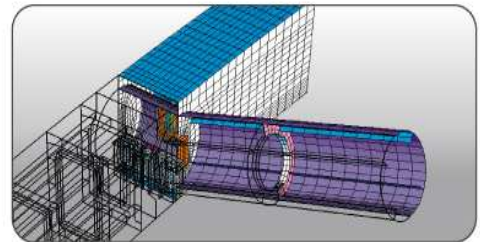
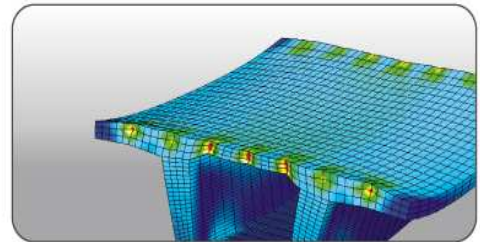
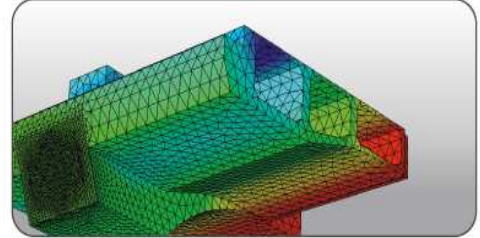




<http://eng.midasuser.com/fea>

*New Paradigm for
advanced nonlinear and detail analysis*



midas **FEA**

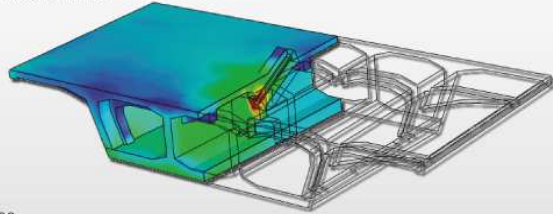
Product Portfolio



New Paradigm for advanced nonlinear and detail analysis

midas FEA is state of the art software, which defines a new paradigm for advanced nonlinear and detail analysis for civil and structural engineering applications including plain and reinforced concrete structures, concrete damage and cracking, plain and reinforced masonry structures, composite structures, steel structures, foundations, and offshore structures.

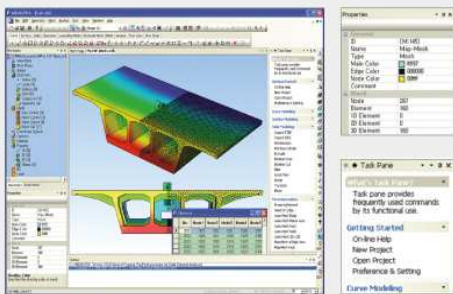
midas FEA, combining a powerful pre/post processor and solver co-developed by MIDAS IT and TNO DIANA, stands for reliability and accurate solutions and is founded on expertise in geometry modeling, Auto-mesh generation, contemporary graphics and analysis technologies.



FEA Modeling

Task-Oriented User Interface

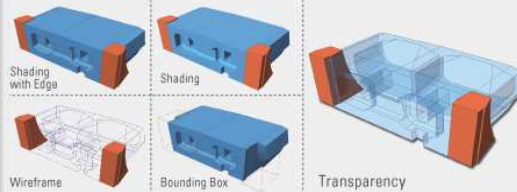
midas FEA provides **user-friendly** Task-oriented User Interface, which guides the user through the work process by **dynamically changing the dialog boxes** based on the current menu and operation step.



