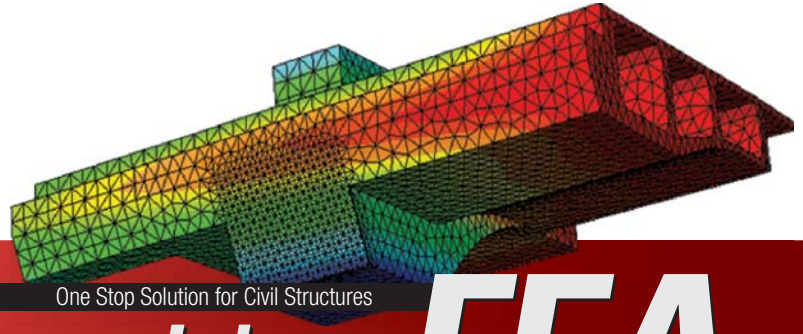


One Stop Solution for Civil Structures

midas FEA

Advanced Nonlinear and Detail Analysis System

Global Network

Modeling, Integrated Design & Analysis Software

We Analyze and Design the Future.

MIDASoft USA / New York

5 Penn Plaza, Suite 2324
New York, NY 10001, USA
+1 212 835 1666 / midasoft@midasuser.com

MIDAS IT China / Beijing

Rm.1307, 13th Floor, Fangyuan 56(Yi),
Zhongguancun Nan da Jie Haidian
District, Beijing, China
+86 010 51658210 / beijing@midasuser.com

MIDAS R&D Center India / Mumbai

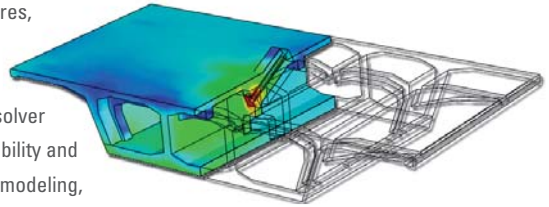
804 & 805, 8th Floor, Maithili's Signet,
Plot No. 39/4, Sector 30A Vashi,
Navi Mumbai – 400705
+91 22 6721 1001~3 / india@midasuser.com

MIDAS IT Japan / Tokyo

Akihabara OS Building 7th fl., 5-3-1 Sotokanda,
Chiyoda-ku, Tokyo, 101-0021, Japan
+81 3 5817 0789 / midas_japan@midasit.com

midas FEA About midas FEA

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.

2006.11.Strategic alliance with TNO DIANA

Why midas FEA?

01. Unique
- Distinct and sophisticated visual interface
- Versatile representations of analysis results
- Powerful geometry modeling capabilities

02. Easy
- Intuitive Framework
- Task-oriented User Interface
- Modeling complex 3D geometries
- Automatic Mesh Generation
- Mesh manipulation

03. Powerful
- Numerous Analysis Features
- Static and Dynamic Load and Boundary Conditions
- Unprecedented Fast Analysis Speed

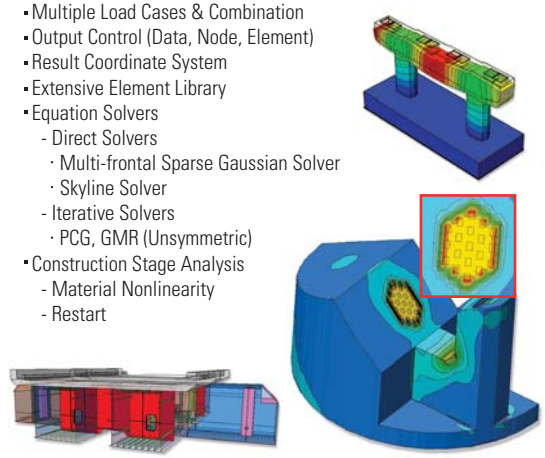
04. Practical
- Evaluation of results
- Intuitive Construction Stage Definition
- Analysis Control
- Report Generation

05. Reliable
- MIDAS Quality Control System (MQCS)
- Certification for ISO 9001
- Verifications Examples
- Responsive user support

midas FEA Application Areas

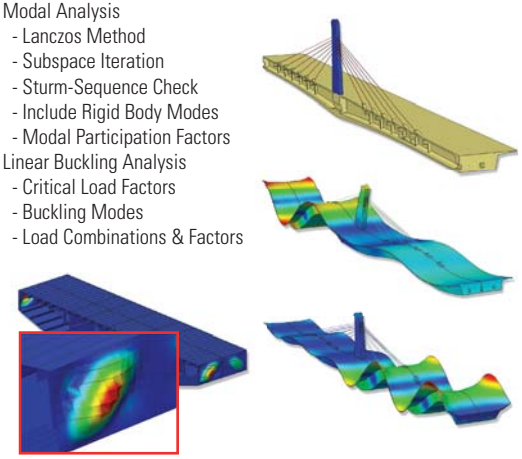
01. Linear Static Analysis

- 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, GMR (Unsymmetric)
- Construction Stage Analysis
  - Material Nonlinearity
  - Restart



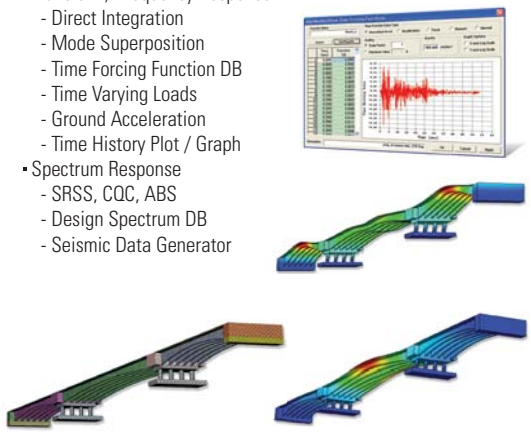
02. Eigenvalue Analysis

- 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



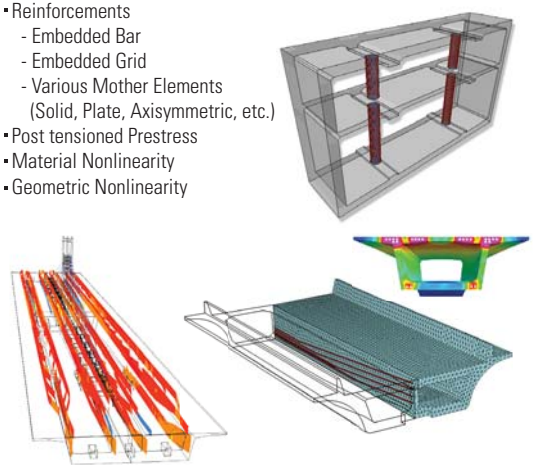
03. Dynamic Analysis

- Transient / Frequency Response
  - Direct Integration
  - Mode Superposition
  - Time Forcing Function DB
  - Time Varying Loads
  - Ground Acceleration
  - Time History Plot / Graph
- Spectrum Response
  - SRSS, CQC, ABS
  - Design Spectrum DB
  - Seismic Data Generator



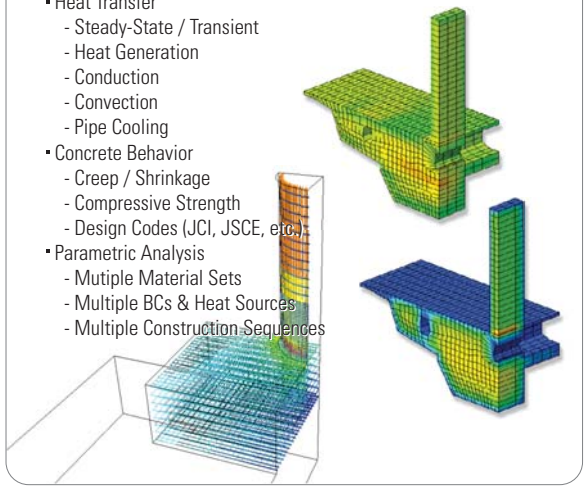
04. Reinforcement Analysis

- Reinforcements
  - Embedded Bar
  - Embedded Grid
  - Various Mother Elements (Solid, Plate, Axisymmetric, etc.)
- Post tensioned Prestress
- Material Nonlinearity
- Geometric Nonlinearity



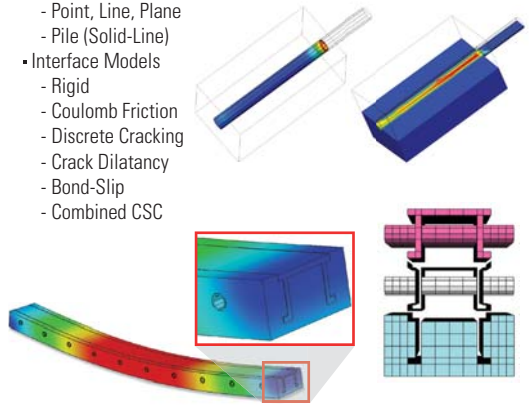
05. Heat of Hydration Analysis

- Heat Transfer
  - Steady-State / Transient
  - Heat Generation
  - Conduction
  - Convection
  - Pipe Cooling
- Concrete Behavior
  - Creep / Shrinkage
  - Compressive Strength
  - Design Codes (JCI, JSCE, etc.)
- Parametric Analysis
  - Mutiple Material Sets
  - Multiple BCs & Heat Sources
  - Multiple Construction Sequences



