Overview

Geometry Modeling 15
Mesh Generation 23
Analysis 40
Post-processing 64
FEA is a state-of-the-art integrated finite element analysis system for nonlinear and detail simulations of civil and building structures...
Overview

Foundation

Advanced Nonlinear and Detail Analysis Program

midas FX+
Modeling, Meshing & Post-processing

MIDAS Solver
Linear, Nonlinear (Material/Geometry)
Contact, Heat Transfer, Fatigue

Co-Dev. with TNO DIANA
Crack, Reinforcement, Interface

midas FEA
“Integrated Solution for Advanced Analysis in Civil Structural CAE”
### Analysis Overview

#### Static Analysis
- Construction Stage Analysis
- Moving Load Analysis

#### Modal Analysis
- Linear Buckling Analysis
- Transient / Frequency / Response Spectrum Analysis

#### Material / Geometry Nonlinearity Analysis
- Interface Nonlinearity Analysis
- Reinforcement Analysis
- Cracking Analysis
- Static/Explicit Contact Analysis

#### Heat Transfer Analysis
- Fatigue Analysis
- Fluid Dynamics Analysis
Overview

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, Interface and Reinforcement Nonlinear Analysis

- Detail analysis of composite structure (steel + concrete)
  Thermal analysis and differential shrinkage analysis of steel-concrete composite girder, concrete filled steel tube and core of the SRC pier and so on
- 3D detail analysis considering steel, concrete and reinforcement simultaneously
- Detail analysis of CFT and analysis of the long-term behavior (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

**Overview**

**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 (de-bonding 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 (CFD, Contact, Fatigue, etc.)**

- Crack and fatigue analysis of the surface of structures
- Damage estimation of pier/waterbreak impacted by ship
- Life-cycle prediction of steel-box bridges based-on moving load analysis
- Fluid dynamics analysis of bridges, high-rise buildings and tunnels
- Semi-coupled fluid-structure interaction analysis
- Direct analysis of soil-structure interaction
- High-end detail analysis (crash, fatigue, fracture mechanism)
Overview

Configuration

Advanced Nonlinear and Detail Analysis Program

Overview

Geometry Modeler

Report Generator

Mesh Generator

Post-processor

FEA

FEM Pre-processor

FEM Solver
Overview

Framework

Advanced Nonlinear and Detail Analysis Program

Developed based-on Task-oriented Design Paradigm
Overview

Graphic Display - Geometry

Shading with Edge

Wireframe

Transparency

Shading + Transparency
Overview

Graphic Display - Mesh

- Shrink
- Wireframe (Free-Face)
- Shading
- Feature-Edge
Virtual Mesh Transformation

Virtual Transformation (Translation, Rotation, Scaling) by Mouse Dragging
Overview

Flying View

Advanced Nonlinear and Detail Analysis Program
Overview
Data Exchange

**Geometry Model Data**

- **Import (Geometry)**
  - STEP, IGES
  - ACIS*, Parasolid*
  - SolidWorks*, Inventor*, etc.
  - AutoCAD DWG / DXF
  -> ‘*’ marked CAD interfaces are options.

- **Export (Geometry)**
  - STEP, IGES

**Analysis Model Data**

- **Import (Analysis Data)**
  - DIANA, MSC/NASTRAN
  - Neutral (Text)

- **Export (Analysis Data)**
  - MIDAS/Civil, MIDAS/Gen
  - Neutral (Text)

**Standards for CAD Data Exchange**

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

Overview ................................................................. 02
Geometry Modeling ............................................... 15
Mesh Generation .................................................. 23
Analysis ................................................................. 40
Post-processing ..................................................... 64
Advanced modeling functions support both top-down and bottom-up approaches in surface & solid modeling.
### Geometry Modeling

#### Curve Modeling

**Generation**

- Line
- Arc
- Circle
- Ellipse
- Parabola
- Hyperbola
- B-Spline
- Polyline
- Rectangle
- Polygon
- Profile
- Tunnel
- On-Surface Curve
- Shortest Path Line
- Surface Intersection
- Offset Curve
- Extrude Vertex
- Tangent Line

