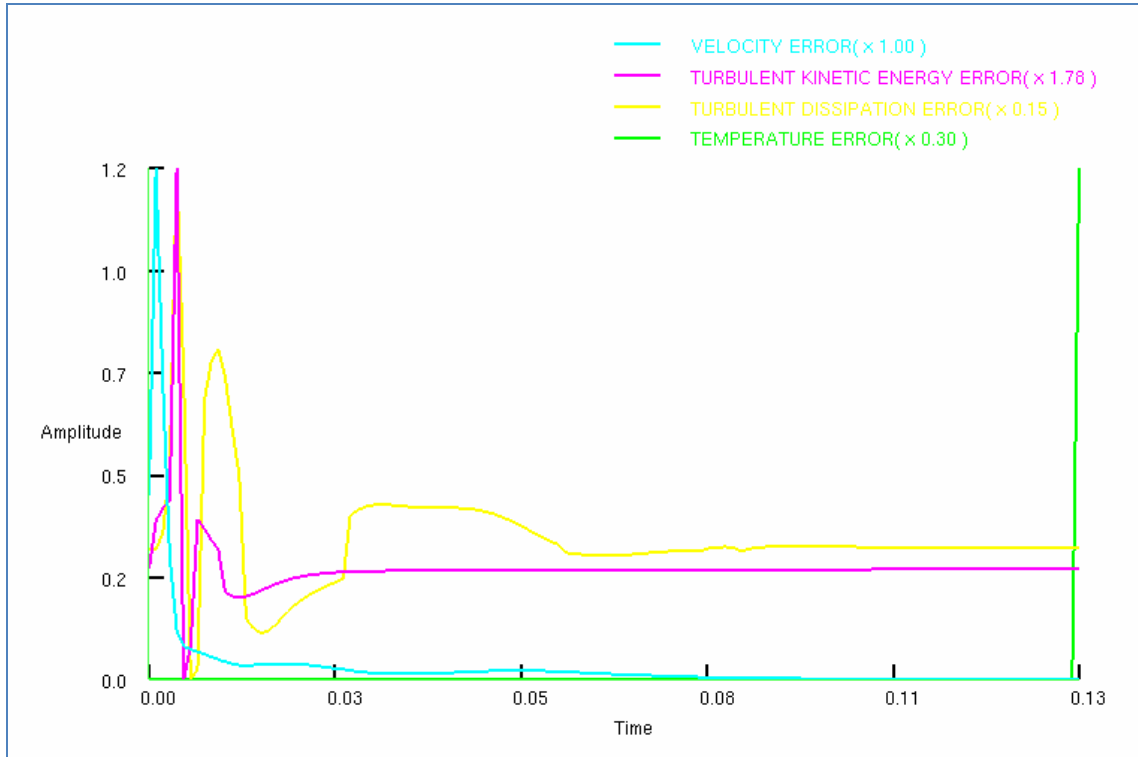


26. Facility to define general shape beam section is provided. Data input form for generally-shaped section is shown in the following figure. This is the main form for these sections and the user can select the appropriate pick buttons to enter data for nodes, members and cells.



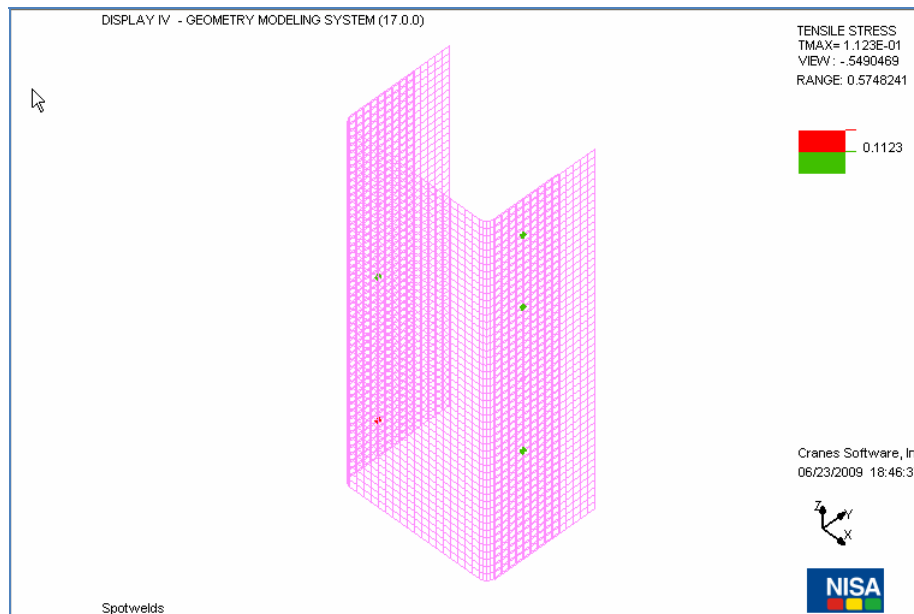
27. Using the option FEM→Elements→Modify→NKTP12 ->39, 3-D Beam Element (NKTP = 12) can be converted to 3-D General Beam Element (NKTP = 39). The cross sectional data of NKTP12 is taken from the #.dbs file.
28. Assignment of existing beam elements (NKTP=12) as Weld elements is added. Operations such as plot, erase, search and color are supported for these elements. Static failure parameters (Maximum values of tension, compression and shear) for welds can be input under \*MATERIAL data card.
29. Generation of the K-node at the center of the curvature of an elbow is done automatically.  
*Post-processing in DISPL AY IV*
30. Direct Frequency Response Analysis  
Contour plot of the responses like displacements, velocity, acceleration, stresses is supported. Deformation, history plots are supported for Direct Frequency Response Analysis. Post status form is added for Direct Frequency Response Analysis in order to change Frequency and Amplitude/Phase.
31. Writing multiple external results is supported.
32. History plot for nonlinear results is now available from external results.

33. Viewing results for complete model from the symmetric model is available with External results.
34. Convergence plot of fluid flow and heat transfer analysis is added.