06. Interface Nonlinearity Analysis

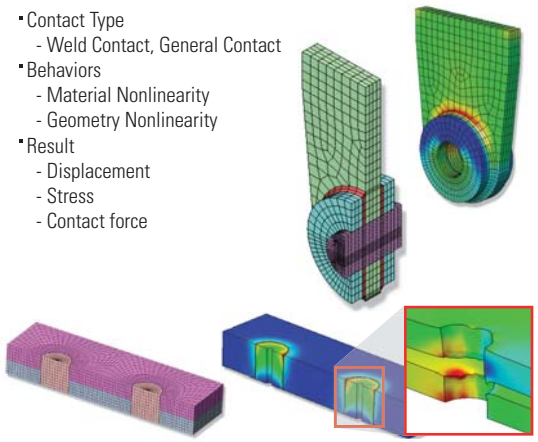
- Interface Elements
  - Point, Line, Plane
  - Pile (Solid-Line)
- Interface Models
  - Rigid
  - Coulomb Friction
  - Discrete Cracking
  - Crack Dilatancy
  - Bond-Slip
  - Combined CSC



midas FEA Application Areas

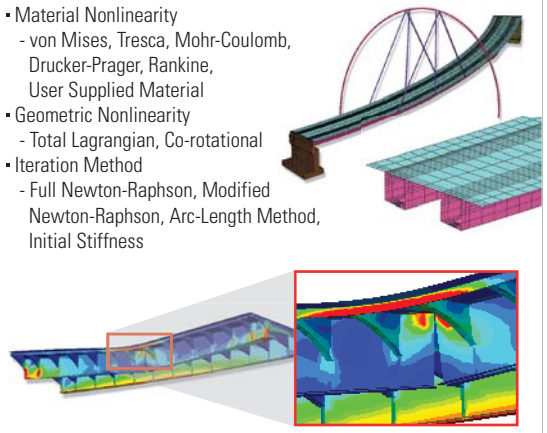
07. Contact Analysis

- Contact Type
  - Weld Contact, General Contact
- Behaviors
  - Material Nonlinearity
  - Geometry Nonlinearity
- Result
  - Displacement
  - Stress
  - Contact force



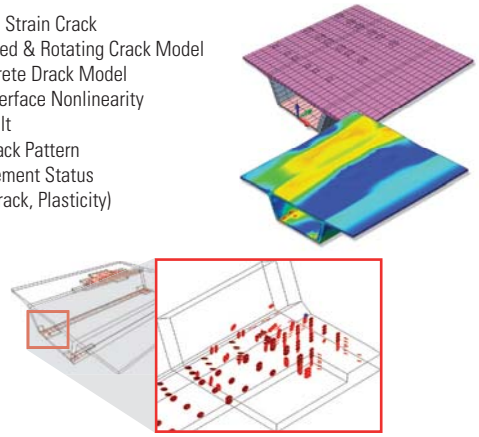
08. Nonlinear Static Analysis

- Material Nonlinearity
  - von Mises, Tresca, Mohr-Coulomb, Drucker-Prager, Rankine, User Supplied Material
- Geometric Nonlinearity
  - Total Lagrangian, Co-rotational
- Iteration Method
  - Full Newton-Raphson, Modified Newton-Raphson, Arc-Length Method, Initial Stiffness



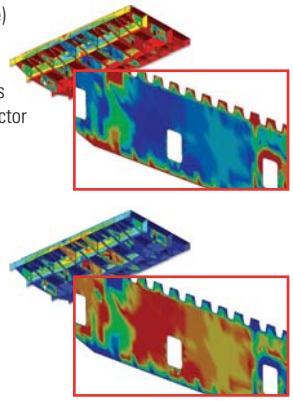
09. Cracking Analysis

- Total Strain Crack
  - Fixed & Rotating Crack Model
- Discrete Drack Model
  - Interface Nonlinearity
- Result
  - Crack Pattern
  - Element Status (Crack, Plasticity)



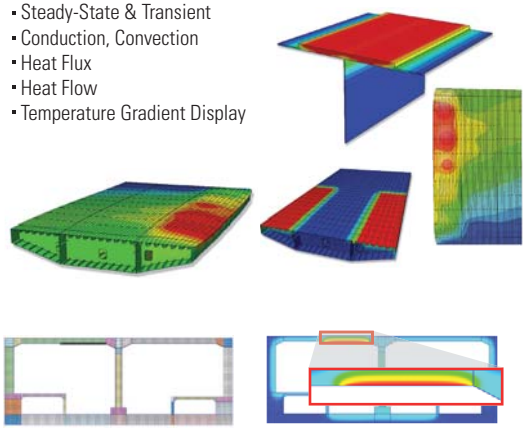
10. Fatigue Analysis

- Methods and Parameters
  - S-N Method (Stress-Life)
  - Load / Stress History
  - Rainflow Counting
  - Mean Stress Corrections
  - Stress Concentration Factor
  - Modifying Factors
- Results
  - Cycles to Failure
  - Safety Factor (Cycles to Failure / Desired Repetition)
  - Damage estimation



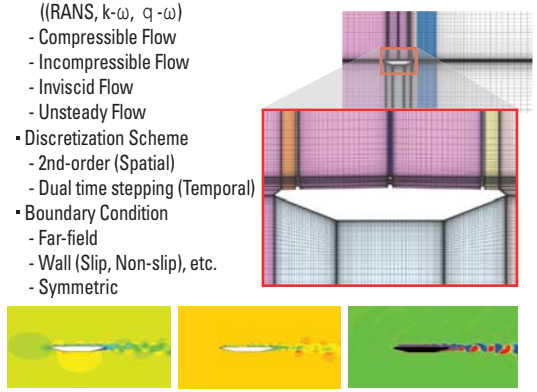
11. Heat Transfer/Stress Analysis

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



12. CFD Analysis

- CFD Models
  - Turbulence Models ((RANS, k-ω, q -ω)
  - Compressible Flow
  - Incompressible Flow
  - Inviscid Flow
  - Unsteady Flow
- Discretization Scheme
  - 2nd-order (Spatial)
  - Dual time stepping (Temporal)
- Boundary Condition
  - Far-field
  - Wall (Slip, Non-slip), etc.
  - Symmetric



midas **FEA**

"Advanced Nonlinear and Detail Analysis System"

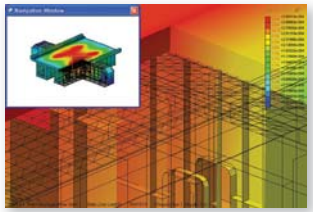
01. Graphical User Interface

Providing an integrated and efficient work environment for
Modeling, Visualization and Simulation

midas FEA - Framework

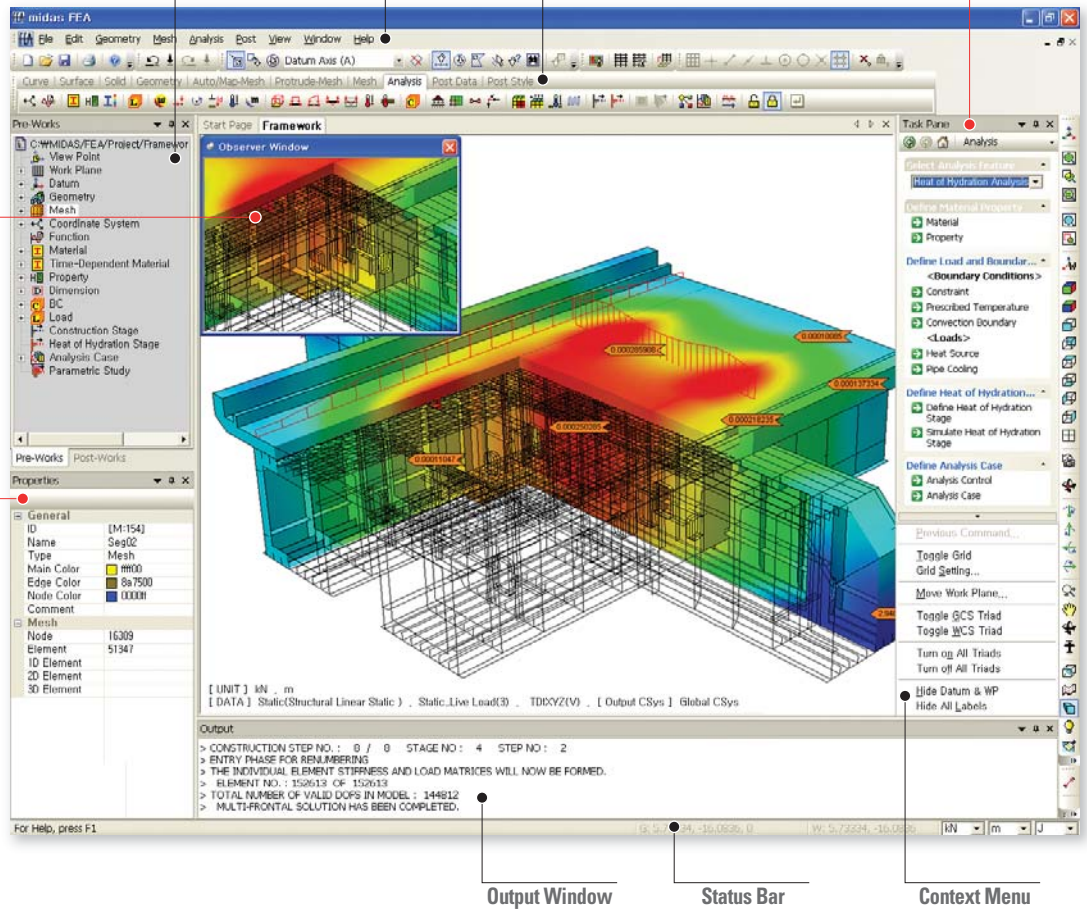
Observer / Navigation Window

Quick review for both global and detail view of analysis results



Property Window

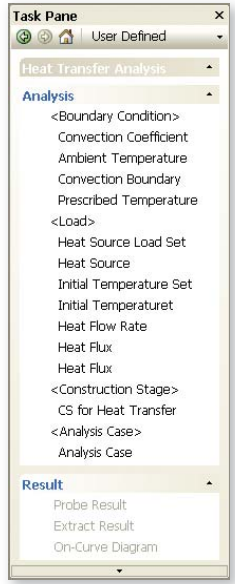
- Property Window provides various property information for quick query/edit.
- Customize detailed graphic output for result generation