**Modification**

- Fillet / Chamfer
- Trim / Extend
- Merge / Break
- Intersect
- Align, Coincide
- Make Wire

---

**Arc**

- Polyline
- Line
- Circle (Polyline+Tangent Arc)

**Surface Intersection**

- Tunnel Section

**Imported DXF**

- B-Spline
Geometry Modeling

Surface Modeling

Advanced Nonlinear and Detail Analysis Program

- Co-planar Curves
- Plane Patch
- Virtual Grid (M X N)
- Grid Patch
- Elevation
- Vertex Cloud
- Vertex Patch
- 2~4 Curves
- Coons Patch
- Arbitrary Curves (Boundary/Tangent/Internal)
- NURBS Patch
Solid Modeling

Advanced Nonlinear and Detail Analysis Program

Geometry Modeling

Trim

Divide

Stitch to Solid (Face → Solid)

Boolean Operation

Fuse (A ∪ B)

Cut (A - B)

Embed (A - B) + (A ∩ B)

Boolean Common (A ∩ B) operation is also provided.
Advanced Nonlinear and Detail Analysis Program

Geometry Modeling

Advanced Modeling

- Trim Surfaces
- Offset
- Chamfer
- Fillet
- Local Prism (Fuse: Defined Height)
- Shell
Frame→Solid Wizard automatically generates Solid Geometry & Mesh by importing Frame Model (*.MCS) from Civil and Gen.
Mesh Generation
Various of methods for generating **Reinforcements** and **Interface Elements** are provided. (auto & manual)
Mesh Generation

2D Quadrilateral Mesh

3D Hexahedral Mesh

1D Linear Mesh

3D Tetrahedral Mesh
## Automatic Surface Meshing

<table>
<thead>
<tr>
<th></th>
<th>Regularity Uniformity</th>
<th>Boundary Sensitive</th>
<th>Orientation Insensitive</th>
<th>Sizing Control (&lt; 1/2)</th>
<th>Internal Curve/Point</th>
</tr>
</thead>
<tbody>
<tr>
<td>Loop Mesher</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Grid Mesher</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Delaunay Mesher</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
</tbody>
</table>

**Mesh Generation**

**Loop Mesher (Full Quad)**

**Grid Mesher (Quad+Tria)**
FEA provides a number of modeling and meshing functions for non-manifold surface models.
- FEA provides **automatic defining & meshing function** which defines mesh-able domains from curves (without creating surfaces) and then generates meshes for each domain.

- **Automatic defining & meshing function** is very useful for **complex 2D models** which were originally modeled in **AutoCAD**.
- 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.
FEA’s **Map Mesher** generates **structured (regular & orthogonal) mesh** both in surfaces and solids.
FEA's Solid Map Mesher generates hexa and/or penta mesh in simple solids by full mapping or combination (auto+map).
- FEA is under implementation of **H-Morph Meshing** to generate **Hexa-dominant mesh**.
- **H-Morph** is a method to generate **boundary conforming, hexa-dominant mesh** for arbitrary solid geometries. (FEA uses **Q-Morph** and **H-Morph** algorithms proposed by S.Owen.)
- FEA will also provide **Prism Layer Meshing** function. (Outer:Prism – Inner:Tetra)

S.Owen (1999)  
H-Morphing Procedure (Tetra→Hexa)
- FEA is under implementation of **Sub-mapped Meshing** functions for **pseudo-Cartesian geometries**.
- FEA adopts **Volume Sub-mapping** algorithms proposed by D.White and M.Whiteley.
- **Volume Sub-mapping** is enhanced 3D mapping technique which sub-divides geometry into volume mappable sub-regions.

Pseudo-Cartesian shapes have interior and exterior angles that are close to $\pi/2$. 

