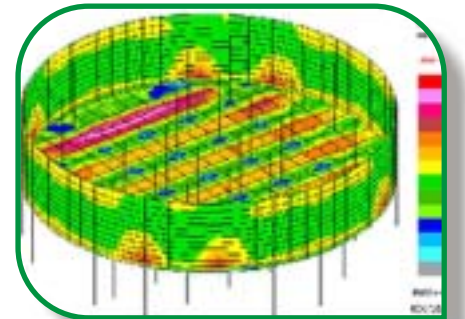
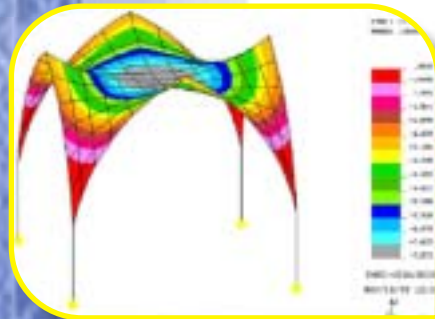
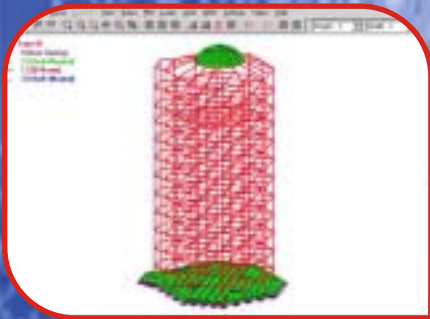
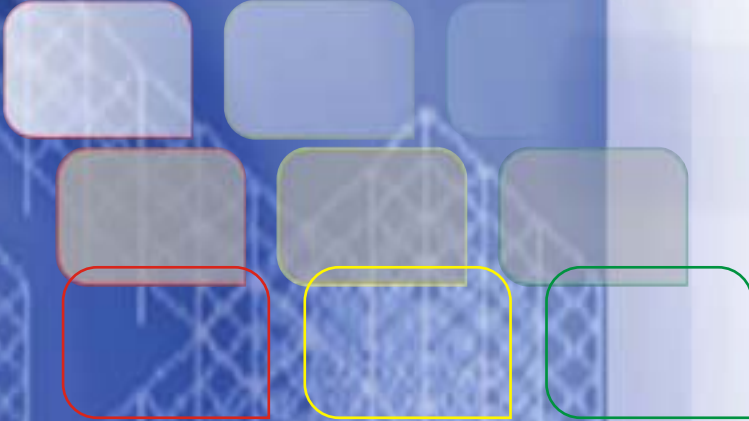


NISA - Civil



Choice of Civil
& Structural
Engineers Worldwide

NISA - Civil



NISA/CIVIL, from NISA family of finite element programs offers CAD based solutions to a wide variety of problems encountered in the Analysis and Design of Reinforced Concrete and Steel Structures. Backed by powerful NISA II Analysis and DISPLAY III/IV the graphical Pre and Post processor of NISA family of programs, NISA/CIVIL provides excellent tools for modeling, associating design information and carry out design process in Limit state and working stress methodologies of design. Design results are processed to produce structural engineering drawings in AutoCAD environment. Equipped with an extremely user friendly GUI and graphic displays, NISA/CIVIL, presents an elegant platform for analysis and design of different types of structures encountered in practice.



Finite Element Modeling

Structure primitives for 2D and 3D rigid frames and trusses.



Library of Industrial Structures and Structures Gallery.



Civil engineer friendly graphic editor for creating FE model of skeletal structures, shell structures and spring elements with the following options:

|| Nodes: Create / XYZ coordinates / Between two nodes || Nodes: Move | Translate / Rotate / Mirror / Scale || Nodes: Copy | Translate / Rotate / Mirror / Scale || Nodes: Erase || Nodes: Delete || Elements: Add || Elements: Modify || Elements: Move / Translate / Rotate / Mirror || Elements: Copy / Translate / Rotate / Mirror || Elements: Split || Elements: Merge || Elements: Extend || Elements: Erase || Elements: Delete || Connect Elements || Import Structure || Model verification || Undo/Redo Operations for Editing ||



Powerful model verification allows for checking & rectifying the presence of unconnected nodes on members (floating connections) duplicate members and coincident nodes in the FE model. The user can automatically, compact nodes and elements, reset element node order based on geometric near node, reset origin to (Xmin, Ymin and Zmin), renumber nodes and elements based on geometric nearness.

