Overview

Geometry Modeling
Mesh Generation
Analysis
Post-processing
FEA is a state-of-the-art integrated finite element analysis system for nonlinear and detail simulations of civil and building structures...
Advanced Nonlinear and Detail Analysis Program

**Overview**

**Foundation**

- **Modeling, Meshing & Post-processing**
- **MIDAS Solver**
  - Linear, Nonlinear (Material/Geometry)
  - Contact, Heat Transfer, Fatigue
- **Co-Dev. with TNO DIANA**
  - Crack, Reinforcement, Interface

**midas FX+**
Modeling, Meshing & Post-processing

**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
Overview
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 (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

- Geometry Modeler
- Mesh Generator
- FEM Pre-processor
- FEM Solver
- Post-processor
- Report Generator
Developed based-on **Task-oriented Design Paradigm**
Overview

Graphic Display - Geometry

- Shading with Edge
- Wireframe
- Transparency
- Shading + Transparency
Overview

Graphic Display - Mesh

Advanced Nonlinear and Detail Analysis Program

- Shrink
- Mesh Shading
- Feature - Edge
- Wireframe (Free-Face)
- Shading
- Feature - Edge
Overview

Virtual Mesh Transformation

Transformation Control Box

Virtual Transformation (Translation, Rotation, Scaling) by Mouse Dragging
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.
Advanced Nonlinear and Detail Analysis Program

Geometry Modeling
Curve Modeling

**Generation**
- Line
- Arc
- Circle
- Ellipse
- Parabola
- Hyperbola
- B-Spline
- Polyline
- Rectangle
- Polygon
- Profile
- Tunnel

**Modification**
- On-Surface Curve
- Shortest Path Line
- Surface Intersection
- Offset Curve
- Extrude Vertex
- Tangent Line
- Fillet / Chamfer
- Trim / Extend
- Merge / Break
- Intersect
- Align, Coincide
- Make Wire

[Images of geometric shapes and curves, illustrating generation and modification processes.]
Surface Modeling

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

Advanced Modeling

- Trim Surfaces
- Local Prism (Fuse: Defined Height)
- Offset
- Chamfer
- Fillet
- 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
### Mesh Generation

#### 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>

- **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 an enhanced 3D mapping technique which sub-divides geometry into volume mappable sub-regions.

David R. White (1996)

Pseudo-Cartesian shapes have interior and exterior angles that are close to $\pi/2$. 
FEA provides various size control methods and **adaptive seeding function** based on **user-specified mesh size** and **geometric characteristics**.
Mesh Generation

Mesh Protrusion

Advanced Nonlinear and Detail Analysis Program

**Extrude** (2D→3D)

**Fill** (Curve→2D)

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

**Revolve** (2D→3D)
Mesh Protrusion

**Mesh Generation**

- **Project** (Curve → 3D)
  - Top
  - Bottom
  - Same Topology

- **Fill** (2D → 3D)

- **Sweep** (2D → 3D)
  - Guide Curve
  - Scaled Sweep

- **Offset** (2D → 3D)
**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
**Modeling Method**
- Embedded Bar (In-compatible Mesh)
- Truss (Compatible Mesh) + Interface (Slip, Friction)
Mesh Generation

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
Geometry Modeling
Mesh Generation
Analysis
Post-processing