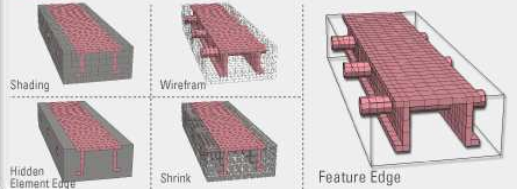
midas FEA Framework

Effective Visualization

Geometry

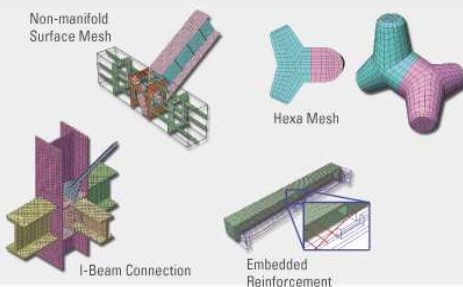


Mesh



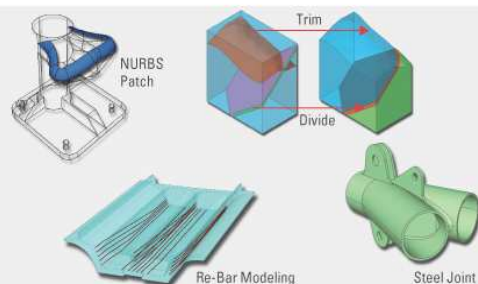
Powerful and Easy Mesh Generation

- Auto-mesh generation for surface and solid (Loop, Grid, Delaunay)
- Map-mesh generation (Quad, Hexa, Penta)
- Enhanced protrusion methods (Extrude, Revolve, Sweep, Fill, Project)
- Convenient interface element generation
- Easy reinforcement bar and grid definition



Intuitive Geometry Modeling

- Various surface type (Plane, Coons, NURBS)
- Advanced modeling tools
 - Surface Intersection
 - Surface-Surface Divide and Trim
- Extrude, revolve, loft, sweep, etc
- Boolean Operation and much more advanced tools



Linear Static Analysis · Construction Stage Analysis · Modal Analysis · Linear Buckling Analysis · Transient / Spectrum Response Analysis
Heat of Hydration Analysis · Material / Geometry Nonlinearity Analysis · Interface Nonlinearity Analysis · Reinforcement Analysis · Concrete
Cracking Analysis · Contact Analysis · Fatigue Analysis

▶ Analysis Features

■ Linear Static Analysis

Multiple Load Cases & Combination

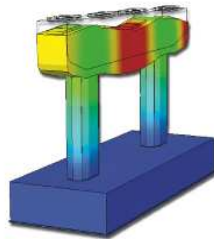
Output Control (Data, Node, Element)

Result Coordinate System

Extensive Element Library

Equation Solvers

- Direct Solvers
- Multi-frontal Sparse Gaussian Solver
- Skyline Solver
- Iterative Solvers
- PCG, GMRES (Unsymmetric)



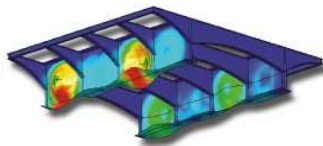
■ Eigenvalue Analysis

Modal Analysis

- Lanczos Method
- Subspace Iteration
- Sturm-Sequence Check
- Include Rigid Body Modes
- Modal Participation Factors

Linear Buckling Analysis

- Critical Load Factors
- Buckling Modes
- Load Combinations & Factors



■ Heat of Hydration Analysis

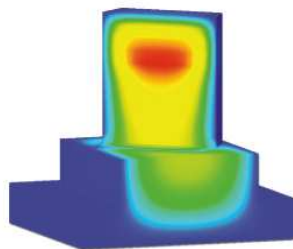
Heat Transfer

- Transient
- Heat Generation
- Conduction / Convection
- Pipe Cooling

Concrete Behavior

- Creep / Shrinkage
- Compressive Strength
- Design Codes (CEB-FIP, ACI, AASHTO, etc.)