New

- Rigid links can be specified and viewed
- Specified Property Ids (for example braces) can be ignored during panel generation
- Beam elements can be extended/ shortened in the editor
- Shell elements can be converted from linear to quadratic and vice-versa

- Structural shape Library including generation of Domes, Cones (frustums), Cylindrical shells (single, multiple), and folded plates, with provision for generating a reticulate model (model with beam elements)
- Easy to use feature of adding two different structural models from Library / Gallery/ Industrial Structure / Bridge form/ Shape Library with and without coordinate offsets
- Additional member renumbering schemes such as beams columns, or columns beams

Bridge Module: Automatic FE-Generation of Bridge deck as Beam/Grillage model.

New

- Bridge form can be generated with automatic shell element F.E. model based on longitudinal and C/S properties.



Interface to DISPLAY III/IV: Complex and intricate structures with shell elements can be modeled using DISPLAY III/ IV; the Graphical Pre and Post processor of NISA Family of finite element programs.

Interface to AutoCAD: Export finite element data from NISA/CIVIL as geometry data into AutoCAD. Import geometry data in DXF format from AutoCAD as finite element data into NISA/ CIVIL. Elements are automatically split and nodes are introduced at points of intersections.



New

- Improved AutoCAD DXF file interface with facility to import member cross sections and column locations from plan
- Facility to add multiple DXF files for different plans with automatic generation of columns

NISA - Civil

User defined vehicle

- Automatic Generation of curved vehicle and curved paths
- Generation of Offset Wheel Loads with torsion (transfer of load on single member)
- Specify path by picking the nodes in sequence
- Specify path which is pre-defined in a text file
- Automatic generation of offset path for an existing path Id
- Automatic extension of existing path for a specified extension length
- Reversing direction of an existing path automatically
- Members of selected property Ids only can be considered to distribute wheel loads



- Vehicle loading height can be considered in case of (Wind/ Braking/Seismic) forces
- Lateral loads can be considered perpendicular to the path

Pressure Loads

Pressure Loads can be specified on shell elements. Different values of pressure can be specified on each node of a shell Element.



New

- Shell elements subjected to varying pressures
- Shell pressures can be applied in global directions

Load case dependent Member End Release

End release specification is linked to load case Id. The main advantage of this approach is that, different end release condition for an element can be specified in conjunction with loads, as analysis with different load cases and can be done in single session. Remodeling, multiple analysis and manual superposition of results are avoided.

Load case dependent Inactive Member specification

This feature is particularly useful for performing construction sequence analysis for a particular loading condition. This avoids creating different types of structures and to perform independent analysis as members specified as 'Inactive' will not be included while formulating stiffness matrix and load vector.

- Line Loads: It is possible to specify varying line load on set of elements which are on the same line
- Load information can be copied from one load case to another load case
- It is possible to change physical property of elements in a load case



UNITS SPECIFICATION

SI, MKS, FPS and User Defined units supported. The User Defined option has dual units for length and force. Desired system of units can be specified at the beginning or as and when necessary.

NISA ANALYSIS

- NISA/CIVIL provides a seamless interface with NISA II for performing linear static and dynamic (response spectrum) analysis of two/ three-dimensional structures
- Multiple analysis features such as constructions sequence analysis and P-Delta analysis are supported with specific reference to multi-storey frames
- In-addition to Linear Elastic response, Non-linear response due to material, geometric or combined nonlinearity for a spectrum of structures can be predicted



DESIGN MODES

Three modes: Integrated On line, Integrated Off line and Interactive.

Integrated Off line mode in NISA/CIVIL is a special and Civil Engineer friendly option. Alternative designs can be easily performed without repetitive analysis. Interactive option is more functional as multiple elements can be designed in a single session.

STRUCTURAL DESIGN

Design modules are developed as per American, British, European, German, Indian and Japanese standards.