Task Pane

A new concept tool, which enables the user to freely set optimal menu system

- A new concept menu system comprising frequently used menu
- Procedural sequence defined by the user for maximum efficiency
- Auto-links to manuals, technical papers and tutorials
- Links to corresponding dialog boxes for ease of checking input data

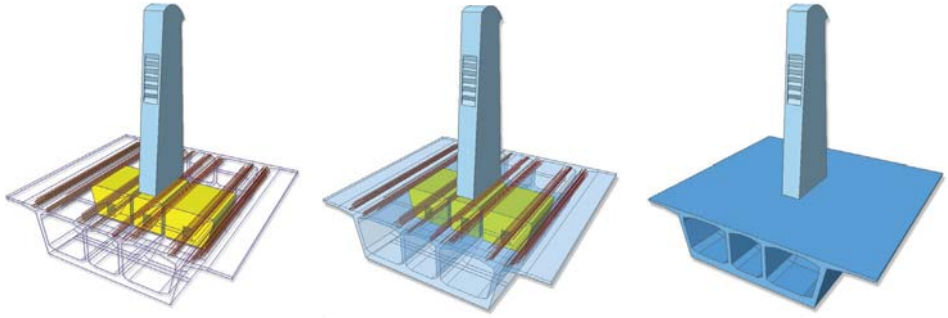


02. Pre Processor

The powerful pre-processor is founded on the expertise
in geometry modeling, auto-mesh generation and powerful 3D graphic tools

Display Mode

Geometry Display Mode



Wireframe | Transparency | Shading

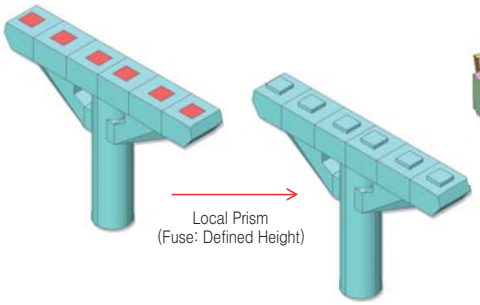
Mesh Display Mode

Feature Edge | Transparency | Shading

- Customization controls made for easy viewing geometry or mesh sets
- Full graphics control for geometry and mesh sets
- Full graphics control to view interior shapes embedded in complex structures by transforming to wireframe, or control transparency for local/global parts of the model

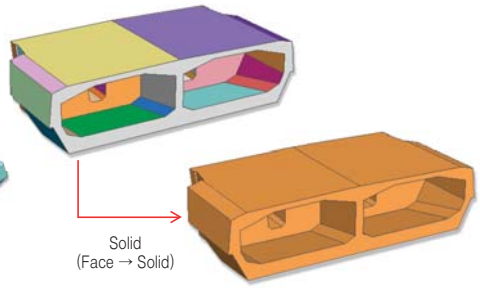
Geometry Modeling

Local Prism



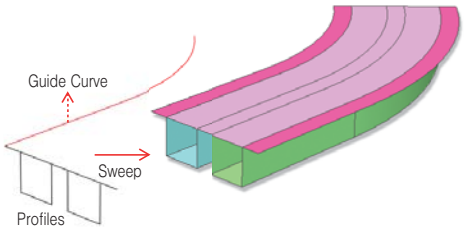
Forming a solid model by extracting specific surface

Stitch to Solid



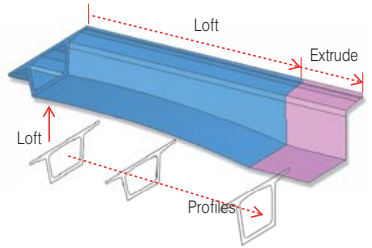
Forming a solid model by sewing surfaces

Sweep



Extrude along guide curve

Loft & Extrude



Enables to create a solid model with
variation of cross section (Tapered section)

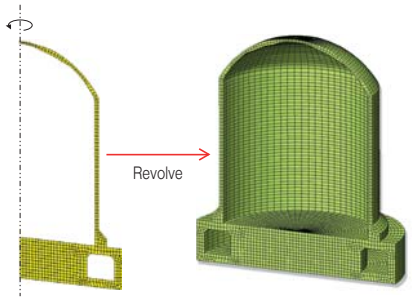
- Perform the Boolean operation (Fuse, Cut, Embed) for geometry compatibility with more than two solid objects
- Using advanced geometry tools (Loft, Sew, Sweep, Local Prism) to convert 2D geometry models to 3D geometry shapes

02. Pre Processor

The powerful pre-processor is founded on the expertise
in geometry modeling, auto-mesh generation and powerful 3D graphic tools

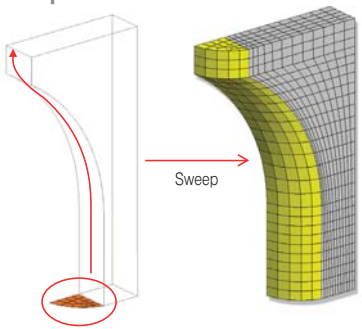
Mesh Generation

Revolve Mesh



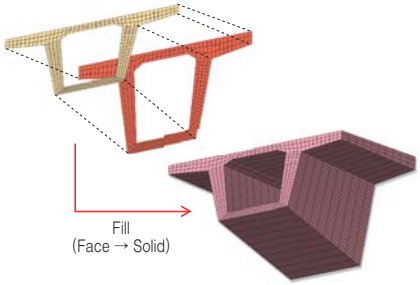
Automatically generate solid mesh rotating
2D-Mesh using reference axis

Sweep Mesh



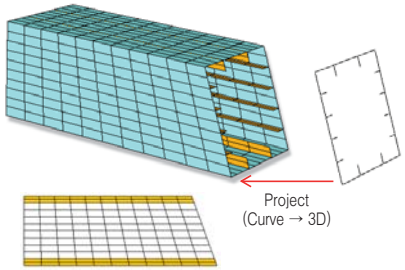
Automatically generate solid mesh by
extruding 2D-Mesh using guide curve

Fill Mesh



Fill mesh creates 3D mesh sets between
two 2D mesh shapes with similar element
size discretization.

Project Mesh

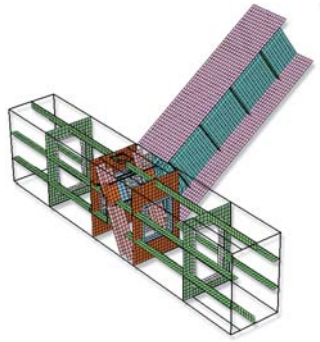


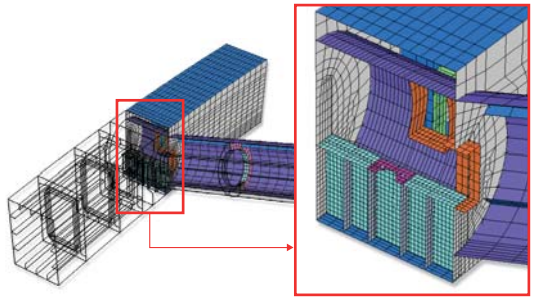
Automatically generate mesh by projecting
a mesh on a reference plane

- Includes advanced meshing function : Auto-Mesh, Map-mesh & manual generation
- Mesh extraction method from 2D/3D mesh sets or directly from a geometry object
- Capability of generating 3D mesh for complex shapes using 2D Mesh sets

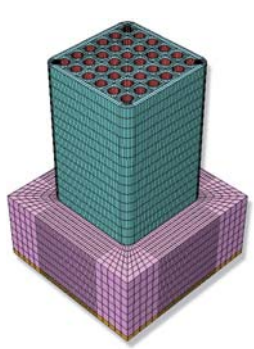
Automatic Surface/Solid Meshing

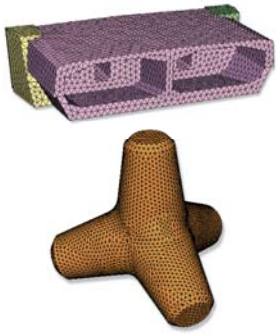
Surface Automatic Mesh





Solid Automatic Mesh





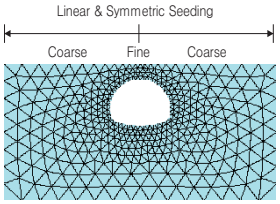
- Supports advanced meshing functions for complex geometry modeling : Manifold and on-manifold shapes
- Provides advanced mesh function such as Hybrid Meshing, Sub-Meshing as well as Auto-mesh, Map-mesh
- Fast mesh generation: 200,000 tetra elements per-minute

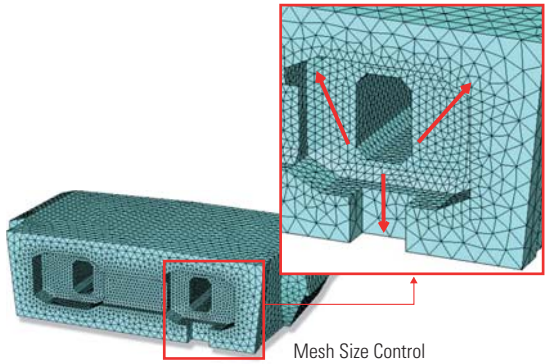
02. Pre Processor

The powerful pre-processor is founded on the expertise
in geometry modeling, auto-mesh generation and powerful 3D graphic tools