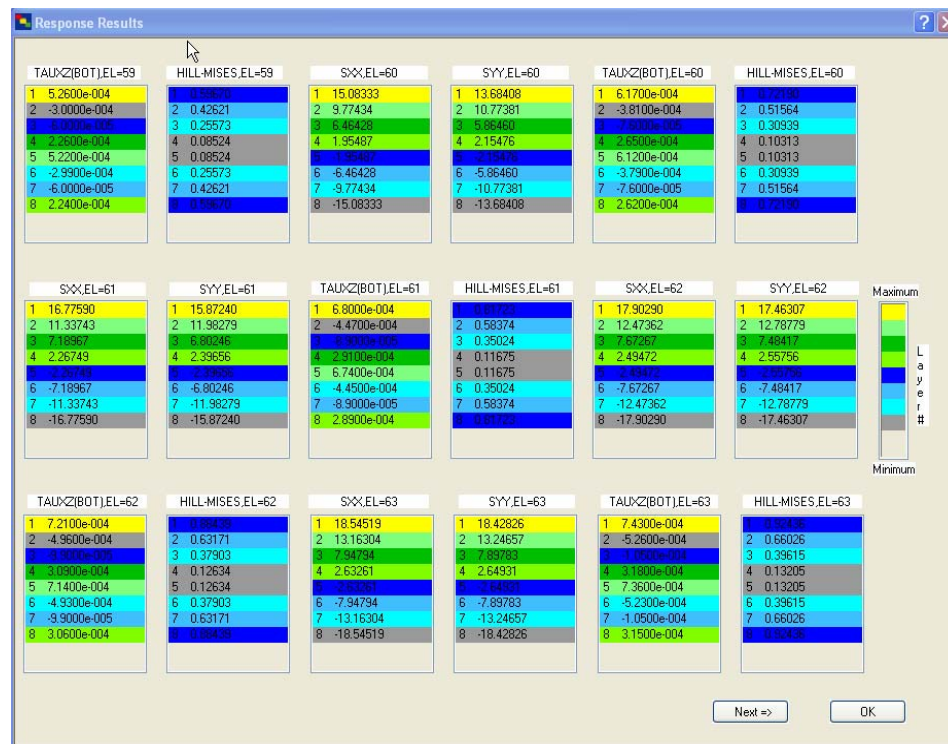


35. Post processing of Piezo Electric analysis is added.  
Contour plot of the responses like displacements, velocity, acceleration, stresses is supported for Piezoelectric - Static/Eigen/Ltransient Analysis. Deformation, history plots are supported.
36. Animation control is enhanced to create GIF output files option and the option to have only animation is introduced.
37. Post processing of reaction forces is incorporated. The values of reaction forces and moments are written along with the constrained nodes in the view area and an option provided to write the same in the information window.

- 38. Static failure contours for weld elements (\*WELEMENTS) are made available under Post Results | Misc. Results | Weld element results. Tension, compression and shear failure contours are supported.



- 39. Combining Linear/Nonlinear and Dynamic analysis results and writing to the output file for Shells and Solid Elements is added. To access Post→Misc Results→Seismic Analysis
- 40. Option to view composite post results for many elements at a time is added.



**CAD TRANSLATORS****Sat2Neu, CATIA V4/V5, Pro/E, Inventor, UG, STEP, Adv. IGES to DISPLAY Neutral**

All Translators are upgraded with ACIS R19 to support following CAD system versions:

<b>File Format</b>	<b>File Extensions</b>	<b>Version supported with NISA V16</b>	<b>Versions supported with NISA V17</b>
CATIA V4	#.model, #.exp	No Change	CATIA 4.1.9 – CATIA 4.2.4
CATIA V5	#.CATPart, #.CATProduct,	V5R2-V5R16	V5R2 – V5R19 V5R17→V5R19 (new)
IGES	#.igs, #.iges	No Change	Up to 5.3
Inventor	#.ipt	Inventor Versions 6, 7, 8, 9 and 10	6 - 2009
	#.iam	Assemblies are not supported	11 – 2009 Assembly support (new)
Pro/E	#.prt.*,#.asm.*	Pro/E 16 to Wildfire2	16 – Wildfire 4 Wildfire3-4 (new)
STEP	#.stp, #.step	No Change	AP203, AP214 (Geometry Only)
Unigraphics	#.prt	UG versions 11 to 18 and versions NX1, NX2, and NX3	11 to 18, NX to NX6 NX4-6 (new)

1. Translate as sheet bodies option is supported for IGES part files. To access from Application Launcher Settings→CAD Translator

## FEA TRANSLATORS

### FE data translation from NISA to ANSYS

1. Nodal temperature and temperature difference (\*NDTEMPER and \*NDTEMPDIF) are supported.
2. Plane stress triangular element NKTP=1, NORDR=10; axisymmetric element NKTP=3, NORDR=1, 2; torsional spring element NKTP=21, NORDR=1; laminated general sandwich shell element NKTP=33, NORDR=2, 11; thin shell element NKTP=40, NORDR=1 are supported.

## PROCESSORS (SOLVERS)

### NISA II

1. Large input format:

Restriction of one million nodes and elements is removed. This feature enables the user to have nodes and elements ID number beyond 1 million (I6) and up to 100 million (I8).

2. Shifted block-Lanczos eigen solver:

A multi-shifted block-Lanczos feature is introduced for faster eigenvalue extraction of large structures requiring higher number of modes. For example, to extract 1800 eigenvalues for a structure with 170,860 active DOF the computational time saving is 87 % as compared to Block Lanczos.

3. Direct Frequency Response Analysis:

User can analyze structures under support excitation with specified amplitude and phase spectra.

Spring (NKTP=17, 21) and damper (NKTP=48) elements can now be modeled with frequency dependent characteristics.

Computation of derived quantities is possible with a restart option.

The analysis is suitable for analyzing structures with damper elements to simulate for example, shock absorbers. Also, for large structures the direct frequency analysis is expected to yield more reliable result compared to modal frequency response analysis.

4. Contact Analysis:

A robust Lagrangian algorithm for surface to surface contact element is introduced to model both 2D and 3D contact elements. User input is minimized with respect to the penalty parameters. The concept of pinball region is introduced in order to reduce the search time over the contact surfaces. On defining the pinball region, the search process is confined to this small region around the slave node, thereby making the search process faster and accurate. This method, however, works only with the PARDISO solver.

5. An option is introduced in parallel solver (PARDISO) to make the system use either in-core or out-of-core memory. For very large sized problems when the system RAM is insufficient the solver can be made to use the system hard disk space by selecting the out-of-core option in the solver type.