Reinforced Concrete Elements

- ACI 318M-(Revised 1994), BS 8110-1997 (Part1), DD ENV 1992 1-1: 1992
- IS 456-1978, IS 456-2000, AIJ Standards for Steel Reinforced Concrete Structures (1987)
- IRC-1966 (REV -2000), IS 3370 (1987): Design for uncracked sections

Structural Steel Elements

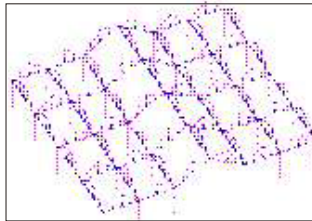
- AISC-ASD, AISC-LRFD, BS 449: Part 2: 1969, BS 5950:Part1: 1990, DD ENV 1993 1-1: 1992
- DIN 18 800, Parts 1 and 2, 1990, IS: 800 1984, AIJ Specifications, Society of Steel Construction of Japan



Automatic Load Generation

Frame Loads

- Transfer of floor loads such as DL, LL and Snow loads from panels to supporting members based on either Two-way or One way distribution
- Automatic Generation of floor loads on supporting members for non-rectangular shaped slab panels and sloped panels
- Joint loads due to Wind and Seismic effects (seismic coefficient or response spectrum method with accidental torsion) as per Code Provisions or user specified values

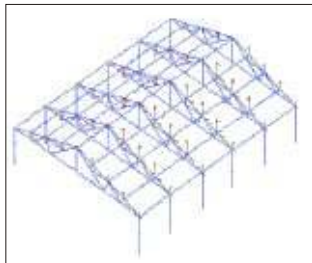


New

- Specification of sloping slabs made simpler
- Seismic loading computation extended to inactive load cases
- Automatic generation of panels, assignment of loads etc for preliminary designs

Truss Loads

- Joint loads due to dead and live loads on trusses
- Joint loads due to Wind and Seismic effects (seismic coefficient or response spectrum method) as per Code Provisions or user specified values



Loads on structure are automatically generated based on following codes of practice

- ANSI/ASCE 7-95, DD ENV 1998 1-1 Eurocode 8, IS: 1893 1984
- IS: 875 - 1987 (Parts 2, 3 and 4), BS: 6399 Part2 1997, BS: 6399-1998 (Part3)
- CP3: Chapter V-1972 (Part 2), AIJ Recommendations for Loads on Buildings, IS: 1893 2002 (Part 1)

New

- Automatic identification of trusses based on property Ids of Elements

Prestressing Loads

- Loads due to prestressing (either before or after placing). Cable profile can be either parabolic or linear

- Cable Profile along a line: It is possible to specify Cable Profile (Linear/Parabolic) along a set of elements which are on the same line

Dynamic Loads

Mass elements for Eigen analysis and load combinations to account for reversal of forces from subsequent response spectrum analysis for seismic design purposes are automatically generated

New

- Pre stored spectra as per IS 1893 2002 for Dynamic loading.
- Missing mass correction specification including cutoff frequency

Temperature Loads

Thermal loads can cause axial and bending deformation of the beam element. Nodal temperatures or temperature differences can be specified at joints. Nodal temperatures are used to compute axial expansion or contraction. Nodal temperature differences will be used in conjunction with the sectional dimensions to compute bending in local XY and XZ planes.



Moving Loads

NISA/CIVIL has powerful Moving Load generation algorithms to generate automatically, the magnitude of loads transferred to members due to movement of either single or group of vehicles along different vehicle paths on a bridge structure.



In addition to load cases corresponding to static loads, distinct load cases corresponding to different vehicle positions are also created automatically. Static analysis is performed for all the load cases.

Following vehicles are included in vehicle database.

- AASHTO Nominal Hs20-44 loading for different vehicle lengths (8.5344 13.4112 M)
- American Railway Engineering Association Cooper E-80, American Axle Loads for Two Units of Heavier Diesel Locomotives
- BS: 5400, Nominal HB Loading, RU & RL Railway Bridge Live Load
- Indian Railway Stds MBG-1987, MMG-1988
- I.R.C Tracked & Wheeled Vehicles (Class AA, 70R, A & B)



New

- Steel designs as per ASME code
- Summary Table for Steel Design is provided



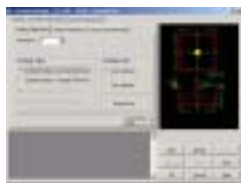
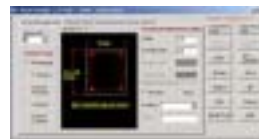
Following structural elements can be designed

RC Slab panels with different support conditions and Flat slab systems. Short and long term deflection computation in slab panels. Design of sector, circular, triangular and skew panels, waffle slabs and spherical domes based on theory of plates and shells.