Mesh Size Control/Quality Assurance

Mesh Size Control

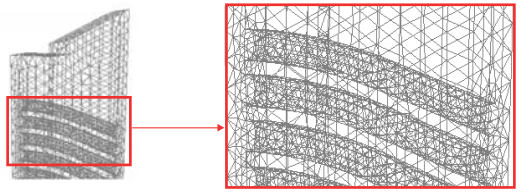




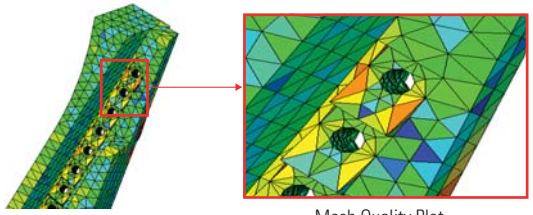
Mesh Size Control

Mesh 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



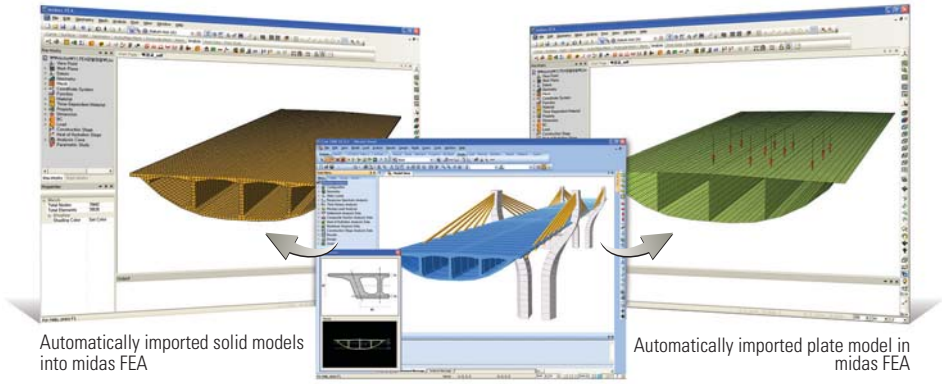
Check Free Face



Mesh Quality Plot

- Various size control functions available to modify density of nodes and elements for complex geometry or variation of material properties
- Advanced size control functions such as Adaptive Seeding, which automatically controls mesh density based on shape characteristics
- Standard Mesh quality and assurance features to detect incompatibility or poor element quality

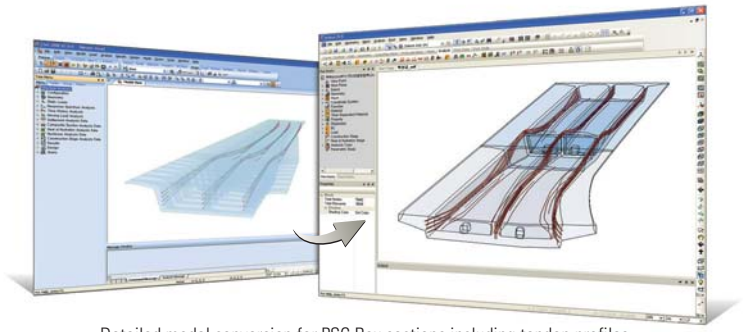
Frame to Solid (Direct Date Transfer with midas Civil)



Automatically imported solid models into midas FEA

Automatically imported plate model in midas FEA

Frame model in midas Civil



Detailed model conversion for PSC Box sections including tendon profiles

- Convert frame model and PSC section types in midas Civil into plate/solid models in midas FEA
- Convert tendon profiles and embed them as bars from midas Civil into midas FEA

midas FEA

"Advanced Nonlinear and Detail Analysis System"

03. Analysis

Providing high-end linear/nonlinear analysis
capabilities optimized for bridge and structural engineering

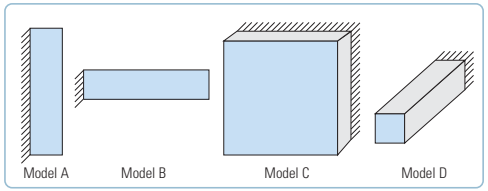
Linear Static Analysis

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



		Model A	Model B	Model C	Model D
Element Type		Plate	Plate	Solid	Solid
No. of Elements		30,000	30,000	125,000	40,000
No. of DOF's		180,600	181,800	390,150	132,300
Solution Time[sec]	Multi-frontal	35	41	3,244	262
	PCG	179	188	817	139

Net Solution time: Intel® Core™ Duo CPU @ 3GHz 2GB RAM

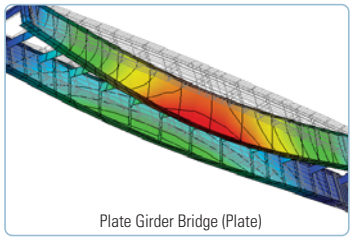
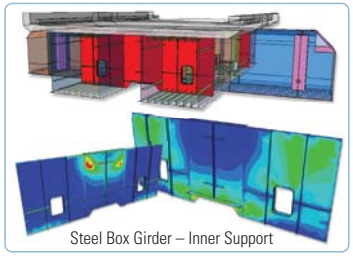
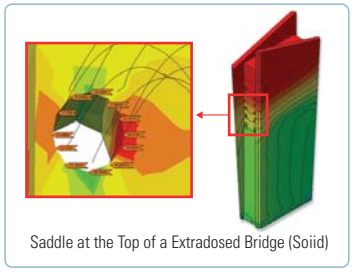


Plate Girder Bridge (Plate)



Steel Box Girder – Inner Support



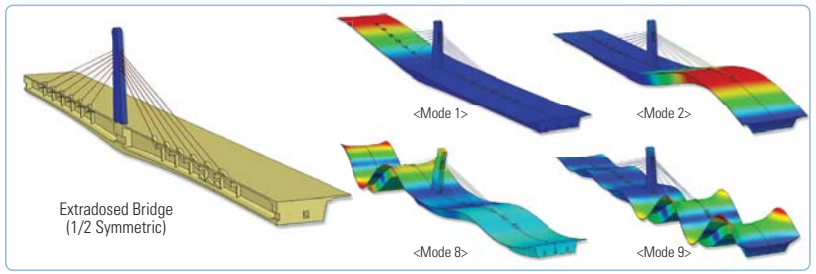
Saddle at the Top of a Extradosed Bridge (Soiid)

- Fast and accurate analysis by using Multi-frontal Sparse Gaussian Solver
- Various load and boundary conditions(Applied load at any position, Distributed load for live/dead load, Multi Point Constraint etc.)
- Accurate analysis using higher order element

Eigenvalue Analysis

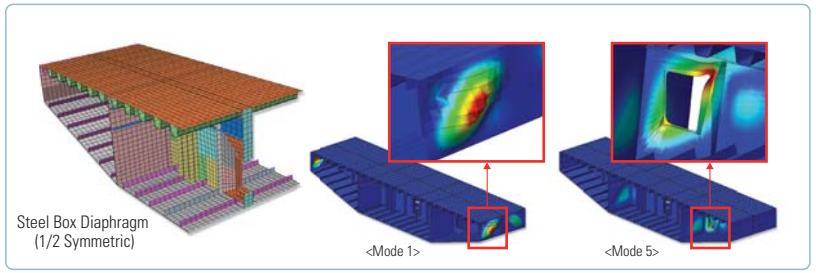
Modal Analysis

- Lanczos Method
- Subspace Iteration



Linear Buckling Analysis

- Critical Buckling Modes
- Buckling Modes
- Regulatory Load Combinations



- Eigenvalue Analysis : Lanczos Method & Subspace Iteration Method
- Generate load combination for buckling analysis
- Output buckling mode shape which has positive critical load factor (Option)
  → Removing unnecessary buckling mode

midas FEA

"Advanced Nonlinear and Detail Analysis System"

03. Analysis

Providing high-end linear/nonlinear analysis
capabilities optimized for bridge and structural engineering

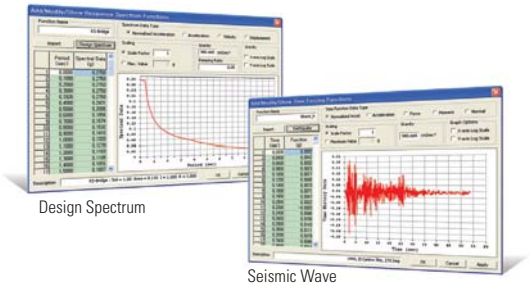
Dynamic Analysis

Transient Response Analysis

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

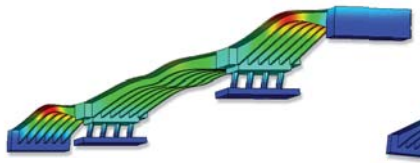
Spectrum Response Analysis

- SRSS, CQC,ABS
- Design Spectrum DB

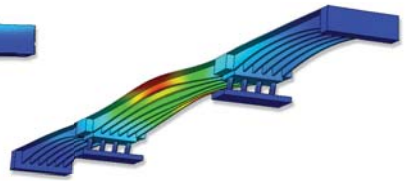


Design Spectrum

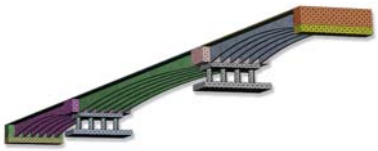
Seismic Wave



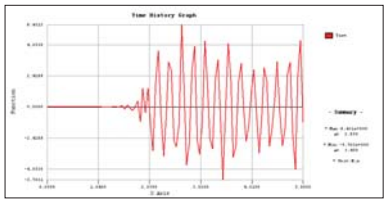
Response to the
Longitudinal Seismic Force



Response to the
Transverse Seismic Force



Integral Bridge Modeling



Acceleration at the Top of the Pier

- Supports the design response spectrum based on seismic database library
- Provides time history analysis by Direct Integration Method and Modal Time History Analysis
- Output various time history analysis result graph : nodal displacement/speed/acceleration, member force stress/strain of element