David R. White (1996)
FEA provides various size control methods and **adaptive seeding function** based on **user-specified mesh size** and **geometric characteristics**.
Mesh Generation

Mesh Protrusion

**Extrude** (2D→3D)

**Fill** (Curve→2D)

**Section**

**Simulate**

**Extrude** thru Node Sequence (Curve→2D)

**Revolve** (2D→3D)
Mesh Protrusion

- **Project (Curve → 3D)**
- **Fill (2D → 3D)**
  - Top
  - Bottom
  - Same Topology
- **Sweep (2D → 3D)**
  - Guide Curve
  - Scaled Sweep
- **Offset (2D → 3D)**
**Mesh Generation**

**Interface Elements**

- **Generation Method**
  - Select Nodes
  - Input Node IDs
  - Extract from Element Boundary
  - Extract from Free-Faces
  - Insert Both Sides of Beam/Plate
  - Convert Elements

Select Nodes
Input Node IDs

Extract from Element Boundary
Extract from Free-Faces

Insert Both Sides of Beam/Plate
Mesh Generation

Reinforcement Elements

- **Modeling Method**
  - Embedded Bar (In-compatible Mesh)
  - Truss (Compatible Mesh) + Interface (Slip, Friction)
Check & Quality Assurance

Check & Verify
- Free Edges
- Free Faces
- Manifold Edges
- Non-manifold Edges
- Check & Align ECS

Quality Assurance
- Aspect Ratio
- Skew Angle
- Taper (2D)
- Warpage (2D)
- Jacobian Ratio
- Twist
- Collapse (Tetra)
- Length / Area

Check Free Face
(Unconnected Element Face)

Twisted Penta

Collapsed Tetra
(Near Zero Volume)

Mesh Quality Plot
Analysis

Overview 02
Geometry Modeling 15
Mesh Generation 23
Analysis 40
Post-processing 64
Analysis Overview

- Static Analysis
- Construction Stage Analysis
- Moving Load Analysis

- Modal Analysis
- Linear Buckling Analysis
- Transient / Frequency / Response Spectrum Analysis

- Material / Geometry Nonlinearity Analysis
- Interface Nonlinearity Analysis
- Reinforcement Analysis
- Cracking Analysis
- Static/Explicit Contact Analysis

- Heat Transfer Analysis
- Fatigue Analysis
- Fluid Dynamics Analysis
<table>
<thead>
<tr>
<th>Category</th>
<th>Elements</th>
<th>Order</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td>Structural</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1D</td>
<td>Truss (Gap / Hook / Cable)</td>
<td>1st</td>
<td>Total Lagrangian</td>
</tr>
<tr>
<td></td>
<td>Beam</td>
<td>1st</td>
<td>Total Lagrangian</td>
</tr>
<tr>
<td>2D</td>
<td>Plane Stress (Quad / Tria)</td>
<td>1st, 2nd</td>
<td>Total/Updated Lagrangian</td>
</tr>
<tr>
<td></td>
<td>Plane Strain (Quad / Tria)</td>
<td>1st, 2nd</td>
<td>Total/Updated Lagrangian</td>
</tr>
<tr>
<td></td>
<td>Axisymmetry (Quad / Tria)</td>
<td>1st, 2nd</td>
<td>Total/Updated Lagrangian</td>
</tr>
<tr>
<td></td>
<td>Plate (Quad / Tria)</td>
<td>1st, 2nd</td>
<td>Total/Updated Lagrangian</td>
</tr>
<tr>
<td></td>
<td>Shell (Quad / Tria)</td>
<td>1st, 2nd</td>
<td>Total/Updated Lagrangian</td>
</tr>
<tr>
<td>3D</td>
<td>Brick / Wedge / Tetra</td>
<td>1st, 2nd</td>
<td>Total/Updated Lagrangian</td>
</tr>
<tr>
<td>Nonstructural</td>
<td>Connection</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>General Link</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>Rigid Link</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>Mass Spring</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>Point</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>Matrix</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Interface</td>
<td>3D Point</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>2D</td>
<td>1st, 2nd</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>3D (Quad / Tria)</td>
<td>1st, 2nd</td>
<td>-</td>
</tr>
<tr>
<td>Reinforcement</td>
<td>Embedded Bar</td>
<td>1st, 2nd</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>Embedded Grid (Quad / Tria)</td>
<td>1st, 2nd</td>
<td>-</td>
</tr>
<tr>
<td>Heat Transfer</td>
<td>1D, 2D, 3D, Cooling Pipe, Thermal Link</td>
<td>1st, 2nd</td>
<td>-</td>
</tr>
</tbody>
</table>
Load & B.C.