6. The tapered beam element is modified to provide an easier way to specify the beam cross sectional variation.

7. User Plasticity and Creep routines interface using DLL interface:  
User plasticity routines interface using DLL interface is added. This feature enables user to provide his own DLLs, using any programming language, for specifying the plasticity model.
8. Auto constraining rigid body motions for inertia relief:  
In inertia relief analysis, user is required to constrain the free body motions. The new feature can automatically select the right boundary conditions to take out rigid body motions without over constraining the body.
9. Failure criteria for composite shell element in nonlinear analysis:  
User can input the strength properties in \*MATERIAL data group to evaluate layer failure based on maximum stress, Hill-Mises, Tsai-Wu, and Tsai-Hill (Tsai-Hill criterion can be replaced by a user-defined criterion) criteria for nonlinear analysis also.
10. General beam element (NKTP=12) is added to nonlinear analysis for special application:  
General beam element (NKTP=12) is added to nonlinear analysis for special application where the beam elements are in linear condition and the joint springs are under plastic deformation (plastic joint). This special feature is used in NISA/CIVIL module.
11. User specified coefficient for AUTO constraint:  
User is provided an option to specify a fictitious value for torsional stiffness of the normal rotation (drilling) DOF of shell element in executive card.
12. Partial Fixity:  
Partial fixity is a special feature for beam elements (NKTP=12, 39) to simulate plastic joint condition. It allows the specified percentage of beam end forces to transfer to other beam member at the beam joint.
13. Improve available memory for sparse solver:  
The available memory for sparse solver is estimated internally by subtracting the memory size provided in executive card in input NISA file from the total system memory. This facility helps the sparse solver to optimize the usage of available RAM without utilizing the swap memory. User still has the option to input the sparse solver memory size, in case the estimated memory is insufficient.
14. Static failure of welds:  
Static analysis of welds for failure in, tension, compression and shear is available with the implementation of a new data card - \*WELEMENTS. The welds can be post processed in DISPLAY IV.

## 15. Eigenvalue Analysis Restart 2:

This restart feature is useful in modal dynamics run. With this option, user can run eigenvalue analysis again with \*MODEOUT and \*EIGOUT cards activated, in order to obtain the modal stresses, reactions, internal forces and/or pseudo static solution of missing mass correction.

If pseudo static solution is activated, a new pseudo static solution is computed and overwrites the one from the previous run. Since the pseudo static computation requires stiffness and mass matrix, both file 23 and 24 should be saved from the previous run.

**ADVANCED DYNAMICS**

1. Critical damping value for material damping data in \*MATDAMP:  
User can now specify the critical damping value (CDAMP) for which the default limit is 0.2. This limits the computed modal damping to CDAMP value if it exceeds this specified damping value.

**ROTOR DYNAMICS**

1. Parallel solver (PARDISO) is supported for rotor dynamics for large problems and faster computation.

**ENDURE**

1. Fatigue of spot welds:  
A new feature for estimation of fatigue life for spot welds in the structural components is implemented. The structural stress method is used where the spot welds are modeled as circular beams. The new data card \* WELEMENTS is introduced #.NIS file and the corresponding weld material and dimensional properties are automatically extracted. An option for plotting the weld contours is provided in Endure Shell.  
This feature is especially suited to life estimation of spot welds in automobile components.
2. Enhancement of Endure Shell:  
User can specify any material ID number (previously it was limited to the sequential numbers 1 to 5).  
For Initiation module, a) the dialog boxes are re-organized to make it user friendly for defining input load, b) option for plotting #.a and #.b post files is made available only in DISPLAY IV, c) user needs to specify only 'post file prefix'. Consistent file name for output is automatically chosen by the program and d) user needs to choose only #26.dat file for pre- and post-processing. The #27.dat file is automatically identified by the program.

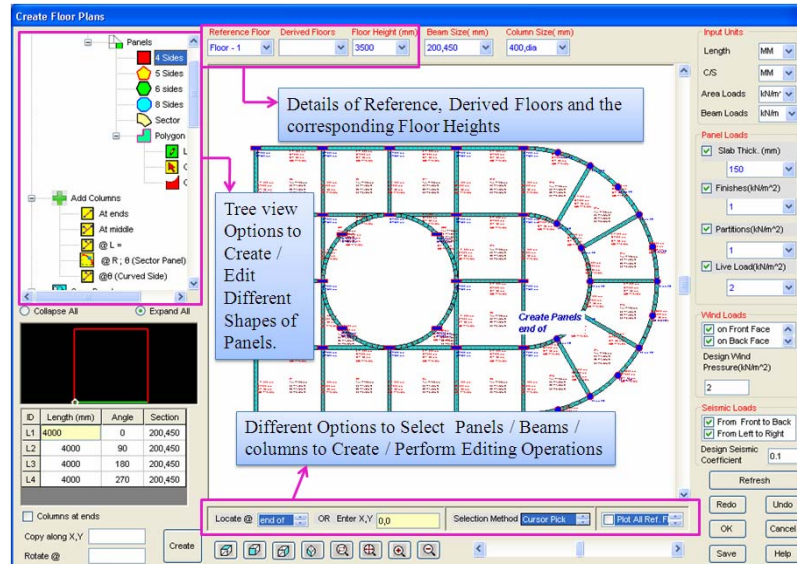
**EMAG**

1. MEG independent execution is introduced. The MEMORY executive data card can be used by the user to avail this facility. This feature fully utilizes the available system memory.
2. HEAT III is incorporated in EMAG. This facility enables user to utilize the enhanced features of HEAT III, to solve coupled EMAG, heat problem.
3. An option is introduced in parallel solver (PARDISO) to make the system use either in-core or out-of-core memory. For very large sized problems when the system RAM is insufficient the solver can be made to use the system hard disk space by selecting the out-of-core option in the solver type.
4. Option is provided to remove negative pivot in system matrix for iterative solver. User should be aware that iterative solver can only solve problems with positive definite matrix. This new option is to modify the negative pivot due to, for example, bad elements in the model.