New

- Design of slabs with Concentrated loads using Pigeaud's Curves
- **RC Beams** subjected to Flexure, Shear, and Torsion
- RC Columns subjected to Axial loads with Uniaxial and Biaxial bending based on Interaction or equilibrium approach.
- **RC Footings:** Isolated footing of constant and varying thickness with or without pedestals, combined footings Solid Slab, Beam and Slab
- RC Shell elements



New

- Automatic Sub-Load Combinations (16 No.) for shell design under dynamic loads
- Structural Steel Elements: Code checking of Standard (Channel, I, Angle, T, Pipes and RHS/SHS) or user defined sections subjected to axial, bending and torsional effects along with recommendations in case of inadequacies.

Design of different types of built-up sections and Plate girders

- Double channel box, Double channel back to back
- Double I section, Crane Girder section, I section with cover plates
- Double angle back to back, starred equal angle section, four

equal angles section

- Double Boxes section (Horizontal and Vertical)

RC Retaining Wall T or L shapes with or without keys and batter towards heel or toe.



Design of Overhead and Ground Level Water Tanks of following types based on IS 3370 parts I - IV

- Intz tanks with/without Bottom Dome
- Intz tanks resting on Columns/ Cylindrical Shafts
- Conical tanks with/without Bottom Dome.
- Rectangular, Cylindrical, Annular Cylindrical Ground level Water Tanks



Design of Pile Foundations: Design of Bored cast Insitu, Friction, End Bearing, Under Reamed and Precast-Driven Piles with pile caps as per IS -2911.

- Design of General shaped columns (viz. T, L, + etc.) with generalized steel arrangement.
- IRC-1966 (REV-2000): Design of members conforming to IRC Code which includes design of Slabs, Beams, Footings, Shell Elements, and Retaining walls
- Working stress Method (IS456-2000): Design of members conforming to working stress method which includes design of Slabs, Beams, Footings, Shell Elements, and Retaining walls
- IS 3370: 1987: Design of members conforming to IS 3370: -1987 which includes uncracked design of Slabs, Beams, Footings, Shell Elements, and Retaining walls
- Implementation of Seismic Design as per IS 1893-2002
- Steel Design optimization: It is now possible to optimize the section in steel Design for elements individually and/or group wise; with automated reanalysis using a given set of properties
- Sectional forces of Structural members selected for design with respect to Load cases/ combinations can be stored in a file along with force envelope values. For RC sections stresses can also be printed for both regular and general shaped cross sections.

CAD DRAWINGS

In NISA/CIVIL post-processing is not limited to analysis results only. At the click of a button, Design results are processed to produce design drawings of good quality in AutoCAD environment.

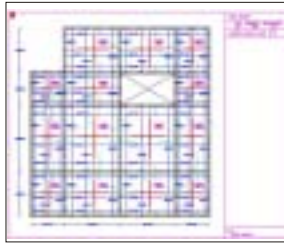
Separate drawings are made for different structural elements even though all of them are designed in the same session.

Drawing entities are present in different layers and colors for easy identification and editing. They can be customized as per requirements.

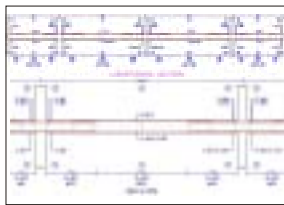
NISA - Civil

Following sectional views can be drawn

- **RC Slab Panels:** Sectional top view indicating top and bottom reinforcement at different floor levels



- **RC Beams:** Longitudinal Section and Cross sections at support and span regions indicating rebar arrangement and stirrup details along with framing Plan.



- **RC Columns:** Cross section along with reinforcement details in tabular form. Columns having similar reinforcement arrangement are grouped together

- **RC Footings:** Sectional top and front views indicating reinforcement details for Isolated and combined footing of solid slab type and cross sections in beam and slab footing

- **RC Retaining Walls:** Longitudinal and cross sectional views indicating reinforcement details in stem, base and keys if any

- **Steel Structural Elements:** Front view along with section designation, sectional details and profile of the section of individual elements in a tabular form

- **Structural Layout drawings:** Top view indicating Structural Grid, Columns and Footings layout along with area of excavation.