- **Loadings**
  - Body Force
  - Force / Moment
  - Mass
  - Displacement
  - Pressure
  - Beam Load
  - Pre-stress
  - Temperature

- **Boundary Conditions**
  - Constraint
  - Multi-Point Constraint
  - Contact Conditions
  - Convection
  - Radiation

- **Velocity / Acceleration**
  - Heat Generation
  - Heat Flux
  - Time Forcing Function
  - Time Varying Load
  - Ground Acceleration
  - Response Spectrum Function
FEA provides **arbitrary loading** function which can be applied to **arbitrary locations/areas regardless of node and/or element connection**.
**Linear Static Analysis**
- Multiple Load Cases
- Result Combination and Transformation

**Equation Solvers**
- **Direct Solvers**
  - Multi-frontal Sparse Gaussian Solver (Default)
  - Skyline Solver
- **Iterative Solvers**
  - Preconditioned Conjugate Gradient
  - Generalized Minimal Residual

### Table: Net Solution Times (Pentium IV 3GHz, 1GB RAM)

<table>
<thead>
<tr>
<th>Element Type</th>
<th>Model A</th>
<th>Model B</th>
<th>Model C</th>
<th>Model D</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of Elements</td>
<td>30,000</td>
<td>30,000</td>
<td>125,000</td>
<td>40,000</td>
</tr>
<tr>
<td>No. of DOF's</td>
<td>180,600</td>
<td>181,800</td>
<td>390,150</td>
<td>132,300</td>
</tr>
<tr>
<td>Solution Time [sec]</td>
<td>Multi-frontal</td>
<td>35</td>
<td>41</td>
<td>3,244</td>
</tr>
<tr>
<td></td>
<td>PCG</td>
<td>179</td>
<td>188</td>
<td>817</td>
</tr>
</tbody>
</table>

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

*Stress Distribution of Jacket*
## Analysis

### Eigenvalue Analysis

- **Modal Analysis**
  - Lanczos Method
  - Subspace Iteration
  - Ritz Vector

- **Linear Buckling Analysis**
  - Critical Buckling Modes
  - Buckling Modes
  - Load Combination

---

**Simply Supported Stiffened Plate (Plate + Beam)**

- **1st Mode (64.58 Hz)**
- **2nd Mode (106.05 Hz)**
- **3rd Mode (208.96 Hz)**
- **4th Mode (270.00 Hz)**
- **5th Mode (440.58 Hz)**
- **Transient Response Analysis**
  - Direct Transient Response
  - Modal Transient Response
  - Time Forcing Function DB
    (54 Earthquake Acceleration Records)
  - Nonlinear Analysis
  - Boundary Nonlinear Analysis
    (Damper, Viscous Boundary, etc.)

- **Frequency Response Analysis**
  - Direct Frequency Response
  - Modal Frequency Response
  - Frequency-dependent Excitation

- **Spectrum Response Analysis**
  - SRSS, CQC, ABS
  - Design Spectrum DB
**Material Models**
- von Mises
- Tresca
- Mohr-Coulomb
- Drucker-Prager
- Rankine
- User-Supplied Material

**Nonlinear Behaviors**
- Hardening (Iso/Kinematic/Mixed)
- Softening

**Iteration Method**
- Full Newton-Raphson
- Modified Newton-Raphson
- Arc-Length Method
- Constant Stiffness
- Displacement Control

*Tendon Anchorage (Solid) – von Mises*

*Pinched Cylinder (Plate) – von Mises Material & Geometry Nonlinear Analysis*
In FEA, users can use their own defined material models via Fortran-coded library file.

- FEA’s user-supplied material model supports nonlinear elastic and elasto-plastic behaviors.

- User-supplied material can be used seamlessly with all elements which allow material nonlinear behaviors.