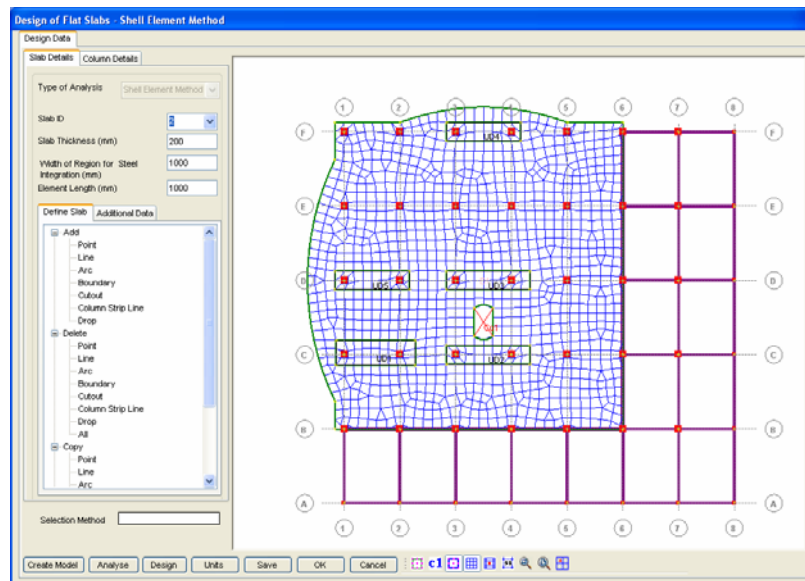
**NISA/CIVIL**

*Modeling (Pre-processing)*

1. Geometry based building wizard is introduced in NISA/CIVIL to facilitate the user to easily model floors in a structure, which can be further analyzed and designed by specifying the columns and loads data.



2. Flat slab modeling is extended for irregular geometries also. With a set of easy to use tools, the flat slab geometry can be defined and the shell model is generated automatically, which can be graphically verified by the user. The design strips identification can be automatic or user specified. Analysis and design can be performed from the integrated environment itself.



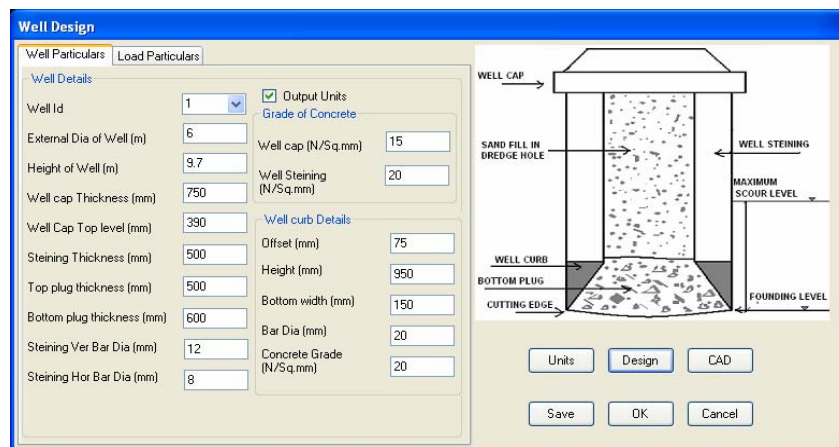
3. Scripting is introduced in NISA/CIVIL for FE modeling through command mode.
4. ASCII input file with extension \*.prc is introduced in NISA/CIVIL.

### Analysis

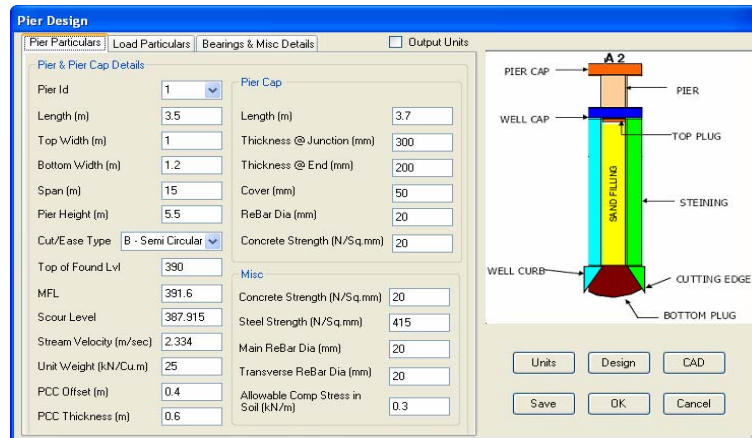
5. A facility is introduced to specify partial fixity at selected beam ends.
6. Member local offsets for beam elements have been introduced.
7. A facility is introduced to offset the shell elements to facilitate the realistic FE modeling.

### Design

8. Structural design of steel members subjected to axial compression, tension, bi-axial bending and shear forces conforming to IS 800:2007 has been implemented.
9. Structural design of steel members subjected to axial compression, tension, bi-axial bending and shear forces conforming to BS EN 1993-1-1:2005 Euro code 3 has been implemented.
10. Structural design of concrete members such as Reinforced Concrete (RC) Slab, RC Beam, RC Column, RC footings, RC Shell elements & RC Retaining walls conforming to BS EN 1992-1-1:2004 Euro code 2 has been implemented.
11. Structural design of gravity walls conforming to IS 456 -2000 has been implemented in interactive mode of design.
12. Structural design of well foundation conforming to Indian standards has been implemented in interactive mode of design.



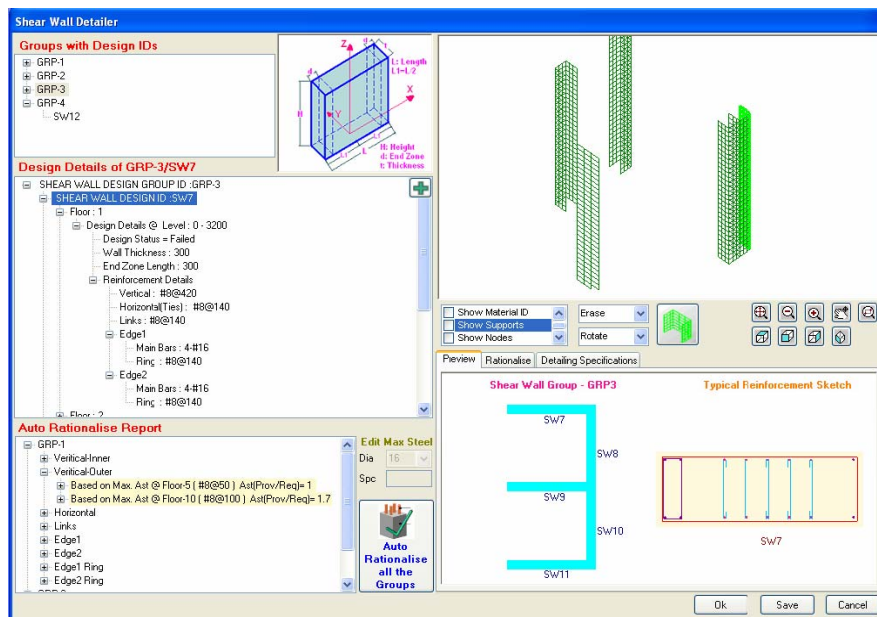
13. Structural design of bridge piers (solid type) and bridge bearings (neoprene) as per Indian standards is implemented.