- **Over Head and Ground Level Water tanks:** Plan, Sectional Elevation, cross-sections for beams and columns are generated.

- **Shells:** Reinforcement requirement in the form of color contours.

- **Seismic detailing for Beams and columns**

- At least 1/3 rd span steel is extended to the supports in Design of RC Beams for Indian Standard



Processing of design results to produce structural engineering drawings along with concrete and bar wise steel quantity is a special feature in NISA/ CIVIL.

PRE AND POST PROCESSING

- Options to view the structure in different planes and isometric view with zoom pan and erase facilities
- Members can be erased or selected or deleted by several options such as cursor pick, window, crossing and global options such as

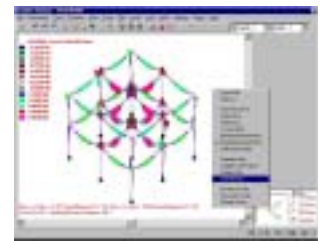
elements parallel to X/Y/Z axes, elements parallel to XY/YZ/ZX planes, elements having specific property ids or material ids or NKTP values

- With 'Erase' option you can temporarily switch off selected elements from display. You can view them again by using 'Reset' option
- Online display of FE model data for selected Nodes, Elements and Panels

- Pop-up Menus for frequently used menu items

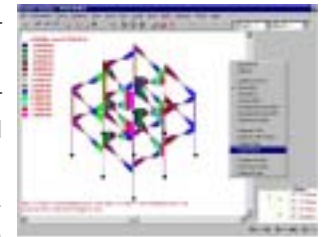
- Search specific Node or Element in graphic display

- Associating design data specification to Beams, Columns, Footings and Steel sections with reference to Design Group Ids & Element Ids



- Display of loading diagram for different load cases

- Listing of member end forces for different load cases for a selected member



- Display of bending moment, shear force and deflection diagrams along with listing at twenty different sections

- Animation of deflected shape, Eigen modes & Stress Contours

- Design Results viewer for Individual or failed elements

- Distance between Nodes

- Numeric count of visible entities

- Facility has been provided in Graphic Editor to model Shell and Spring Elements

- Facility has been provided in Graphic Editor to reverse shell faces.

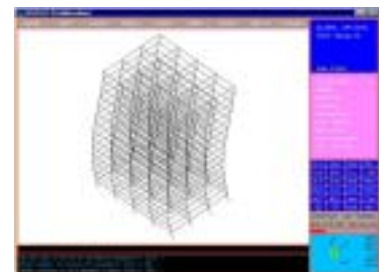
- BMD/SFD can be viewed with color band contours

- Selection of entities by Border method (entities fully inside/ crossed)

- Force Envelope Diagram for complete model in graphic display for selected Load cases/ Load combinations with automatic identification of moving loads

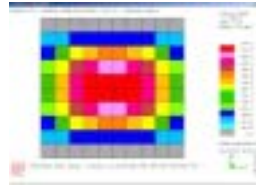
- Line elements in a model can be graphically viewed along with their respective lengths

- Load values can be displayed along with the



loading diagram

- Slab Panels with different load intensities can be displayed in different colors



New

- Designs Interrogation Design parameters can be viewed on screen
- Selected elements are highlighted with a different line thickness. The same applies to viewing by property ID
- Force factors such as $P_u / f_{ck}BD$ and $M_u / f_{ck}BD^2$ used in Concrete Design can also be plotted as is done for BMD / SFD etc.
- Concrete Beam Design for Indian code can also be displayed graphically (Contours and Curves)
- Steel Interaction Ratios (after design) can also be displayed graphically
- Values of Bending Moments etc can be displayed at specified intervals along the length of members
- The Colour Band Labels, Information Data Grid, Axis Display etc are floated so that they can be positioned anywhere on the screen
- Bending Moments, Force Factors etc can be output as a table in a report file
- Loads can be viewed simultaneously with corresponding BMD/SFD. Also loads are retained with change in view
- Displacements and Reactions can be displayed at the nodal locations for documentation
- User defined view orientations can be stored and retrieved easily. These can be obtained by view rotation
- The Element Property ID, Material ID and Lengths can be displayed along with Element Ids
- Viewing of Mass, Pipe and 3D Solid elements are supported including realistic plot
- Realistic plot improved for better representation of actual dimensions and CG
- BMD, SFD etc can be exported to AUTOCAD