Static Nonlinear Analysis

Material Models

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

Material Nonlinearity

- Hardening (Iso/Kinematic/Mixed)
- Softening

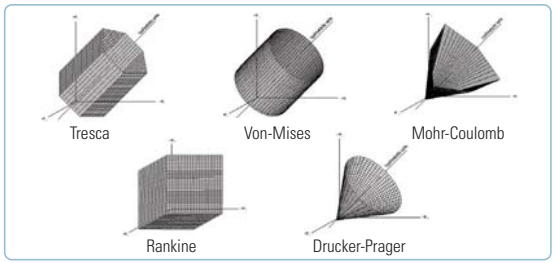
Geometric Nonlinearity

- Total Lagrangian
- Co-rotational

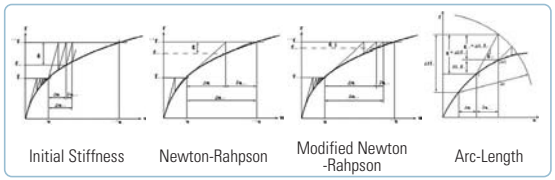
Iteration Method

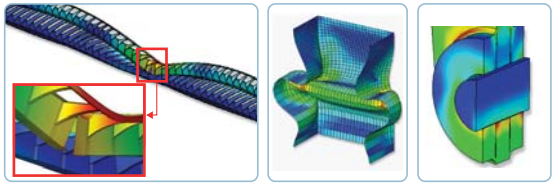
- Full Newton-Raphson
- Modified Newton-Raphson
- Arc-Length Method
- Initial Stiffness

■ Failure Criteria



■ Iteration Method





- Includes total of five constitutive models in addition to an user supplied material option for all types of material models including concrete and steel
- Consider effects of transverse varying stiffness and large deformation for structures undergoing material nonlinearity effects and large axial forces(Frame, Plate, and Solid models)
- Consider geometry and material nonlinear analysis simultaneously for frame, plate and solid elements
- Iterative analysis techniques : Initial Stiffness, Modified/Newton-Raphson, and Arc-Length
- Auto load step and restart options

midas FEA

"Advanced Nonlinear and Detail Analysis System"

03. Analysis

Providing high-end linear/nonlinear analysis
capabilities optimized for bridge and structural engineering

Reinforcement Analysis

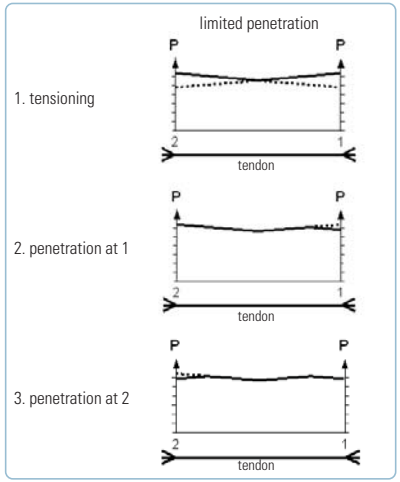
Reinforcements (Rebar & Tendon)

- Embedded Bar/Grid
- Line Type → Applied to Plate, Solid & Plain Stress
- Point Type → Applied to Plain Strain & Axisymmetric

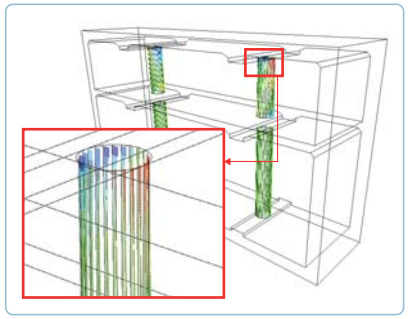
Prestressing Tendon

- Pretension & Post-Tension
- Short-Term Loss: Friction, Slip & Elastic Deform ation
- Long-Term Loss: Relaxation & Creep/Shrinkage

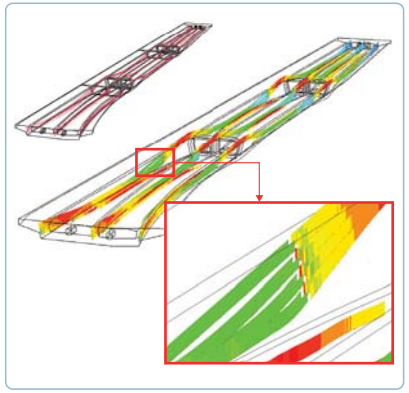
■ Loss of Pretension



■ Column Reinforcements of a Subway Station



■ Prestressing Tendons in a PSC Box Bridge



- Unique embedded techniques enables users to model tendons and reinforcement regardless of neighboring elements and nodal connectivity
- Analyses considering immediate loss(friction, slip and elastic deformation and long-term loss (tendon relaxation, creep/shrinkage effect)

Cracking Analysis

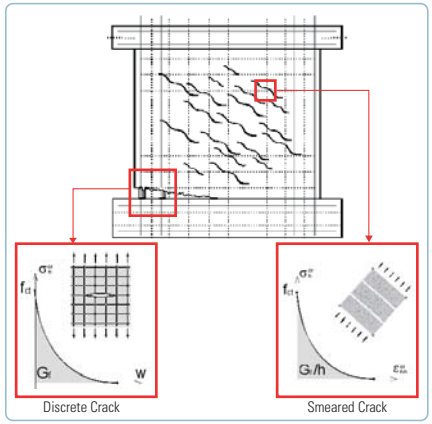
Cracking Models

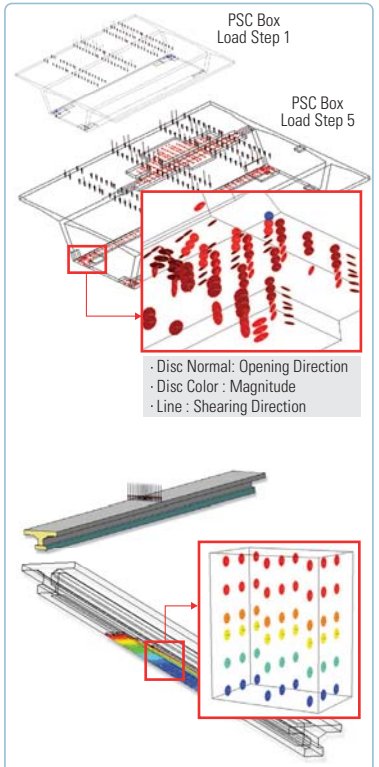
- Total Strain Crack
  - Fixed Crack Mode & Rotating Crack Model
- Discrete Crack Mode
  - Interface Nonlinearity

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

■ Concrete Crack Models





<Crack Visualization>

- Includes TSC (Total Strain Crack) for concrete crack analysis: Fixed and Rotating models
- Includes Reinforcement Analysis to portray the main concrete tension reinforcement
- Incorporate impact of rebar in tensile behavior of concrete structures
- Include analysis results and visualization of crack pattern, direction and status of the element

midas **FEA**

"Advanced Nonlinear and Detail Analysis System"

03. Analysis

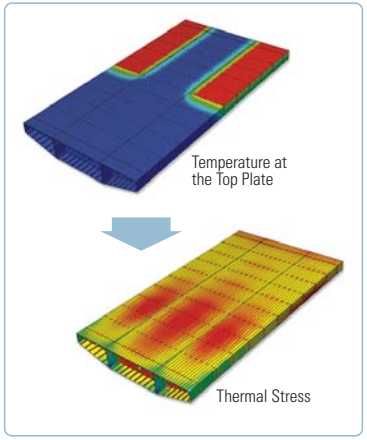
Providing high-end linear/nonlinear analysis
capabilities optimized for bridge and structural engineering

Heat Transfer Analysis

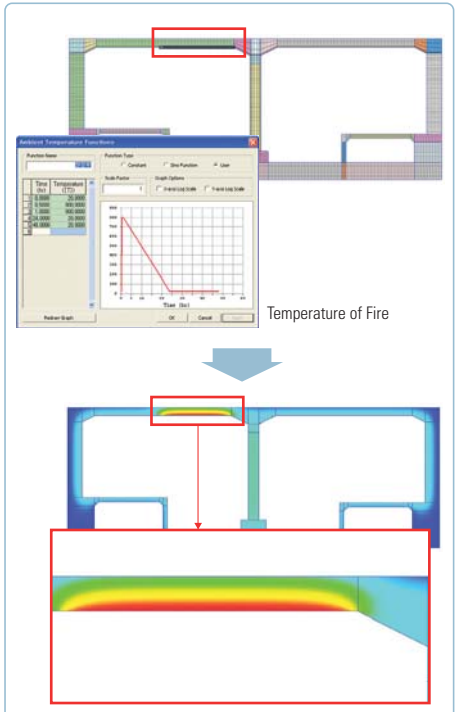
Heat Transfer

- Steady-State & Transient
- Conduction, Convection, Radiation
- Heat Flux
- Heat Flow
- Temperature Gradient Display

■ Guss Asphalt Pavement



■ Fire in a Subway Structure



- Determine the time-dependent temperature distribution and heat transfer characteristics for structures exposed to high temperatures
- Thermal stress analysis considering heat transfer phenomenon such as conduction, convection
- Perform linear analysis considering the variance in material's characteristics (intensity, specific heat, conductivity) with respect to temperature

Heat of Hydration Analysis

Visco-Elastic Models

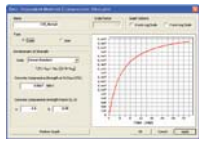
- Creep-Shrinkage (Design Code)
- Temperature-Dependent Material