**User-Supplied Materials**

<table>
<thead>
<tr>
<th>Model Type</th>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>User-supplied material</td>
<td>USRMAT.DLL</td>
</tr>
</tbody>
</table>

**USM Dialog**

- User-Supplied Material Library File: C:\FEA Works\USRMAT.dll
- Number of Parameters (NLV): 0
- Number of Internal State Variable (NSV): 0
- Number of Integer Indicator Variable (NIIV): 0

**Add/Modify User-defined value**

- User-defined Parameters:
  - Parameter: 1, Value: 0.0000
  - Parameter: 2

**Input**

- User-defined Parameters Input Dialog

**Output**

- Strain
- Total Stress
- Stiffness Matrix

**USRMAT.DLL**

```fortran
! USER SUPPLIED MATERIAL SUBROUTINE

SUBROUTINE USRMAT (EPS0, EPS, NS, IELEM, INTIP, COORD, &
  & UE, ITER USPFL, MUX, USRMAT, MUX, USRMAT, &
  & NUX, SIG, STIFF, ID)

  ! ... INPUT DATA READ...!

  REAL(I), INTENT(IN) :: NS
  INTEGER, INTENT(IN) :: IELEM
  INTEGER, INTENT(IN) :: INTIP
  INTEGER, INTENT(IN) :: ID
  INTEGER, INTENT(IN) :: NUX
  INTEGER, INTENT(IN) :: NUI
  REAL(I), INTENT(IN) :: EPS0(NS)
  REAL(I), INTENT(IN) :: EPS(NS)
  REAL(I), INTENT(IN) :: COORD(I)
  REAL(I), INTENT(IN) :: SIG(NS)
  REAL(I), INTENT(IN) :: STIFF(NS, NS)
  REAL(I), INTENT(IN) :: USRMAT(NUX)
  REAL(I), INTENT(IN) :: USRMAT(NUI)

  INTEGER :: I, J
  REAL(I) :: EMOD, EPS(NS)

  SIG(1:NS) = 0.0D0
  EMOD = USRVAL(1)
  EPS(NS) = EPS0(1:NS) + EPS(1:NS)

  ! TOTAL STRESS

  SIG(1) = EMOD * EPS(1) + 1000.0D0 * EMOD * EPS(1)**2
  SIG(2) = EMOD * EPS(2) + 1000.0D0 * EMOD * EPS(2)**2
  SIG(3) = EMOD * EPS(3) + 1000.0D0 * EMOD * EPS(3)**2
  SIG(4) = EMOD * EPS(4) / 2.0D0
  SIG(5) = EMOD * EPS(5) / 2.0D0
  SIG(6) = EMOD * EPS(6) / 2.0D0

  ! MATERIAL STIFFNESS MATRIX

STIFF(1,1) = EMOD + 2000.0D0 * EMOD * EPS(1)
STIFF(2,2) = EMOD + 2000.0D0 * EMOD * EPS(2)
STIFF(3,3) = EMOD + 2000.0D0 * EMOD * EPS(3)
STIFF(4,4) = 0.0D0 * EMOD
STIFF(5,5) = 0.0D0 * EMOD
STIFF(6,6) = 0.0D0 * EMOD

RETURN
END
```

**<Ex> Nonlinear Elastic Material for Solid Element**
**Methods**
- Updated Lagrangian
- Total Lagrangian
- Co-rotational

**Iteration Method**
- Full Newton-Raphson
- Modified Newton-Raphson
- Arc-Length Method
- Constant Stiffness
- Displacement Control

*Rectangular Tube (Plate) – Co-rotational*

*Ring (Solid) – Total Lagrangian*
Interface Nonlinearity Analysis

- **Interface Models**
  - Coulomb Friction
  - Discrete Cracking
  - Crack Dilatancy
  - Bond-Slip
  - Combined (Cracking-Shearing-Crushing)

**Deformation** (Discontinuity between Steel & Concrete)

**Principal Stress** (Virtually Transformed & Clipped View)
Reinforcement Analysis

- **Reinforcements**
  - Embedded Bar/Grid (Bonded/Unbonded)
  - Truss + Interface (Slip/Friction)

*2-Span Double-T Type Prestressed Concrete Girder*

*Stress of Embedded Reinforcements*

*Deformation*

*Maximum Principal Stress of Concrete with Deformation*
Cracking Analysis

- **Cracking Models**
  - Total Strain Crack
  - 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

- Symbols at Gauss Points
  - **Disc Normal**: Opening Direction
  - **Disc Color**: Magnitude
  - **Line**: Shearing Direction