02 15 23 40 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>Truss (Gap / Hook / Cable)</td>
<td>1&lt;sup&gt;st&lt;/sup&gt;</td>
<td>Total Lagrangian</td>
</tr>
<tr>
<td></td>
<td>Beam</td>
<td>1&lt;sup&gt;st&lt;/sup&gt;</td>
<td>Total Lagrangian</td>
</tr>
<tr>
<td></td>
<td>Plane Stress (Quad / Tria)</td>
<td>1&lt;sup&gt;st&lt;/sup&gt;, 2&lt;sup&gt;nd&lt;/sup&gt;</td>
<td>Total/Updated Lagrangian</td>
</tr>
<tr>
<td></td>
<td>Plane Strain (Quad / Tria)</td>
<td>1&lt;sup&gt;st&lt;/sup&gt;, 2&lt;sup&gt;nd&lt;/sup&gt;</td>
<td>Total/Updated Lagrangian</td>
</tr>
<tr>
<td></td>
<td>Axisymmetry (Quad / Tria)</td>
<td>1&lt;sup&gt;st&lt;/sup&gt;, 2&lt;sup&gt;nd&lt;/sup&gt;</td>
<td>Total/Updated Lagrangian</td>
</tr>
<tr>
<td></td>
<td>Plate (Quad / Tria)</td>
<td>1&lt;sup&gt;st&lt;/sup&gt;, 2&lt;sup&gt;nd&lt;/sup&gt;</td>
<td>Total/Updated Lagrangian</td>
</tr>
<tr>
<td></td>
<td>Shell (Quad / Tria)</td>
<td>1&lt;sup&gt;st&lt;/sup&gt;, 2&lt;sup&gt;nd&lt;/sup&gt;</td>
<td>Total/Updated Lagrangian</td>
</tr>
<tr>
<td></td>
<td>Brick / Wedge / Tetra</td>
<td>1&lt;sup&gt;st&lt;/sup&gt;, 2&lt;sup&gt;nd&lt;/sup&gt;</td>
<td>Total/Updated Lagrangian</td>
</tr>
<tr>
<td>Nonstructural</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>Point</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>Matrix</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>3D Point</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>2D</td>
<td>1&lt;sup&gt;st&lt;/sup&gt;, 2&lt;sup&gt;nd&lt;/sup&gt;</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>3D (Quad / Tria)</td>
<td>1&lt;sup&gt;st&lt;/sup&gt;, 2&lt;sup&gt;nd&lt;/sup&gt;</td>
<td>-</td>
</tr>
<tr>
<td>Reinforcement</td>
<td>Embedded Bar</td>
<td>1&lt;sup&gt;st&lt;/sup&gt;, 2&lt;sup&gt;nd&lt;/sup&gt;</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>Embedded Grid (Quad / Tria)</td>
<td>1&lt;sup&gt;st&lt;/sup&gt;, 2&lt;sup&gt;nd&lt;/sup&gt;</td>
<td>-</td>
</tr>
<tr>
<td>Heat Transfer</td>
<td>1D, 2D, 3D, Cooling Pipe, Thermal Link</td>
<td>1&lt;sup&gt;st&lt;/sup&gt;, 2&lt;sup&gt;nd&lt;/sup&gt;</td>
<td>-</td>
</tr>
</tbody>
</table>
**Loadings**
- Body Force
- Force / Moment
- Mass
- Displacement
- Pressure
- Beam Load
- Pre-stress
- Temperature

**Velocity / Acceleration**
- Heat Generation
- Heat Flux
- Time Forcing Function
- Time Varying Load
- Ground Acceleration
- Response Spectrum Function

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

**Spatially Varying Pressure (Function Applied)**

**Constraint based-on CSys.**
FEA provides **arbitrary loading** function which can be applied to **arbitrary locations/areas regardless of node and/or element connection**.

- **Point Load (1D, 2D, 3D)**
- **Edge Load (1D, 2D, 3D)**
- **Rectangular Pressure (2D, 3D)**
- **Circular Pressure (2D, 3D)**
- **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>
<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>

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

**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

---

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)

**Simply Supported Stiffened Plate (Plate + Beam)**
- **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 Nonlinearity Analysis

- **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.

<Ex> Nonlinear Elastic Material for Solid Element

### Model Type
- User supplied material

### Parameters
- User Supplied Material Library File
- Number of Parameters (NLV)
- Number of Internal State Variable (NSV)
- Number of Integer Indicator Variables (NIV)

### USM Dialog

### User-defined Parameters Input Dialog

#### Input
- Strain

#### Output
- Total Stress Stiffness Matrix

#### USRMAT.DLL
- C:\FEA Works\USRMA.DLL
Geometry Nonlinearity Analysis

- **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*

- Deformation
- Maximum Principal Stress of Concrete with Deformation

Stress of Embedded Reinforcements
**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

**Steel Reinforced Concrete Bracket**

**Crack Pattern (Disc Plot)**

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

- **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)

**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
  - $U_x = 10^-3 \frac{(2x+y+z)}{2}$
  - $U_y = 10^-3 \frac{(x+2y+z)}{2}$
  - $U_z = 10^-3 \frac{(x+y+2z)}{2}$

→ Constant Stress (OK)

- Dimension
  - $L_x = 0.24$, $L_y = 0.12$
- <1> Membrane
  - $U_x = 10^-3 \frac{(x+y)}{2}$
  - $U_y = 10^-3 \frac{(y+x)}{2}$
- <2> Bending (Transverse)
  - $U_z = 10^-3 \frac{(x^2+xy+y^2)}{2}$
  - $R_x = 10^-3 \frac{(y+x/2)}{2}$
  - $R_y = -10^-3 \frac{(x+y/2)}{2}$

→ Constant Stress (OK)

All constant stress values are always checked and verified!
Refined Mesh Test

Pinched Cylindrical Shell

- Theoretical deflection is $4.5197 \times 10^{-4}$.
- FEA shows superior and monotonic convergence in various mesh divisions.