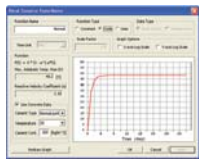
Heat Transfer

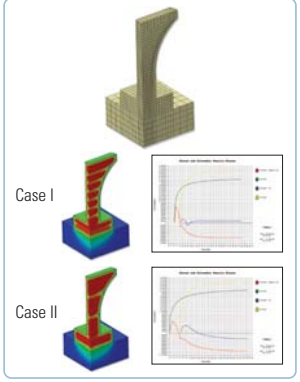
- Steady-State & Transient
- Conduction, Convection, Radiation
- Pipe Cooling

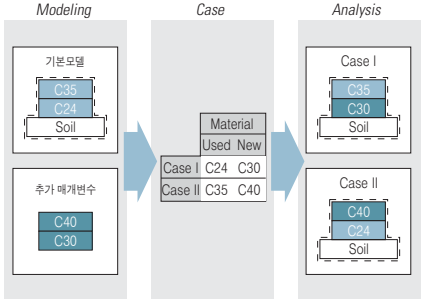
Parametric Analysis

- Multiple Material Sets
- Multiple BCs & Heat Sources
- Multiple Construction Sequences

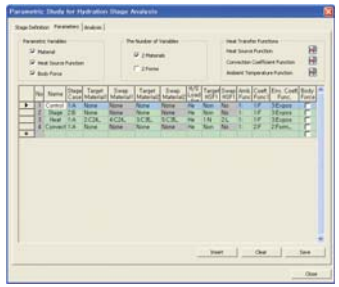

Creep/Shrinkage Function


Heat Source Function





Process of Parametric Analysis


Parametric Analysis Control

- Heat transfer analysis by convection, conduction of the concrete hydration heat and Thermal stress analysis which considers creep/dry shrinkage
- Perform creep/dry shrinkage and heat transfer analysis : heat of hydration and thermal coefficient
- Use general data to perform a parametric analysis
- Model high order elements to obtain accurate results (output result : nodal temperature, nodal displacement, element stress/strain, heat flux, heat flow, temperature gradient display, crack index)

midas FEA

"Advanced Nonlinear and Detail Analysis System"

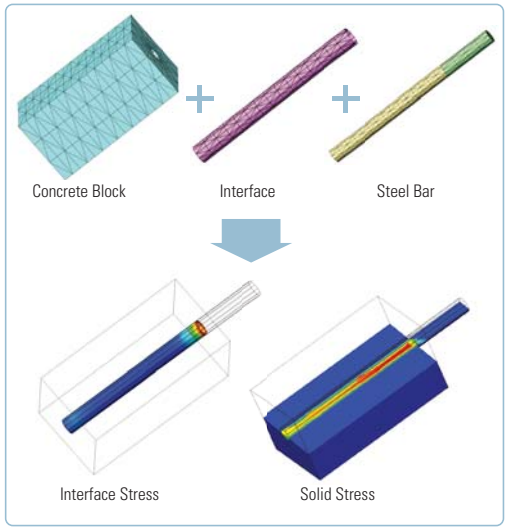
03. Analysis

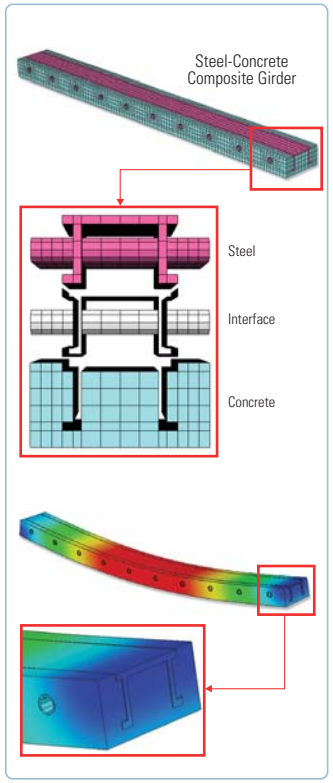
Providing high-end linear/nonlinear analysis
capabilities optimized for bridge and structural engineering

Interface Nonlinear Analysis

Interface Models

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





- Simulate frictional behavior for heterogeneous material such as reinforced concrete, etc.
- Provides 5 interface models to describe nonlinear behavior of heterogeneous material at contact surfaces
- Interface Model: Coulomb Friction, Discrete Cracking, Crack Diatancy, Bond-slip,Combined (Cracking-shearing-Crushing)

Static Contact Analysis

Contact Types

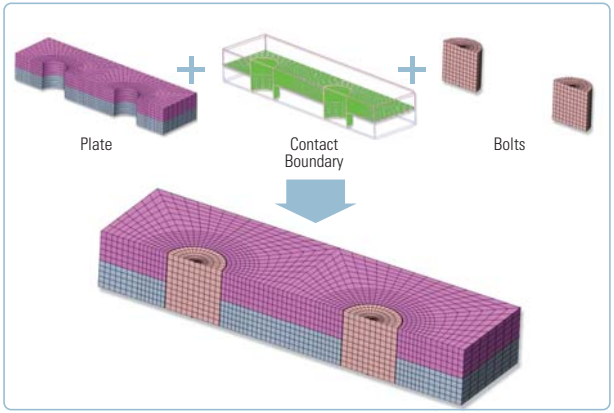
- Weld Contact
- General Contact

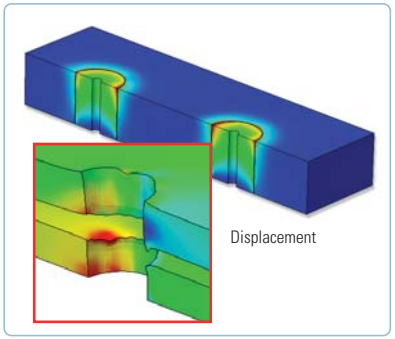
Behaviors

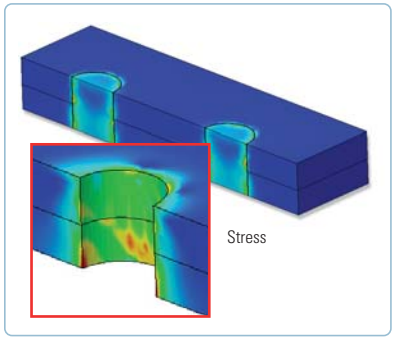
- Material Nonlinearity
- Geometry Nonlinearity

Results

- Displacement
- Stress
- Contact Force







- Perform contact analysis for structural parts (joint & connections) to simulate connections, cyclic loadings and contact frequency (Considering linear and nonlinear material models)
- Penalty method assigned for springs between contacting surface and node

midas FEA

"Advanced Nonlinear and Detail Analysis System"

03. Analysis

Providing high-end linear/nonlinear analysis
capabilities optimized for bridge and structural engineering

Fatigue Analysis

Methods and Parameters

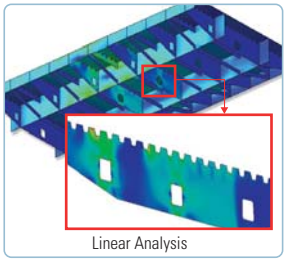
- S-N Method (Stress-Life)
- Load / Stress History
- Rainflow Counting
- Mean Stress Corrections
- Stress Concentration Factor
- Modifying Factors

Calculation Objects

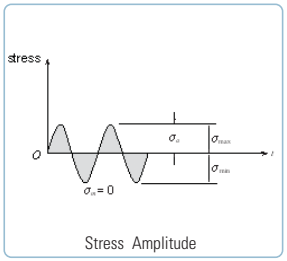
- Boundary Nodes Only (Default)
- Nodes of Selected Mesh Sets

Results

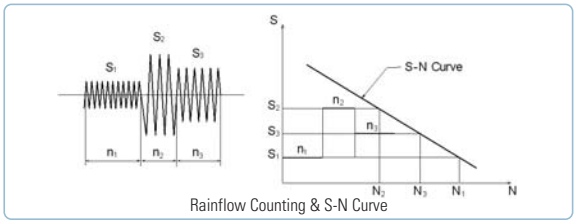
- Cycles to Failure
- Safety Factor
  (Cycles to Failure / Desired Repetition)



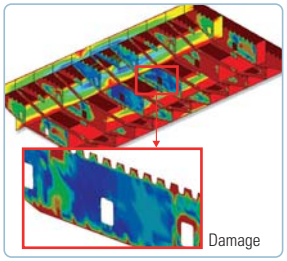
Linear Analysis



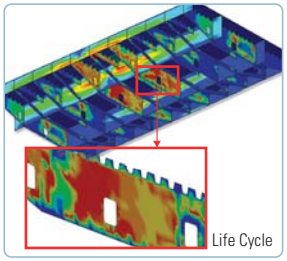
Stress Amplitude



Rainflow Counting & S-N Curve



Damage



Life Cycle

- Fatigue Analysis : Display structure failure phenomenon based on a cyclic loading pattern with reduced yield strength
- Calculate stress amplitude → Applying Rainflow Counting & S-N Curve → Calculating the fatigue life (life cycle) and extent of damage

CFD (Computational Fluid Dynamics) Analysis

CFD Models

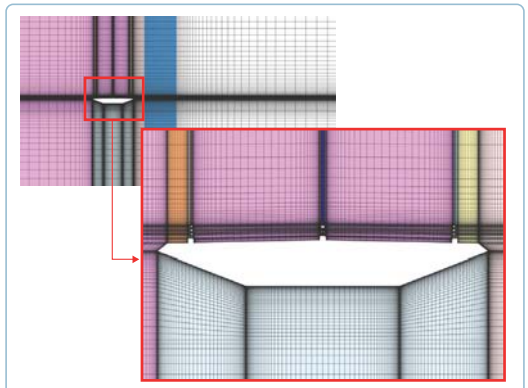
- Turbulence Models
  (RANS, k-ω, q-ω)
- Compressible Flow
- Incompressible Flow
- Inviscid Flow
- Unsteady Flow

Discretization Scheme

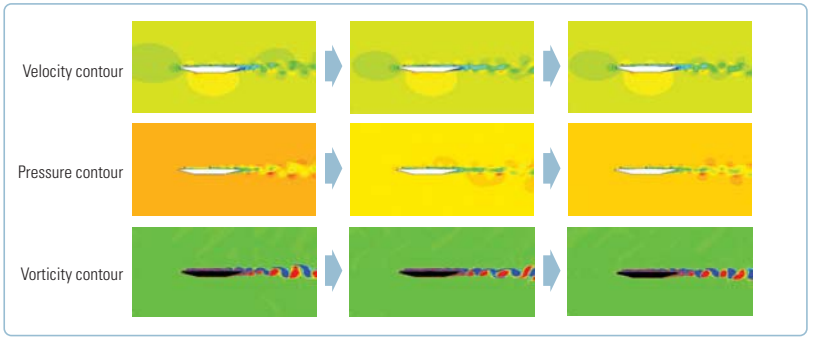
- 2nd-order (Spatial)
- Dual time stepping (Temporal)

Boundary Condition

- Far-field
- Wall (Slip, Non-slip), etc.