*Steel Reinforced Concrete Bracket*

*Crack Pattern (Disc Plot)*
Heat of Hydration Analysis

- **Visco-Elastic Models**
  - Kelvin
  - Maxwell
  - Creep-Shrinkage (Design Code)
  - Temperature-Dependent Material

- **Heat Transfer**
  - Steady-State
  - Transient
  - Conduction, Convection, Radiation
  - Pipe Cooling

*Pier Table (Construction Stage) - Temperature*

*Pier Table (Construction Stage) - Stress*
**Fatigue Analysis (Wizard)**

**Advanced Nonlinear and Detail Analysis Program**

- **Methods and Parameters**
  - S-N Method (Stress-Life)
  - E-N Method (Strain-Life)
  - Load / Stress History
  - Rainflow Counting
  - Mean Stress Corrections
  - Stress Concentration Factor
  - Modifying Factors

- **Calculation Objects**
  - Boundary Nodes Only (Default)
  - Nodes of Selected Mesh Sets

- **Results**
  - Cycles to Failure
  - Safety Factor
    (Cycles to Failure / Desired Repetition)

---

**S-N Curves of Fatigue Design Codes**

- **JSSC**
  - $1\text{MPa} = 10.1972\text{kgf/cm}^2$
  - $1\text{ksi} = 70.3081\text{kgf/cm}^2$

- **ASSHTO**
  - $1\text{MPa} = 10.1972\text{kgf/cm}^2$

---

**Contour Plot of Cycles**
## Verification of Element Formulation (In Development Stage)

<table>
<thead>
<tr>
<th>Step</th>
<th>Test Type</th>
<th>Verification Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Single Element Test</td>
<td>Verify Stiffness Matrix (Simplest Check)</td>
</tr>
<tr>
<td>2</td>
<td>Patch Test</td>
<td>Verify Stability (Element Shape &amp; Configuration)</td>
</tr>
<tr>
<td>3</td>
<td>Refined Mesh Test</td>
<td>Verify Convergence (Mesh Division vs. Stress)</td>
</tr>
<tr>
<td>4</td>
<td>Eigenvalue Test</td>
<td>Verify Mass Matrix (using Lumped Mass)</td>
</tr>
<tr>
<td>5</td>
<td>Benchmark Test</td>
<td>NAFEMS, NASTRAN, DIANA, ABAQUS, etc.</td>
</tr>
</tbody>
</table>

## System Test (After Development Stage)

<table>
<thead>
<tr>
<th>Step</th>
<th>Test Type</th>
<th>Verification Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>Coverage Analysis</td>
<td>Verify Full Coverage of Test Problems</td>
</tr>
<tr>
<td>7</td>
<td>Regression Test</td>
<td>Automated Test (Over 1,000 Problems Weekly)</td>
</tr>
</tbody>
</table>
**Single Element Test**

- Pure Compression & Shear
- Pure Bending
→ Constant Stress (OK)

**Patch Test**

- Dimension
  \[ L_x = L_y = L_z = 1 \]
- Pure Compression & Shear
  \[
  \begin{align*}
  U_x &= 10 - 3 \left( 2x + y + z \right) / 2 \\
  U_y &= 10 - 3 \left( x + 2y + z \right) / 2 \\
  U_z &= 10 - 3 \left( x + y + 2z \right) / 2
  \end{align*}
  \]
→ Constant Stress (OK)

- Dimension
  \[ L_x = 0.24, L_y = 0.12 \]
- <1> Membrane
  \[
  \begin{align*}
  U_x &= 10 - 3 \left( x + y \right) / 2 \\
  U_y &= 10 - 3 \left( y + x \right) / 2
  \end{align*}
  \]
- <2> Bending (Transverse)
  \[
  \begin{align*}
  U_z &= 10 - 3 \left( x^2 + xy + y^2 \right) / 2 \\
  R_x &= 10 - 3 \left( y + x / 2 \right) \\
  R_y &= -10 - 3 \left( x + y / 2 \right)
  \end{align*}
  \]
→ Constant Stress (OK)

All constant stress values are always checked and verified!
Pinched Cylindrical Shell