14. Structural design of irregular flat slab conforming to Indian, British, American & Euro codes of practice is implemented.

### Post-processing

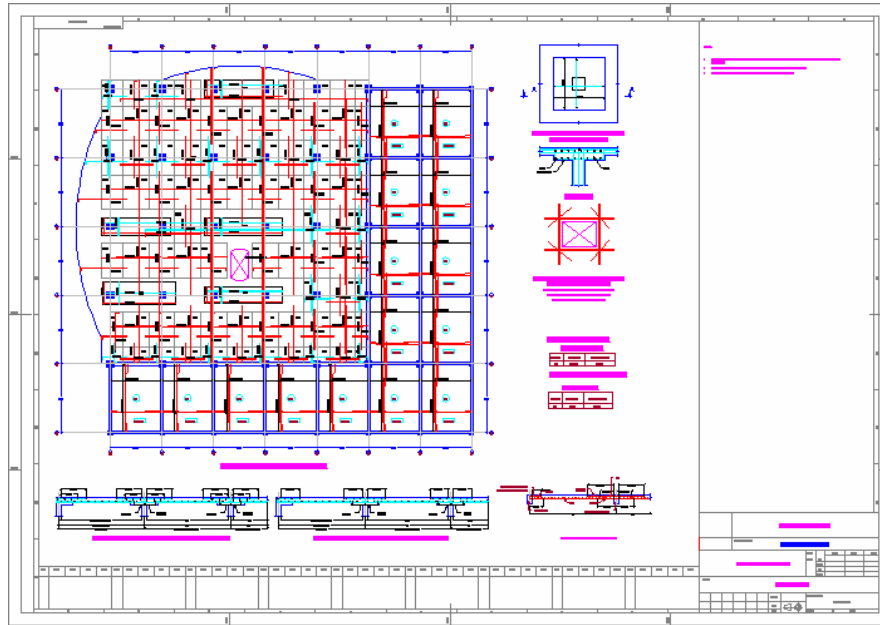
15. Auto rationalization of reinforcement for shear wall is introduced along with a user friendly detailer for further rationalization as per user requirements.



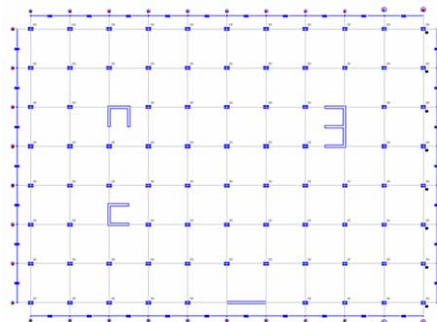
16. Computation of storey drift in pseudo static seismic analysis for framed structures is introduced.
17. Identification & detailing of column strip, middle strip in regular/irregular flat slabs and their re-bar arrangement has been introduced.

*Drawing*

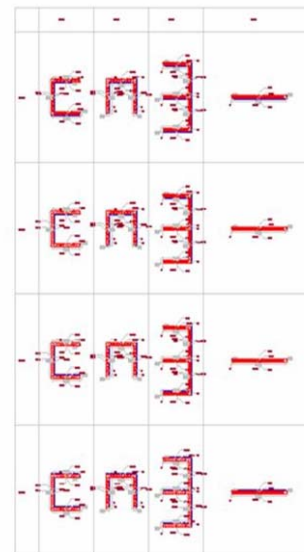
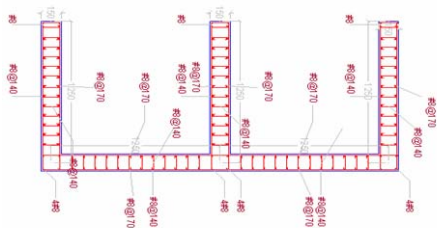
- Automatic generation of structural drawings for irregular flat slab has been introduced.



- Automatic generation of shear wall profile in structural layout plan and as well enlarged views showing structural details of shear walls in tabular format-floor wise is introduced.

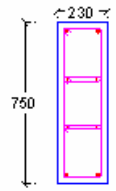
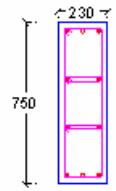
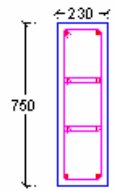
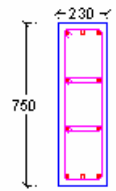


Structural Layout sketch with Shear wall profile.



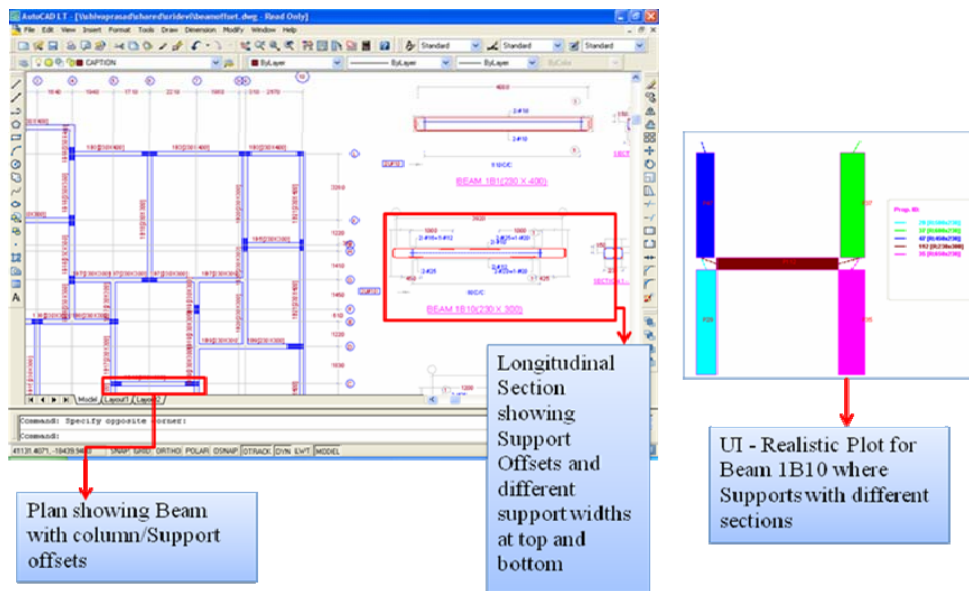
Floor wise detailing sketches of Shear walls

