

New Paradigm in Advanced Structural Analysis



Configuration



Analysis Capabilities

- Static Analysis
- Construction Stage Analysis
- Reinforcement Analysis
- Buckling Analysis
- Eigenvalue Analysis
- Response Spectrum Analysis
- Time History Analysis(Linear/Nonlinear)
- Static Contact Analysis
- Interface Nonlinearity Analysis
- Nonlinear Analysis(Material/Geometric)
- Concrete Crack Analysis
- Heat of Hydration Analysis
- Heat Transfer Analysis
- Slope Stability Analysis
- Seepage Analysis
- Consolidation Analysis
- Coupled Analysis(Fully/Semi)



Applicable Problems

General Detail Analysis (Linear, Material/Geometry Nonlinear)

- General detail FE analysis (linear static/dynamic analysis of concrete and steel)
- Buckling analysis of steel structure with material and geometric nonlinearity

Concrete and Reinforcement Nonlinear Analysis

- Detail analysis of composite structure (steel + concrete)
- 3D detail analysis considering steel, concrete and reinforcement simultaneously
- Detail analysis of CFT (Concrete Filled Tube) Columns and analysis of the longterm behaviour (differential settlement)
- Crack initiation and propagation in concrete structure
- Discrete Modeling and analysis of masonry
- Composite Modeling and analysis of wall in shear
- Detail analysis for tendon anchorage

Applicable Problems

Thermo-Elastic Analysis (Heat Transfer, Heat of Hydration)

- Analysis of heat of hydration (general, special, nonlinear)
- Detail analysis for assessment of shear capacity of pavement (Debonding Failure)
- Analysis of thermal effect due to the asphalt pavement (Guss Asphalt)
- Fire effect on a reinforced concrete slab
- Evaluation of residual stress and integrity of welded part

Special Analysis (Contact, Geotechnical Analysis and etc.)

- Crack and fatigue analysis of the surface of structures
- Soil structure interaction analysis considering soil properties

Framework



Developed based-on Task-oriented Design Paradigm

64-bit environment platform



New Paradigm in Advanced Structural Analysis



Data Exchange

Import (Geometry)

- AutoCAD
- Parasolid / ACIS / STEP
- IGES / Pro-E
- CATIA V4 / V5
- SolidWorks / Unigraphics
- Inventor Part / Inventor Assembly

Export (Geometry)

- Parasolid
- STL

Standards for CAD Data Exchange

- STEP (STandard for the Exchange of Product Model Data)
- IGES (Initial Graphics Exchange Specification)



Generated Mesh

Interface with other MIDAS Software



Frame→Solid Wizard



Geometry Modeling



Advanced Modeling functions support both top-down and bottom-up approaches in surface & solid Modeling.

Curve Modeling

Generation **Modification** • Line Shortest Path Line • Fillet / Chamfer • B-Spline • Arc Polyline Surface Intersection Trim / Extend • Circle Offset Curve Rectangle Merge / Break Polygon • Ellipse Extrude Point Intersect • Tunnel Profile Make Wire Arc Imported DXF Polyline Surface Intersection Line Tunnel **B-Spline** Profile Section Circle (Polyline+Tangent Arc)

Surface Modeling



Solid Modeling



Advanced Modeling



Advanced Modeling



Graphic Display - Geometry



New Paradigm in Advanced Structural Analysis



Mesh Generation



Various of methods for generating Reinforcements and Interface Elements are provided. (auto & manual)

Mesh Generation



Automatic Surface Meshing

	Regularity Uniformity	Boundary Sensitive	Orientation Insensitive	Sizing Control (< 1/2)	Internal Curve/Point
Loop Mesher	0	0	0	0	\triangle
Grid Mesher	0	0	\bigtriangleup	×	0
Delaunay Mesher	Δ	0	0	0	0



Automatic Surface Meshing

FEA provides a number of Modeling and meshing functions for non-manifold surface models.



Automatic Surface Meshing



Automatic Solid Meshing

- FEA's Tetra Mesher auto-generates tetrahedral solid mesh with variable sizes in smooth transition. (200,000 Tetra's/min)
- FEA's Tetra Mesher is capable of including holes, curves and points that are present in/on solids.



Mapped Mesh Generation

FEA's Map Mesher generates structured (regular & orthogonal) mesh both on surfaces and in solids.



Mapped Solid Meshing

FEA's Solid Map Mesher generates hexa and/or penta mesh in simple solids by full mapping or combination (auto+map).



Hexa Dominant Mesh

Transitions from a tetrahedron element to a hexahedron element through a pyramid element



Size Control

FEA provides various size control methods and adaptive seeding function based on userspecified mesh size and geometric characteristics.