Parametric Study Wizard



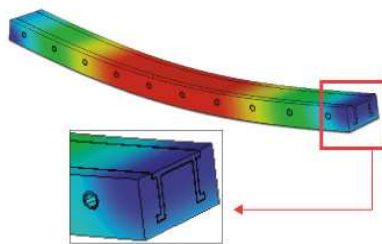
■ Interface Nonlinearity Analysis

Interface Elements

- Point, Line, Plane

Interface Models

- Rigid
- Coulomb Friction
- Discrete Cracking
- Crack Dilatancy
- Bond-Slip
- Combined CSC



Steel-Concrete Composite Girder

■ Construction Stage Analysis

Activation & Deactivation

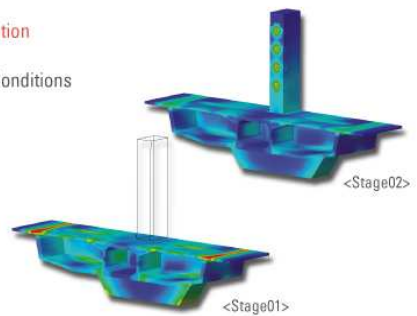
- Elements
- Load & Boundary Conditions

Stage Definition

- Drag & Drop U.I.
- Definition Wizard
- Simulation

Material Nonlinearity

Restart



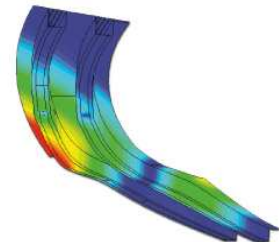
■ Dynamic Analysis

Transient Response

- Direct Integration
- Mode Superposition
- Time Forcing Function DB
- Time Varying Loads
- Ground Acceleration
- Time History Plot / Graph

Response Spectrum

- SRSS, CQC, ABS
- Design Spectrum DB
- Seismic Data Generator



■ Reinforcement Analysis

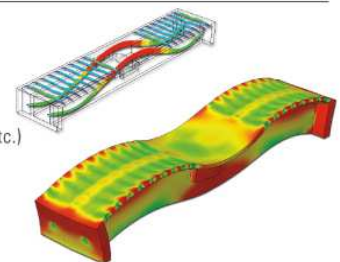
Reinforcement

- Embedded Bar
- Embedded Grid
- Various Mother Elements (Solid, Plate, Axisymmetric, etc.)

Post tensioned Prestress

Material Nonlinearity

Geometric Nonlinearity



Prestressed Concrete Girder (Embedded Bar)

■ Non linear Analysis

Material Nonlinearity

- von Mises, Tresca, Rankine, Drucker-Prager
- Mohr-Coulomb, User-Supplied Material, etc.

Geometric Nonlinearity

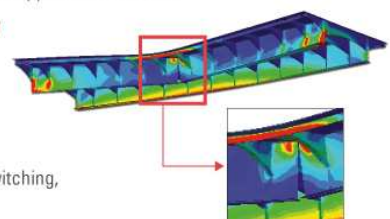
Concrete Crack Model

Contact Nonlinearity

Iteration Schemes

- N-R, Modified N-R
- Arc-Length, Auto-Switching, Auto-Load Step, etc.

Restart



Post-processing

Post-Works Flow

The Post-Works Flow diagram illustrates the process of exporting data from the software. It starts with the 'Post-Works' tree on the left, which lists various analysis stages like 'BT1 Stage 1-Last Step' and 'Reaction'. This leads to a 'Contour Plot' of a mechanical part. Below the plot is a 'Result Table' with columns for Node, T1, T2, and T3. An arrow points from the table to a 'Microsoft Excel' spreadsheet showing the data in a grid format.

Node	T1	T2	T3
1	0.004384	0.003727	-0.004330
2	0.004824	0.003284	-0.004881
3	0.004865	0.003326	-0.004922
4	0.005325	0.003801	-0.005382
5	0.005787	0.004271	-0.005843
6	0.004475	0.003295	-0.004420
7	0.003368	0.002389	-0.003313
8	0.002311	0.001332	-0.002256
9	0.002251	0.001274	-0.002211
10	0.004687	0.003989	-0.004632
11	0.005145	0.004452	-0.005087
12	0.004684	0.003986	-0.004629

Various Contour Display

This section shows six different ways to display contour results on a 3D model:

- Contour with Mesh:** Shows the contour overlaid on the finite element mesh.
- Contour with Feature Edge:** Highlights the contours along the sharp edges and features of the part.
- Contour with Iso-Line:** Displays contour lines (isolines) over the surface.
- Gradient Contour:** Shows the direction and magnitude of the stress or strain gradient.
- 2-Color Contour:** Uses only two colors to represent the range of values.
- Gray Contour:** Displays the contour using a grayscale gradient.