20. Automatic floor wise rationalization of columns with respect to re-bar and their cross sections is implemented.

COLUMNS		C1		C2	
FLOORS					
FLOOR - G	HEIGHT = 3.5m COLUMNS: 12Nos.	4# 16 4# 12	Ties: 8# 120C/C	HEIGHT = 3.5m COLUMNS: 4Nos.	4# 25 2# 12 4# 20
					
	230 X 750	Fe 415	M 20	230 X 750	Fe 415
FLOOR - 1	HEIGHT = 3.5m COLUMNS: 12Nos.	4# 16 4# 12	Ties: 8# 120C/C	HEIGHT = 3.5m COLUMNS: 4Nos.	4# 20 2# 12 4# 16
					
	230 X 750	Fe 415	M 20	230 X 750	Fe 415

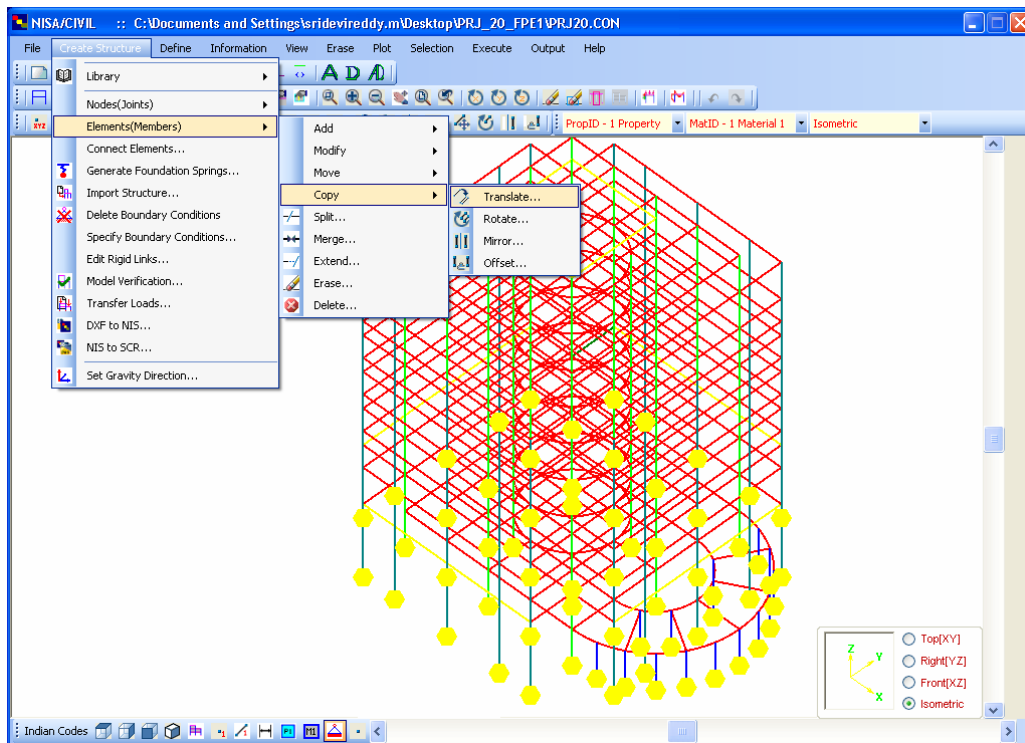
21. Automatic rationalization of RC Slabs with respect to re-bar, edge conditions, spans and their thickness is implemented.

- 22. Generation of general arrangement beam drawings showing plan, longitudinal and cross sections are improved. Automatic rationalization of beams with respect to re-bar, cross sections within the floor and user specified floors.



User Interface

- 23. Improved UI with icons introduced for most of the functionalities, graphic window background gradient option, etc.



24. Tool bars property is now made as undocked so that user can move them around the UI and locate as per user preference.
25. With edit operations, the pop-up menu for selection options is docked to the dialog.

## ERROR CORRECTIONS

### PRE & POST PROCESSORS

#### DISPLAY III

1. Pressure/nodal thickness contours do not plot if same property ID is used for both quadrilateral and triangular elements – Fixed.
2. VEW HID ON is not showing hidden view for some elements when selected using window – Fixed.
3. Problem with adding beam section properties through command mode – Fixed.
4. Session file not reading temperature dependency curve ID applied in property form correctly – Fixed.
5. Problem in writing external results only for active entities. If there is a greater number of load cases, after writing external results for 39 or 40 load cases, DISPLAY gives error – Fixed.
6. DISPLAY III is not plotting lamination angle plot for a composite problem if LAMSQ2 is defined – Fixed.
7. When negative pressure value is given and defined with vector ID, arrow is not plotted in opposite direction and arrow is always plotted in the direction of the vector – Fixed.
8. DISPLAY crashes for history plot when history is not requested in the NIS file – Fixed.
9. Launch Advanced Surface Mesher, click on Add and press Esc key on the keyboard twice, DISPLAY hangs – Fixed.
10. Advanced Surface Mesher, Mesh parameters window does not pop again if it is closed with "Esc" key – Fixed.
11. Advanced Surface Mesher shows wildcards for number of nodes and elements if more than 10E5 – Fixed.
12. Not reading \*SPDISP,\*CFORCE data in NISA file for Linear Transient Analysis using Piezo elements – Fixed.