CFD model= Bridge Section

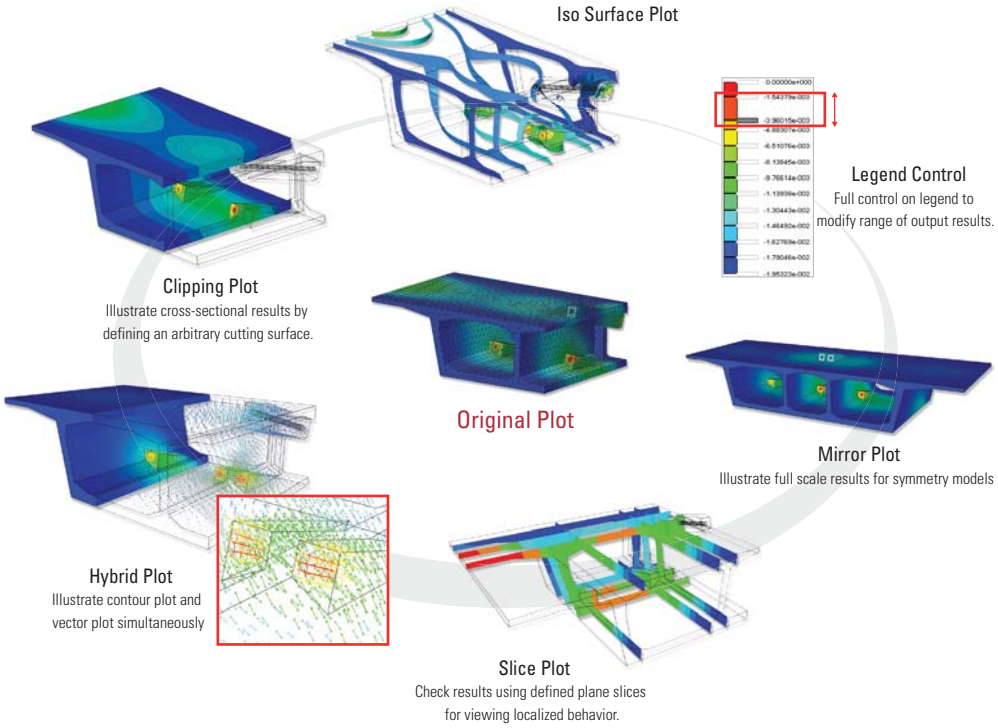


- Investigate bridge's stability resistance to wind by visually determining airflow pattern
- Perform Steady state analysis / Unsteady analysis and Laminar Flow / Turbulence Flow analysis
- Calculate aerodynamic force coefficients and determine pressure distribution & turbulence kinetic energy

04. Post Processor

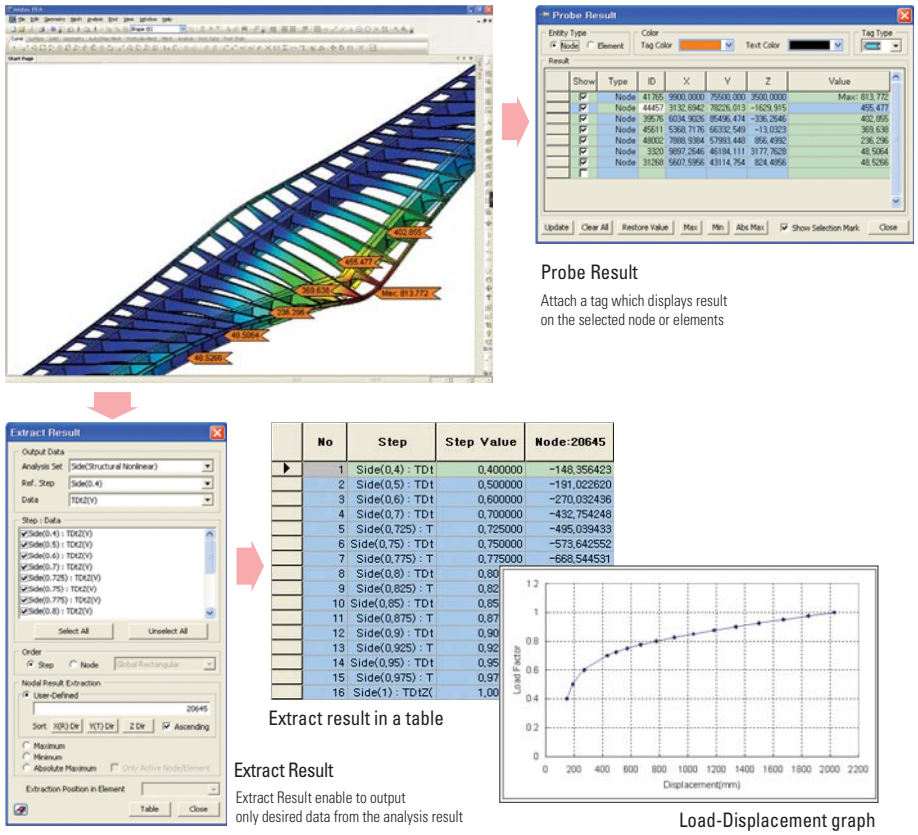
Providing practical output functions and
effective result visualization for analyzing structure behavior

Type of Plots / Interactive Legend Control



- Intuitive graphical output functions to verify and illustrate the analysis results
- Control range of contour or color spectrum with Legend Control function

Extract Result / Probe Result



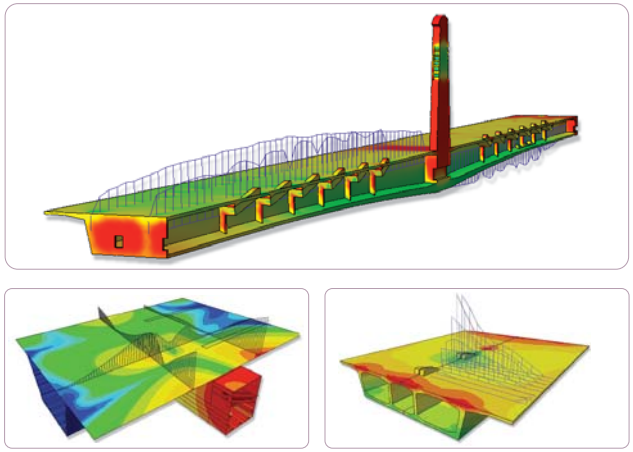
- Extract Results : Extract results directly from nodes or elements for all the stages
- Probe Results : Display tags on nodes or elements with corresponding results
- Easily filter Min or Max results using Probe Results

04. Post Processor

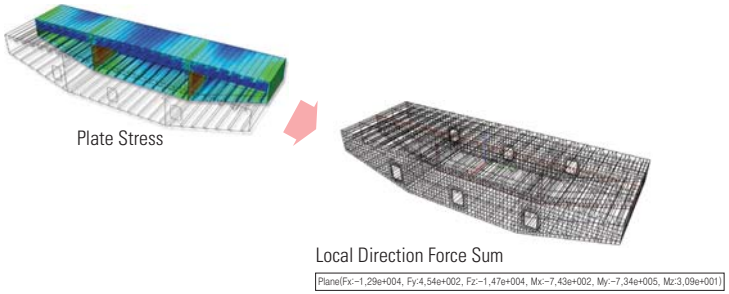
Providing practical output functions and
effective result visualization for analyzing structure behavior

On Curve Diagram / Local Direction Forcce Sum

On Curve Diagram



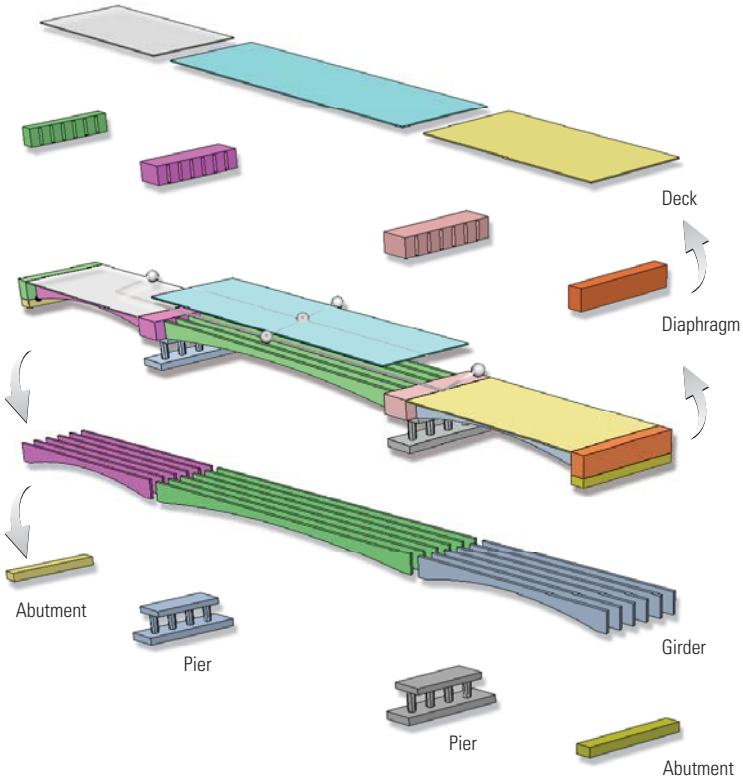
Local Direction Force Sum



- Extract results from two user-defined points or curve to display output diagram and table
- Local Direction Force Sum – Extract member force or moment results at specific section