Symmetric Model

- Theoretical deflection is $4.5197 \times 10^{-4}$.
- FEA shows superior and monotonic convergence in various mesh divisions.
**Geometry Nonlinearity (Solid, T.L.)**

Nodal Force = 1, 2, 5

<table>
<thead>
<tr>
<th>Force</th>
<th>1</th>
<th>2</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hexa</td>
<td>0.01</td>
<td>0.01</td>
<td>0.01</td>
</tr>
<tr>
<td>Penta</td>
<td>0.02</td>
<td>0.02</td>
<td>0.02</td>
</tr>
<tr>
<td>Tetra</td>
<td>0.02</td>
<td>0.02</td>
<td>0.02</td>
</tr>
</tbody>
</table>

Displacement Relative Error Norm w.r.t. DIANA

**NAFEMS (CGS-3): Hertzian Contact**

Maximum Stress (E-ID: 4)

<table>
<thead>
<tr>
<th></th>
<th>ABAQUS</th>
<th>FEA</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>21,169</td>
<td>21,120</td>
</tr>
</tbody>
</table>

Nodal Force = 1, 2, 5

<table>
<thead>
<tr>
<th>Pressure</th>
<th>1</th>
<th>2</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hexa</td>
<td>0.01</td>
<td>0.02</td>
<td>0.02</td>
</tr>
<tr>
<td>Tetra</td>
<td>0.01</td>
<td>0.02</td>
<td>0.01</td>
</tr>
</tbody>
</table>

Displacement Relative Error Norm w.r.t. DIANA

**Analysis**

Benchmark Test
Counts the number of visits in execution to assure all code lines have been tested.

- Test problems are continuously augmented reflecting the result of coverage analysis.
- All test problems are automatically analyzed for regression prevention every week.

Manages and traces new features / bugs / supports with full revision history.
Post-processing

Overview 02
Geometry Modeling 15
Mesh Generation 23
Analysis 40
Post-processing 64
Complete Support for Visualization 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 WMF, BMP, PNG Picture Formats
- State-of-the-art Reports Generated by XML and HTML
Post-processing

Works Tree

Contour Plot

Result Graph

Result Table

MS-Excel

Advanced Nonlinear and Detail Analysis Program
Post-processing

Contour Plot Type

- Contour with Mesh
- Contour with Iso-line
- Contour with Mesh & Iso-line
- Gradient Contour
- 2-Color Contour
- Gray Contour
Post-processing

Deformed Shape

Deformed Contour with Original Shape
(Static Analysis)

Mode Shapes
(Stability Analysis)
Post-processing

Iso-surface Plot

Iso-Surface in Transparent Solid Geometry

Multiple Iso-surfaces

Base Iso-surface

Capped Plot (Lower Part)

Capped Plot (Upper Part)
Post-processing

Slice Plot

Original Plot (Solid)

Multiple Slice Planes

Slice Plot at Arbitrary Plane
Post-processing

Clipping Plot

Original Plot

½ Clipped Plot with Feature-Edge

Clipping Plane Definition by Mouse Dragging

½ Clipped Plot with Diagram on Middle Clipping Plane
Post-processing

Clipping Plot

Original Plot

Multiple Clipping Planes
Post-processing

Partition Plot

Analysis Model

Result (Partition Plot)

Geotechnical Model with Multiple Strata Configuration
Post-processing
Mirror Plot

Symmetry Plane

Symmetric Model

Mirrored Contour Plot

Mirrored Deformed Shape
Post-processing

Diagram Plot

Advanced Nonlinear and Detail Analysis Program

Solid Type

Section Plot of Frames

Diagram Plot with Deformed Shape

Line Type
Vector Plot Option
- Head Type (Both, One, None)
- Constant Head Size
- Constant Body Size
- Color (Contour, Mono)
**Result Extraction**

**Advanced Nonlinear and Detail Analysis Program**

**Post-processing**

- **Start Step / Time**
- **End Step / Time**
- **Step / Output Set**
- **Result Type**
- **Node / Element IDs**

Results can be extracted based on:
- Time (Transient Analysis)
- Step (Nonlinear / Construction Stage Analysis)
- Coordinate (User-defined Coordinate Sys.)