Special Contour Plotting Method

This section illustrates four advanced contour plotting techniques:

- Multiple Iso-surfaces:** Shows multiple surfaces of constant value (isovalue) in a 3D space.
- Capped Iso Surface:** Shows a single isosurface with a flat top or bottom, useful for identifying specific levels.
- Slice Plane:** Allows for viewing the internal stress distribution by cutting through the model with a plane.
- Partition Plot:** Divides the model into sections to compare results across different parts or stages.

Result Evaluation

Probe Result

Displays result values at selected Nodes/Elements

The 'Probe Result' window displays a table of values for selected nodes or elements. The table includes columns for Node, Type, X, Y, Z, and Value. A 3D model of a mechanical part is shown with several red arrows pointing to specific nodes, indicating where the data was extracted.

Node	Type	X	Y	Z	Value
1	Node	20.000	10.000	0.000	1.234567
2	Node	30.000	20.000	10.000	2.345678
3	Node	40.000	30.000	20.000	3.456789
4	Node	50.000	40.000	30.000	4.567890
5	Node	60.000	50.000	40.000	5.678901
6	Node	70.000	60.000	50.000	6.789012
7	Node	80.000	70.000	60.000	7.890123
8	Node	90.000	80.000	70.000	8.901234
9	Node	100.000	90.000	80.000	9.012345
10	Node	110.000	100.000	90.000	10.123456

On-curve Diagram

Plots a graph or table of values along specified lines

The 'On-curve Diagram' shows a 3D model with a red line drawn along a specific path. Below the model is a table of values extracted along this path. The table has columns for ID, X (mm), Y (mm), Z (mm), and Value.

ID	X (mm)	Y (mm)	Z (mm)	Value
1	100.00	50.00	0.00	1.2345
2	100.00	50.00	10.00	2.3456
3	100.00	50.00	20.00	3.4567
4	100.00	50.00	30.00	4.5678
5	100.00	50.00	40.00	5.6789
6	100.00	50.00	50.00	6.7890
7	100.00	50.00	60.00	7.8901
8	100.00	50.00	70.00	8.9012
9	100.00	50.00	80.00	9.0123
10	100.00	50.00	90.00	10.1234
11	100.00	50.00	100.00	11.2345
12	100.00	50.00	110.00	12.3456
13	100.00	50.00	120.00	13.4567
14	100.00	50.00	130.00	14.5678
15	100.00	50.00	140.00	15.6789
16	100.00	50.00	150.00	16.7890
17	100.00	50.00	160.00	17.8901
18	100.00	50.00	170.00	18.9012
19	100.00	50.00	180.00	19.0123
20	100.00	50.00	190.00	20.1234

Extract Result

Extracts results based on:

- Time (Transient)
- Step (Nonlinear / CS)
- Coordinate System

An 'MS-Excel compatible Table' showing a grid of numerical data extracted from the simulation, organized in a standard spreadsheet format.

The 'Graph' section shows a 3D model of a curved beam on the left and a corresponding line graph on the right. The graph plots the results (likely stress or displacement) over time, showing the transient behavior of the structure.

Report Generation

Automatic report generation
Customizable data and graphs
MS-Excel/Word compatible

This section displays two types of reports generated from the software:

- MS-Excel Report:** A screenshot of an Excel spreadsheet containing detailed analysis results, including stress distributions and displacement data.
- HTML Report:** A screenshot of a web-based report showing a 3D model with stress contours and a color scale legend, along with a table of data.