Analysis
**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

Pressure = 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

**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 [MPa]</td>
<td>21,120 [MPa]</td>
</tr>
</tbody>
</table>

| Pressure = 1, 2, 5 |

**Analysis**

Benchmark Test
Advanced Nonlinear and Detail Analysis Program

Coverage Analysis

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.

Dev. Management & Bug Tracking System (Web-based)

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

Overview
Geometry Modeling
Mesh Generation
Analysis
Post-processing
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

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
- Clipping Plane Definition by Mouse Dragging
- ½ Clipped Plot with Feature-Edge
- ½ Clipped Plot with Diagram on Middle Clipping Plane
Post-processing

Clipping Plot

Original Plot

Multiple Clipping Planes
Advanced Nonlinear and Detail Analysis Program

Post-processing

Partition Plot

Analysis Model

Geotechnical Model with Multiple Strata Configuration

Result (Partition Plot)
Symmetry Plane

Symmetric Model

Mirrored Contour Plot

Mirrored Deformed Shape
Post-processing

Diagram Plot

Solid Type

Section Plot of Frames

Diagram Plot with Deformed Shape

Line Type
Post-processing

Vector Plot

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

Vector Plot with Deformation

Vector Plot with Contour

Vector Plot with Transparent Geometry
• Results can be extracted based on:
  - Time (Transient Analysis)
  - Step (Nonlinear / Construction Stage Analysis)
  - Coordinate (User-defined Coordinate Sys.)
On-Curve Diagrams

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

#### Fault Zone

#### 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>102.00</td>
<td>0.0001</td>
</tr>
<tr>
<td>2</td>
<td>100.09</td>
<td>34.75</td>
<td>102.00</td>
<td>0.0001</td>
</tr>
<tr>
<td>3</td>
<td>98.05</td>
<td>34.75</td>
<td>102.00</td>
<td>0.0001</td>
</tr>
<tr>
<td>4</td>
<td>96.01</td>
<td>34.75</td>
<td>102.00</td>
<td>-0.0001</td>
</tr>
<tr>
<td>5</td>
<td>93.96</td>
<td>34.75</td>
<td>102.00</td>
<td>-0.0001</td>
</tr>
<tr>
<td>6</td>
<td>91.92</td>
<td>34.75</td>
<td>102.00</td>
<td>-0.0003</td>
</tr>
<tr>
<td>7</td>
<td>89.88</td>
<td>34.75</td>
<td>102.00</td>
<td>-0.0005</td>
</tr>
<tr>
<td>8</td>
<td>87.84</td>
<td>34.75</td>
<td>102.00</td>
<td>-0.0006</td>
</tr>
<tr>
<td>9</td>
<td>85.80</td>
<td>34.75</td>
<td>102.00</td>
<td>-0.0011</td>
</tr>
<tr>
<td>10</td>
<td>83.75</td>
<td>34.75</td>
<td>102.00</td>
<td>-0.0014</td>
</tr>
<tr>
<td>11</td>
<td>81.71</td>
<td>34.75</td>
<td>102.00</td>
<td>-0.0017</td>
</tr>
<tr>
<td>12</td>
<td>79.66</td>
<td>34.75</td>
<td>102.00</td>
<td>-0.0019</td>
</tr>
<tr>
<td>13</td>
<td>77.62</td>
<td>34.75</td>
<td>102.00</td>
<td>-0.0021</td>
</tr>
<tr>
<td>14</td>
<td>75.58</td>
<td>34.75</td>
<td>102.00</td>
<td>-0.0022</td>
</tr>
<tr>
<td>15</td>
<td>73.54</td>
<td>34.75</td>
<td>102.00</td>
<td>-0.0024</td>
</tr>
<tr>
<td>16</td>
<td>71.40</td>
<td>34.75</td>
<td>102.00</td>
<td>-0.0024</td>
</tr>
</tbody>
</table>

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

### Front View
Advanced Nonlinear and Detail Analysis Program

Post-processing

Probe & Result Tag

Nodal Result

Sectional Result with Clipped Plot (Element Result)

Probe & Add Result Tags at Specified Nodes/Elements
In FEA, legend can be controlled for its position, size, format and range (including min/max value) by mouse dragging.

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

**Drag side to resize legend box**

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