**MS-Excel compatible Table**
(Time & Result Value)

**Graph (Time .vs. Result Value)**

**Transient Seepage**
**Post-processing**

**On-Curve Diagrams**

**Advanced Nonlinear and Detail Analysis Program**

- **On-Curve Diagram**
  - Define Curve
    - 2-Point Line
      - 45, 75, 27

- **Diagram Direction**
  - (+) X-Dir
  - (+) Y-Dir
  - (+) Z-Dir
  - (-) X-Dir
  - (-) Y-Dir
  - (-) Z-Dir
  - 2-Point Vector
    - 0, 0, 0
    - 1, 0, 0

- **Curve Plot**
  - Name
  - Curve Plot
  - Division
  - Add
  - Modify
  - Delete

- **Result Data at User-Specified Sampling Points**

<table>
<thead>
<tr>
<th>ID</th>
<th>X (m)</th>
<th>Y (m)</th>
<th>Z (m)</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>102.13</td>
<td>34.75</td>
<td>132.00</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>100.09</td>
<td>34.75</td>
<td>132.00</td>
<td>0.0001</td>
</tr>
<tr>
<td>3</td>
<td>98.95</td>
<td>34.75</td>
<td>132.00</td>
<td>0.0001</td>
</tr>
<tr>
<td>4</td>
<td>97.61</td>
<td>34.75</td>
<td>132.00</td>
<td>0.0001</td>
</tr>
<tr>
<td>5</td>
<td>93.96</td>
<td>34.75</td>
<td>132.00</td>
<td>-0.0001</td>
</tr>
<tr>
<td>6</td>
<td>91.02</td>
<td>34.75</td>
<td>132.00</td>
<td>-0.0003</td>
</tr>
<tr>
<td>7</td>
<td>89.88</td>
<td>34.75</td>
<td>132.00</td>
<td>-0.0005</td>
</tr>
<tr>
<td>8</td>
<td>67.04</td>
<td>34.75</td>
<td>132.00</td>
<td>-0.0006</td>
</tr>
<tr>
<td>9</td>
<td>65.70</td>
<td>34.75</td>
<td>132.00</td>
<td>-0.0011</td>
</tr>
<tr>
<td>10</td>
<td>63.75</td>
<td>34.75</td>
<td>132.00</td>
<td>-0.0014</td>
</tr>
<tr>
<td>11</td>
<td>61.71</td>
<td>34.75</td>
<td>132.00</td>
<td>-0.0017</td>
</tr>
<tr>
<td>12</td>
<td>73.66</td>
<td>34.75</td>
<td>132.00</td>
<td>-0.0019</td>
</tr>
<tr>
<td>13</td>
<td>77.82</td>
<td>34.75</td>
<td>132.00</td>
<td>-0.0021</td>
</tr>
<tr>
<td>14</td>
<td>75.58</td>
<td>34.75</td>
<td>132.00</td>
<td>-0.0022</td>
</tr>
<tr>
<td>15</td>
<td>73.54</td>
<td>34.75</td>
<td>132.00</td>
<td>-0.0024</td>
</tr>
<tr>
<td>16</td>
<td>71.49</td>
<td>34.75</td>
<td>132.00</td>
<td>-0.0024</td>
</tr>
</tbody>
</table>

- **3D On-Curve Graphs on Contour Plot**

- **Fault Zone**

- **2D On-Curve Graphs on Contour Plot**

- **Front View**
Post-processing

Probe & Result Tag

Advanced Nonlinear and Detail Analysis Program

Nodal Result

Probe & Add Result Tags at Specified Nodes/Elements

Sectional Result with Clipped Plot (Element Result)
In FEA, legend can be controlled for its position, size, format and range (including min/max value) by mouse dragging.

- **Legend Option:**
  - Color (Value, Ratio, Description)
  - Logo
  - Range (including Min/Max/Zero)
  - Format (Fixed/Scientific, Width)

**Property Window (Legend Option)**

**Drag side to resize legend box**

**Drag scale bar to change range**
Thank You!

Advanced Nonlinear and Detail Analysis Program

midas FEA