**DISPLAY IV**

1. Beam splitting limitations of using values like 1000 or more causing unexpected behavior – Fixed.
2. Layer dialog in DISPLAY IV not working as per windows standard – Fixed.
3. Large icons were not saved in the user settings ini file – Fixed.
4. Swapping of data in Moving Load data card under Transient Dynamic analysis – Fixed.
5. Image saving should query the user if the image file with the user specified name already exists – Fixed.
6. Missing icons in all Analysis tool bars – Added.
7. Font color is not changing after selecting color in font dialog – Fixed.
8. Analysis/Fluid+Heat→Material→Fluid, Ratio of Specific Heats (GAMM) remains unchecked with “Add” – Fixed.
9. Analysis→NLStatic→Plastic card's combo boxes problem, when user selects "concrete" from Elasto-plastic material "user defined model" combo should be disabled but is enabled – Fixed.
10. Boundary Condition data related to all analysis is not getting replaced or appended – Fixed.
11. Offset option for shell elements is not reflected in realistic plot – Fixed.
12. CST, FORM command hangs – Fixed.
13. Problem with load/save material library file – Fixed.
14. Discrepancy with macro execution that after empty input values, negative values and giving an input file there is no action – Fixed.
15. Analysis/Heat→Property, Matdir1, Matdir2, Lamangle and Lamsequence2 remains unchecked even after adding the values in respective data – Fixed.
16. DISPLAY IV not saving the modified files if the input file (all supported DISPLAY IV file types) is from Application launcher or through command mode – Fixed.
17. In EMAG Pmagnet data value was not accepting Real or float values and prompting to enter between 0 to 100 – Fixed.
18. Geometry→Circle with Ring option gives full circle – Fixed.
19. Direction arrow size is not proportionate to the entity size – Fixed.
20. After Edit→Erase, FEM→Element→Misc Option→Plot with All option also displays BCs – Fixed.
21. Proper images not saved with image saving command while in session or macro mode – Fixed.

22. In post processing of file when we select Phase plot, the model disappears and only boundary condition is displaced on the screen – Fixed.
23. NISA logo not available in the screen while viewing post results – Added.
24. Information window is having problem. In case of operations like Automesh, deletion of entities in larger numbers etc, which need to show messages continuously, information window goes blank during entire operation and shows the information only at the end – Fixed.
25. In DISPLAY IV, for all Analysis→Executive dialog, "Enter File Name Prefix" is not updating with user given file name prefix – Fixed.
26. File name is not seen on reading the NIS file – Fixed.
27. In View→Solid cut section, Add planes using Grid/Node Ids and after selection of Grids / Nodes, DISPLAY IV crashes – Fixed.
28. Problem in creation of Rectangle and Circle on user-defined Workplane – Fixed.
29. Boundary Condition color not changing on Plot – Fixed.
30. Problem with setting the color for layer – Fixed.
31. Modifications of Layers dialog for ease of use – Fixed.
32. Some dialogs are not appearing when model is in fly view – Fixed.
33. Tool tip in Analysis tool bar displays wrong information for some icons – Fixed.
34. Unnecessary messages are shown in information window for realistic plot – Fixed.
35. Error while compacting the Node IDs as "Error in {MAVAIL}. Module un-defined. !! Please report" – Fixed.
36. Cancel button not doing proper job where incremental ID is present – Fixed.
37. Heat Analysis toolbar icons are wrong – Fixed
38. Beam offset issue while using rectangular cross-section – Fixed.
39. Crash in using domain handlers with translated geometry – Fixed
40. During post-processing toggling between Contour plot, Graph plot, and again Contour plot, results in the color bands and titles to be missing from the Contour plots – Fixed
41. More than 512 characters in command line causes DISPLAY IV to crash– Fixed
42. Method of Formulation of Hyper elastic Material (HYPER) is blank after saving and reading from a NIS file – Fixed.
43. When negative pressure value is given and defined with vector ID, arrow is not plotted in opposite direction and arrow is always plotted in the direction of the vector – Fixed.
44. Beam elements seem broken while plotting post contours, with the exception of von-Mises stresses– Fixed.

45. With menu Elements→Composites→Single Set for a 3D Laminated Composite General Shell element gives Memory error – Fixed.
46. Problem with general shape C section if the feed value in metric system – Fixed.
47. Flexi grid copy and paste not working – Fixed.
48. Animation in DISPLAY IV is slow when compared to DISPLAY III – Fixed
49. Executive setup shows untitled file if a NIS file is opened from Recent file list – Fixed.
50. Geometric Association for Element selection along with Field quantity for pressure is not working – Fixed.
51. Reading nis/dbs/neu/dat/ses file is not working if space is there in the file path – Fixed.

**CAD TRANSLATORS**

1. SAT/IGES to Neutral convertor problem while converting more than 126 parts – Fixed.
2. IGES issue for converting sheet body files – Fixed.

**FEA TRANSLATORS****FE data translation from NISA to ANSYS**

1. Pressure was being applied to wrong surface on NKTP = 4- Corrected.
2. Handling of multiple load cases corrected.
3. NKTP=33, NORDR=1 was being translated to SHELL91- Changed to SHELL181.

**PROCESSORS (SOLVERS)****NISA II**

1. In nonlinear static analysis using arc-length scheme with SPARSE or iterative solver, the program crashed if arc-length scheme was used after the first event – Fixed
2. Arc-length method crashes if number of arc-length steps in one event was more than 53 - Fixed
3. Unnecessary file FORT.4 was created using ITERATIVE solver – Removed.
4. Incorrect reactions for inertia relief if there is a rotated coordinate system defined in the model - Fixed
5. NISA2 crashes in eigenvalue analysis using Guyan reduction method with lumped mass formulation – Fixed
6. For moving path (Linear Static Analysis) has a segment in a triangle elements (NORDR=10 or higher), NISA2 issues an error message "cannot find continuous path"- Corrected.
7. Error of zero record for file 26 in element stiffness and load calculation in nonlinear analysis with composite element using \*LAMSQ2 data card with IESHR (Key to activate equivalent transverse shear modulus computation) = 1 - Fixed
8. Nonlinear beam element (NKTP=39) reports incorrect beam end moments when beam orientation is defined using a vector – Fixed
9. Vector to define beam orientation is not consistent in linear and nonlinear analysis for NKTP=39 - Fixed
10. Beam load results are incorrect if beam offset is defined in beam local axes – Corrected.
11. For PARDISO Solver Wave front optimization is to be skipped – Corrected.
12. PARDISO solver crashes with MPC equation. PARDISO solver interface handled MPC with BC incorrectly – Corrected.
13. Problem cannot converge in nonlinear analysis with MPC equations using CASI sparse solver – Fixed