Mesh Protrusion



Mesh Protrusion



Element Libraries

Category		Elements	Order
	10	Truss	1 st , 2 nd
	טו	Beam	1 st , 2 nd
	2D	Plane Stress (Qaud / Tria)	1 st , 2 nd
Structural		Plane Strain (Quad / Tria)	1 st , 2 nd
Structural		Axisymmetry (Quad / Tria)	1 st , 2 nd
		Plate (Quad / Tria)	1 st , 2 nd
		Shell (Quad / Tria)	1 st , 2 nd
	3D	Hexa / Penta / Tetra	1 st , 2 nd
	Connection	Elastic Link	-
	Connection	Rigid Link	-
	Nodal	Point	-
Nonstructural	Mass	Matrix	-
		3D Point	-
	Interface	2D	1 st , 2 nd
		3D (Quad / Tria)	1 st , 2 nd
Painforcement	Embedded Trus	1 st , 2 nd	
Kennorcement	Tendon	1 st	
Heat Transfer	1D, 2D, 3D, Coo	1 st , 2 nd	

Reinforcement Elements



Embedded Interface Element

Embedded Interface Element

- Auto-generation of embedded line interface element from the line element embedded in a solid
- Auto-generation of embedded point interface elements at each end of the embedded line interface element.



Check & Quality Assurance

Check & Verify

- Feature Edges
- Free Faces
- Non-manifold Edges
- Clamped Element
- Overlapped Element(2D)
- Penetrated Element(2D)



- Aspect Ratio
- Skew Angle
- Taper
- Warpage
- Jacobian Ratio
- Twist
- Length




Graphic Display - Mesh



Flying View









New Paradigm in Advanced Structural Analysis



Support and operation for large-scale models



New framework for 64-bit and improvement in functions through new graphics engine

Support and operation for large-scale models

Bi-section method

- \rightarrow Automatic load step
- → Stable stress convergence regardless of the magnitude of applied load

Consistent tangent matrix

 \rightarrow Faster convergence speed of nonlinear analysis





Linear Static Analysis

Linear Static Analysis

- Multiple Load Cases
- Result Combination and Transformation

Equation Solvers

- Direct Solvers
 - Multi-frontal Sparse Gaussian Solver
 - Dense
 - AMG

Offshore Platform / Steel Frame Composed of Cylindrical Jackets (Plate + Frame)





Eigenvalue Analysis

Modal Analysis

Lanczos Method

Linear Buckling Analysis

- Critical Buckling Modes
- Buckling Modes
- Load Combination



4th Mode (270.00 Hz)

5th Mode (440.58 Hz)

Dynamic Analysis

Transient Response Analysis

- Direct Transient Response
- Modal Transient Response
- Time Forcing Function DB
 (54 Earthquake Acceleration Records)

Spectrum Response Analysis

- SRSS, CQC, ABS, NRL, TENP
- Design Spectrum DB





Material Nonlinearity Analysis

Material Models

- von Mises
- Tresca
- Mohr-Coulomb
- Drucker-Prager
- Rankine
- Hoek Brown
- Hyperbolic
- Strain Softening
- Cam Clay
- Modified Cam Clay
- Jardine
- D-min
- Soft Soil
- Concrete Smeared Crack
- Masonry
- Hardening Soil
- User-Supplied Material

Nonlinear Behaviours

- Hardening (Isotropic)
- Softening

Iteration Methods

- Full Newton-Raphson (Auto Load Step)
- Modified Newton-Raphson
- Arc-Length Method
- Initial Stiffness
- Quasi-Newton(Secant)



<u>Pinched Cylinder (Plate) – von Mises</u> Material & Geometry Nonlinear Analysis

User-Supplied Materials

In FEA NX, users can use their own defined material models via Fortran-coded library file.

IELEMEN

INTPT

SUBROUTINE USRMAT(EPS0, DEPS, NS, IELEMEN, INTPT, COORD, SE, ITER, USRVAL, NUV, USRSTA, NUS, IUSRIND, NUI, SIG, STIFF, ID)

USER SUPPLIED MATERIAL SUBROUTINE

!DEC\$ ATTRIBUTES DLLEXPORT::USRMAT

INTENT(IN

IMPLICIT NONE

INTEGER, INTENT(IN)

INTEGER, INTENT(IN)

INTEGER,

- FEA NX's user-supplied material model supports nonlinear elastic and elasto-plastic behaviours.
- User-supplied material can be used seamlessly with all elements which allow material nonlinear behaviours.