SMART AND CIVIL ENGINEER FRIENDLY FEATURES

- Dual unit specification for data input and output of results
- Display of BMD, SFD with or without values or display of only magnitudes
- Automatic selection of all slab panels for associating panel loads
- Automatic computation of boundary condition for slabs
- Automatic identification of continuous beams in selected or all floors along with Alpha-numeric Group ID definition for beams and columns based on floor position ex: 1B1, 4C12

- Associating design data specification of first panel ID to other panel Ids
- Plot Model with distinct colors based on Material or Property Ids or Element Types and
- Realistic plot
- Single click option to delete definition of Panels, Trusses, Loads and Properties
- Grouping of Nodes, Elements and Panels for selection by group name
- Weight and length of visible line elements
- Design Results viewer for Individual or failed elements
- Failed elements can be viewed graphically after Design
- Automatic generation of Foundation springs
- User Defined element renumbering option provided during Model Verification
- Automatic identification of Boundary beam elements and elements at each level for erase/select operations
- Load information can be copied from one load case to another load case
- It is possible to change physical property of elements in a load case
- View Element Length on the screen
- User specified title for Loads in a Load case
- User specified title for material and sectional property Ids

New

- Loading can be transferred from specified coordinate points to the structural model
- Selection of failed elements for property modification
- Property specification of Pipes, Elbows and Tapered beams. 3D beams can be reassigned to these element types
- Facility to Modify and Save Default design parameters
- Analysis using existing analysis data files, Design of selected components in On-Line Mode
- Stress Computation augmented for changed Cross Sectional Properties

NISA - Civil

REPORTS

Basic Reports

These are ASCII reports. These include

- Pre-analysis report
- Summary and detailed Design reports
- Analysis output report

Selective Reports

- Facility to generate report consisting of details such as, Geometric Info, Member Forces for selected load cases and load combinations and Support reactions in concise format
- Member forces can be generated at various sections. Number of Sections can be in the range of 2 to 11. Locations are computed as ratio of element length



Formatted Reports

- Design Report generator for generating different types of formatted reports in .RTF format which can be used as submission for proof checking
 - Graphics + Formatted text
 - Graphics + Unformatted text
 - Graphics only
 - Formatted text only
- Report Navigator to view or edit or print reports



New

- Improved (and compact) Beam, Column, Footing Design reports

Cranes Software International Limited is a leading provider of Computer Aided Engineering (CAE) services to the Automotive, Aerospace, Energy & Power, Civil, Electronics and Sporting Goods industries. Over 70 dedicated scientists, technology architects and software engineers providing NISA based solutions have helped major engineering companies reduce analysis turnaround time, improve user productivity, and ensure faster return on investments. The Company has its presence in 33 countries across the world and has a user base of more than 350,000.

With a mission statement to provide its customers the best in scientific technology and to enable its customers to define new limits, Cranes is setting new standards in the scientific and engineering field. For more information, please visit www.nisasoftware.com Email: nisa@cranessoftware.com

Cranes Software International Limited



Corporate office
4th Floor, Block 1,
Shankar Naryana Bldg
#25, MG Road,
Bangalore - 560 001
Ph: +91 80 51231275 /6 /7
Fax: +91 80 51231274

Bangalore
29,7th cross,14th Main,
Vasanth Nagar,
Bangalore - 560 052
India
Ph: +91 80 22381740 /41
Fax: +91 80 22268410

Delhi
C -100, Shivalik,
New Delhi 110 017
India
Telefax: +91 11 6674891 /92

Hyderabad
3rd floor, 6-3-346/1, Road.no.1
Banjara Hills
Hyderabad 500 034
Ph: +91 040 55840024 /25 /26
Telefax: +91 040 5584 0028



Wölfel Beratende Ingenieure GmbH + Co. KG
Max-Planck-Str.15 * 97204 Höchberg
Tel.: + 49 931 49708-600 * Fax +49 931 49708-650
E-Mail: wtp@woelfel.de
Internet: www.woelfel.de

©2004-2005. Cranes Software International Limited. All rights reserved.

All other company and product names are trademarks and/or registered trademarks of their respective owners.