**ADVANCED DYNAMICS**

1. Dym2 is crashing for centroidal stresses for shell elements – Fixed.
2. ZPA computed using CFREQ is not scaled up with scale factor in \*GROUND - Fixed

**ENDURE**

1. Error in life estimation using User Defined S-N data – Corrected.

**NISA/CIVIL**

1. The data specified in 'Bridge Form' dialog was not retained after exiting the dialog – Fixed.
2. In 'New project' dialog, Default Units for Input and Output units were disabled – Fixed.
3. In 'Load Cases & Combinations' dialog, selection of P-Delta Load case ID from "Dynamic/P-delta Load Case ID" combo box resulted in run-time error – Fixed.
4. In 'Load Cases & Combinations' dialog, with the selection of Dynamic Load case ID from "Dynamic/P-delta Load Case ID" combo box, the 'Base Shear' values listed against the defined 'Rules' were not updated (values belonged to the project opened earlier in the same session) – Fixed.
5. Execution time for Beam drawing generation reduced considerably.
6. In the Layout drawings, the member offsets for columns were incorrect - Fixed.
7. Exiting 'Selection Method' dialog after storing Ids through graphical selection, graphic window retained "End Selection" & "Reset Selection" buttons and the menus were disabled (performing Elements\Add between Nodes through 'Pick nodes' option and without quitting that mode, immediately performing Id selection through 'Selection Method') – Fixed.
8. For steel design against high axial force, the program generated Fortran error and crashes out (with Codes of practice "BS: 5950-2000", "AISC-ASD-2005", "AISC-LRFD-2002", "AISC-LRFD-2005") – Fixed.
9. In a New Project with no data, selecting Member information with Listing of design results, accessing design summary or using erase options such as Border/Border crossing resulted in application crash – Fixed.
10. When the coordinate origin was not (0,0) for the structure; In Layout sketch, the Horizontal gridlines were shifted – Fixed.
11. In Flat Slab Detailer with American Code of practice, when bar diameter was changed, program issued message to limit maximum spacing as well as minimum spacing – Fixed.
12. In Flat Slab Detailer using the American Code of practice, when the bar diameter or spacing was changed, the program issued a minimum spacing message. Program changed the parameter against cancel operation also. Further, generating drawing resulted in run-time error – Fixed.
13. In the bar diameter text box, when '0' was entered temporarily, and later changed to a valid diameter, the program generated a run-time error – Fixed.
14. In Flat Slab Detailer, the bar diameter was changed for only one of the panels. But, generated drawing includes Quantity Table for bars – Removed.
15. For American Code of Practice, the bar spacing was represented with 14 digits precision – Fixed.

16. Typical section sketches represented reinforcement protruding beyond the section outline – Fixed.
17. RC Column Design Report: With alphanumeric group IDs for columns, execution of design report against Indian Concrete Codes of practice IRC-1966(Rev-2000), IS-456:2000 Working Stress and IS-3370-1967 resulted in run-time error – Fixed.
18. Execution of design report for columns with Uni-axial moment 'My', the application crashed – Fixed.
19. For Circular Columns, tie details were not listed in summary table – Corrected.
20. RC Corbel Drawings: For default data, bottom stirrups along the sloping edge were projecting beyond the corbel edge Line. Also, line color for Bars & dimensions were same (also not uniform with other sketch components) – Corrected.
21. For units other than "SI", bar caption was represented with large precision – Fixed.
22. Quantity Table exceeded the Plot Area Limit (mixed with the Notes area) – Fixed.
23. Retaining Walls: Counterfort Retaining Wall Plan View drawing did not have any Title. For Counterfort defined only on Heel side, drawing represented counterforts at Toe side also. Top width of Counterfort at Heel was not dimensioned – Fixed.
24. For Retaining wall of Type-3 (with Key), the Key width listed in CAD dialog was incorrect – Fixed.
25. For Free Standing Stair, 'Drawing Settings' button was enabled after design completion. Clicking the button resulted in a Run-time error – Fixed.
26. When Design Summary for Footing was selected, right click options in graphic window became inactive (un-selecting the option did not change the status) – Fixed.
27. 'Layout editor' menu item was disabled for a framed structure project (by opening the framed structure project after opening an interactive project) - Fixed.
28. When the element IDs exceeded the total number of elements present, launching Layout editor resulted in Run-time error – Fixed.
29. Dialog caption 'Structure Layout Editor' as described in user manual is not maintained in the application – Corrected.
30. For an interactive project already designed with ASD-2005 code, opening the project in a new session and directly executing Design Report listed only summary table for output writing and the report had only the table title (No member design output & summary values) – Fixed.
31. For Beam Drawings, Table for Shear Reinforcement and Longitudinal Section of Beams overlapped – Fixed.
32. While Property data was not defined for some of the members, generating Steel GA Drawing resulted in run-time error – Fixed.

33. For Analysis report including Tension Only/Compression Only elements, the generated report has BLANK spaces as entries. Also, all the data columns for Element Information are mismatched – Fixed.
34. For Analysis report generated against Dynamic Load case, with the selected dynamic load combination rules for printing are not specified for analysis, the report entries format is not maintained – Fixed.
35. For Analysis report generated against Dynamic Load case, with nodes selected for Reaction or Displacement, the application generates Runtime error and crashes - Fixed.
36. In the Import structure preview window, shell elements are not plotted properly – Fixed.
37. For elements split option with shell elements, 'No of Divisions' field is enabled – Fixed.
38. With Gravity as Y vertical, in the Layout Editor, the views caption, and the displayed view are mismatched – Fixed.

## LICENSING & PACKAGING ENHANCEMENTS

1. This version supports time stamped soft licensing for Windows XP 64bit O/S and VISTA (Business, Enterprise and Ultimate) 32-bit and 64-bit platforms which had some limitations with earlier version.
2. NISA Modules, CAD Translators, and Manuals are packaged in a single DVD. (Distribution of NISA on- CD ROM is discontinued)

## O/S SUPPORT

This version of NISA is targeted for 32-bit as well 64-bit platforms. The Operating Systems supported are:

1. Microsoft Windows XP (SP2 & SP3)
2. Microsoft Windows 2003 (SP1)
3. Microsoft Windows Vista® (Home, Ultimate, Business & Enterprise Editions)