Geometry Nonlinearity Analysis

Iteration Methods

- Full Newton-Raphson (Auto Load Step)
- Modified Newton-Raphson
- Arc-Length Method
- Initial Stiffness
- Quasi-Newton(Secant)



Ring (Solid) – Total Lagrangian

Interface Nonlinearity Analysis

Interface Models

- Coulomb Friction
- Discrete Cracking
- Bond-Slip
- Nonlinear Elastic
- Combined (Cracking-Shearing-Crushing)





Deformation (Discontinuity btwn Steel & Concrete)



Principal Stress (Virtually Transformed & Clipped View)

Auto Contact

Contact Type

- Welded
- General
- Bi-directional Sliding Contact
- Rough
- Breaking-Weld





[Automatic search for free faces and endowment of conditions for rigid contact]

Reinforcement Analysis

Reinforcement

- Embedded Truss/Beam
- Truss + Interface (Slip/Friction)





Deformation



Heat Transfer and Heat of Hydration Analysis

Heat Transfer

- •Steady-State & Transient
- •Conduction, Convection,
- •Cooling Pipe
- •Heat Flux
- •Heat Flow
- •Temperature Gradient Display



Heat Transfer and Heat of Hydration Analysis



Static Contact Analysis







Contact Force



Hanger of Arch Bridge (Lug-Pin Joint)

Deformation and Stress Distribution by Contact



Cracking Analysis (1)

Cracking Models

- Smeared
- Crack Index

Results

- Crack Pattern (Crack Stress/Strain)
- Element Status
 - Cracking: Partially/Fully Open, Closed, Not Yet
 - Plasticity: Previously Plastic, Elastic, Plastic, Critical
 - Contact: No Contact, Slip, Stick



Cracking Analysis (2)

- Crack width calculation in the nonlinear analysis of reinforced concrete elements
 - In concrete Smeard Crack model, the crack width can be determined as the product of the crack-band width (h) and the difference of crack direction strain and crack direction stress divided by the original Young's modulus.



Load & B.C.

Loadings

- Self Weight
- Force / Moment
- Displacement
- Pressure
- Beam Load
- Prestress
- Temperature
- Initial Equilibrium Force

Boundary Conditions

- Constraint
- Constraint Equation
- Contact Conditions
- Convection



- Pipe Cooling
- Time Forcing Function
- Time Varying Load
- Ground Acceleration
- Response Spectrum Function



Constraint based-on CSys.



Spatially Varying Pressure (Function Applied)

Arbitrary Loading

FEA provides arbitrary loading function which can be applied to arbitrary locations/areas regardless of node and/or element connection.



Geotechnical analysis



New Paradigm in Advanced Structural Analysis



Post-processing

Complete Support for Visualisation and Interpretation

- Flexible User-control on Legends, Colors, Fonts, Magnification, etc.
- Multiple Plots, Graphs and Tables in Multiple Windows
- Deformed Shape Combined with Undeformed Shape (including Mode Shape)
- Local Plots defined by Geometrical Topology or User-selection
- Contour Plots and Animations (AVI)
- Iso-value Lines (2D) and Surfaces (3D)
- Clipping Planes and Slice Lines/Planes
- Partitioned Plots
- History Plots in Various Graphs and Animations (AVI)
- Result Values in MS-Excel compatible Tables
- Result Probe and Extraction
- Result Extraction for Construction Stage Analysis and Time History Analysis
- Screen-shots in JPG, BMP, PNG, GIF Picture Formats

Post-processing



Result Table

Post-processing



Printable PDF file that includes an adjustable 3D view and a decibel unit transformation feature for

 $d0 = 10^{-12} m$

 $L_{d} = 20 \log 10 (d/d_{0}) dB$

measuring noises

www.MidasUser.com

Dis. level

Contour Plot Type



Deformed Shape



Iso-surface Plot



Slice Plot



Clipping Plot



Diagram Plot



Vector Plot



Result Extraction



On-Curve Diagrams

	On-Curve Diagram	
	Name Diagram-1	
	Cutting Diagram Mode	
	Define Positions	
	Type 2-Points Line 🔻	
	Point 1 0, 0, 0	
	Point 2 1, 0, 0	
	Direction (+)X Dir. V Division 20	
	Reverse	
	🐺 🖉 OK Cancel Apply	
		3D On-Curve Graphs on Contour Plot
	Fault Zone	
Front View		2D On-Curve Graphs on Contour Plo

Probe & Result Tag


Update geometry using buckling mode shape

Create a new model file in which the geometry of analysis model is revised based on the mode shape of linear buckling analysis for geometric and material nonlinear analysis to find a buckling load considering geometric imperfection.

	Update Model with Imperfections Analysis Case Buckling analysis Buckling Model Eigenvalue(Required):MODE Scaling	ing mode to revise the etry of model.
	Scale Factor Maximum Value I m Silo,fea Update Cancel	
		New analysis model in which nodal coordinates are updated based on the selected buckling
Buckling mode shape		mode snape and scale factor.

•Geometric imperfection can easily be included in the analysis model. The buckling behavior of the model can be further investigated by performing geometric and material nonlinear analysis.



www.MidasUser.com





.

Double Pylon

www.MidasUser.com



www.MidasUser.com

Solid elements Modeling of Pylon Cable Saddle (Leesoonshin Suspension Br.)



Solid elements Modeling of Splay Saddle (New Millennium Suspension Br.)



Solid elements Modeling of Hanger Clamp (Bonny River Suspension Br.)



Applications Contact Analysis



www.MidasUser.com



FEA

• 32-bit pre/post handicapped for big models
 • 10 year old GUI platform (old graphics)
 → don't provide Preview for LBC
 → model is too big and analysis is not

 low computing power in geometric operations for large models and assemblies (Boolean, Division, etc.)

 \rightarrow I cannot divide solids

 limitation to addition/improvement for analysis features of the solver (DIANA Solver)
 → I can do that with Plaxis FEA NX

- Complete support for 64 bits (Pre/Post, Solver)
- Graphics engine supporting large models
 - Unified solver with GPU computing
 - Adoption of Parasolid Kernel (General kernel used by 3D CAD)
 Reliable geometric computation
 - performance/speed
 - Compatible with 3D CAD geometry
 - New Solver fully developed by MIDAS
 - Use of advanced features used in mechanical area (Contacts)
 - New platform for deploying advanced analysis and features

MIDAS

Major Improvements in Pre/Post Process

Improvements in Boolean operations robustness and performance/speed



Faster auto-meshing using multi-thread parallel processing

64 Solids	307 Solids	Mesh Generation in Progress					
		1 2 3 4 5 6 7 8 9 9 10 11	ID 1 2 3 4 5 6 7 8 9 10 11 12	Name O Box Geor Box Box Box Box	Current Step End metry-Mesh Relation End Meshing Solid Start Start Start Start Start Start Start Start Start Start Start Start Start	Progress ▲ 100 % 99 % 99 % 0 % 0 % 0 % 0 % 0 % 0 % 0 % 0 % 0 % 0 % 0 % 0 % 0 % 0 % 0 % 0 % 0 %	
FEA	A REAL PROPERTY OF A REAL PROPER	13 13 Box Start 0 % 14 14 Box Start 0 % Total Progress 0 % 0 % Image: Constraint of the start Update Database Image: Constraint of the start 0 % Image: Constraint of the start Support multi-thread during mesh generation Image: Constraint of the start Image: Constraint of the start					
		Version		64 Solids		307 Solids	;
FEA NX		FEA	(C	326.5 sec	%)	58.4 sec (2 solids failed	d) 90%
		FEA NX	(Co	21.7 sec)%)	9.1 sec (Completed 100)%)
					-		reduce

Tetrahedral auto-meshing generation time is reduced by over 80% (Multi-Thread)



Construction stage sets & auto-generation of construction stages

- Definition/Analysis of **multiple construction stage sets** (can be defined for each analysis type)
- Graphical auto-definition of construction stages by Show/Hide status in the work window



Manage model file, Avoid dual-tasks and Prevent confusion and analysis errors

Additional post-processing features and performance improvements

- Improve the overall speed of processing operations (stress contour of a practical test model: 20.4 sec → reduced to 1.1 sec)
 - 3D dynamic PDF report in which 3D model view can be manipulated
 - Real-time 3D flow path for seepage analysis (animation)
 - Element Contour Plot (simultaneous display of results from different element types)
 - Improvement in results computation (combination of nodal/element results from different stages/analysis cases, conversion of dynamic analysis results to DB, etc.)



Addition of modelling features

- Auto-generation of **co-face** between adjacent solids
- Imprint automatically in faces by selecting edges penetrating solids



Easy enough for beginners to avoid modeling errors from auto-correction



GUI designed for ease of use for beginners

- Simplified menu structure and efficient use of **Ribbon menu/Tapped dialogue box**
 - Intuitive functions/options usage with instructional guide images
 - Mouse motions minimised with enhanced context menu in the work window



Major Improvements in Analysis Features

High quality mesh (hybrid mesh) and bonded contacts

- Addition of **Hybrid mesh** for securing accurate results
- Bonded contacts: Auto-searching adjacent elements and auto-processing **bonded** contacts (for 3D analysis), Continuity – Compatibility Guaranteed



Hexa elem. for accuracy at the boundaries, partial tetra elem. inside the model

Hybrid mesh, bonded contact results comparison



Hybrid mesh + Contact vs. Coincident Nodes



Hybrid mesh, bonded contact results comparison



Hybrid mesh + Contact vs. Coincident Nodes



Comparison of linear-nonlinear time history dynamic analysis

• Areas using Nonlinear Dynamic Analysis: Soil-Structure Interaction Behaviour, Evaluation of embankment stability under dynamic loading, Crash loads, Seismic loads due to structural response assessment, Liquefaction evaluation (user-defined model can be implemented)