Virtual Transformation

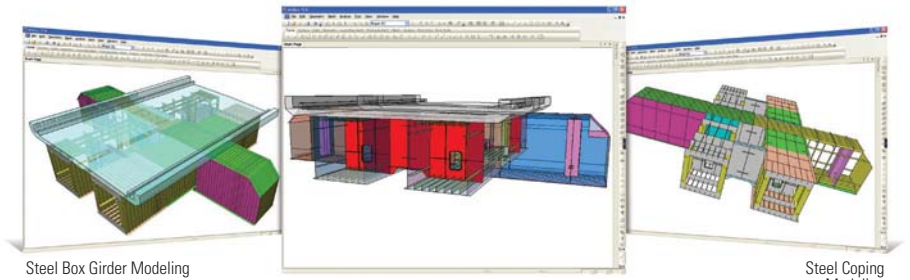
Integral Bridge



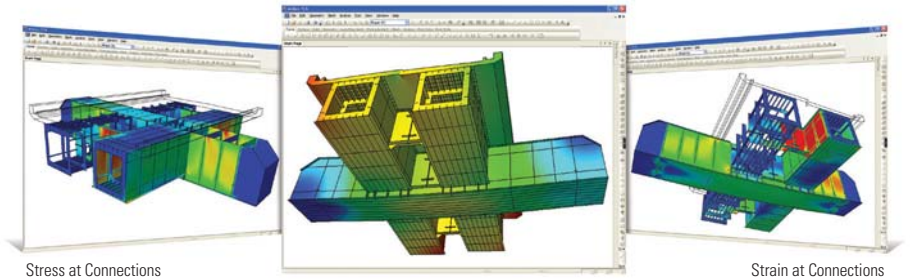
- Intuitive tools to investigate the interior parts of a complex model based on virtual transformation of mesh sets

Detailed Analysis of Steel Box Girder & Coping Connection

Geometry Modeling



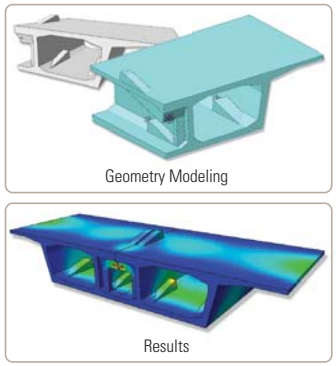
Analysis Results



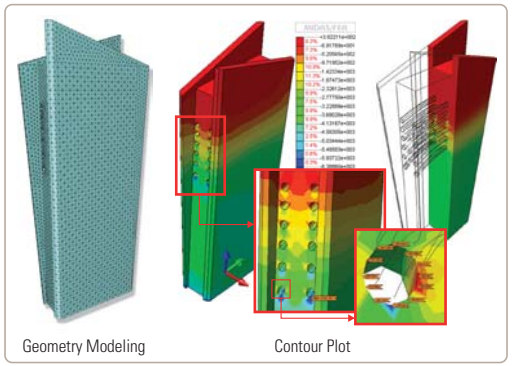
- Perform linear static analysis for connection between steel box girder and coping
- Review stress distribution resultant from bending moment and torsion at cross sectional diaphragms
- Defining rigid diaphragm nodes at cut end boundaries of the structure

Detailed analysis for concrete structures – Stress concentration analysis

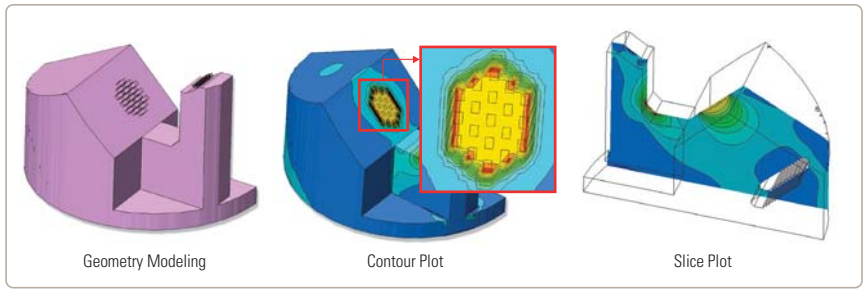
Detailed review of tendon anchorage



Detailed analysis effects for cables installed on bridge saddle



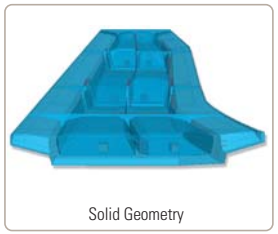
Detailed review of Anchorage block

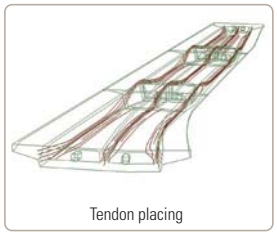


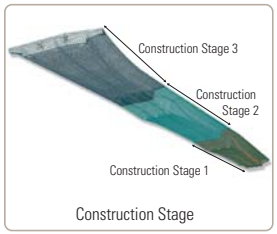
- Perform linear static analysis for stress concentration for anchorage block
- Review tensile stress and bursting stress for zones with high stress concentrations (tendon, cable anchor and suspension bridge saddle)

3D FEM Analysis for Long-span FSM Bridge

Geometry Modeling

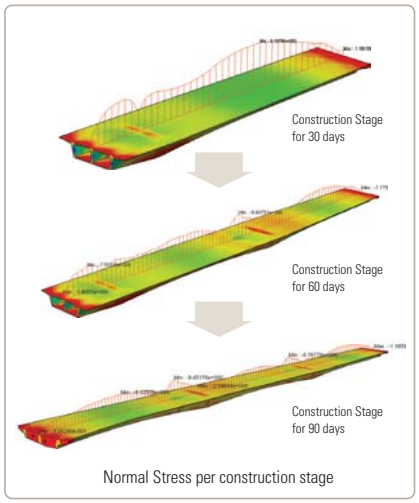

Solid Geometry


Tendon placing

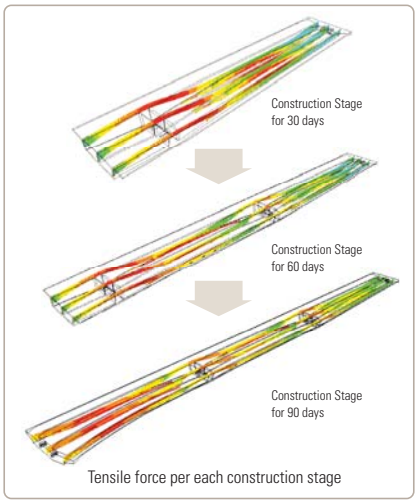


Construction Stage

Construction Stage Analysis



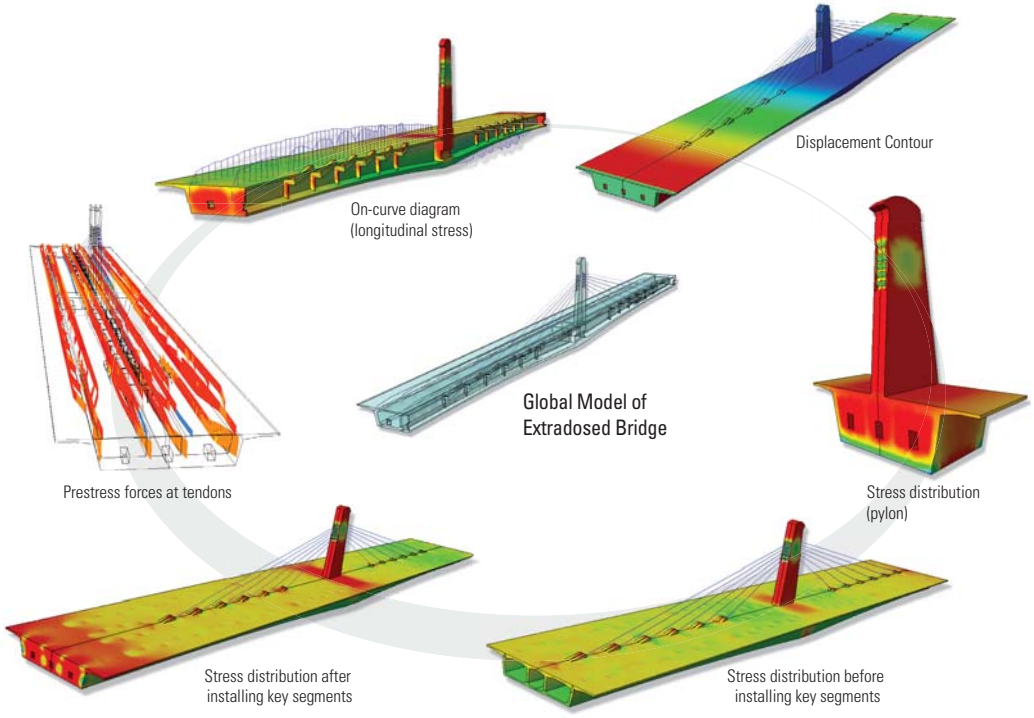
Normal Stress per construction stage



Tensile force per each construction stage

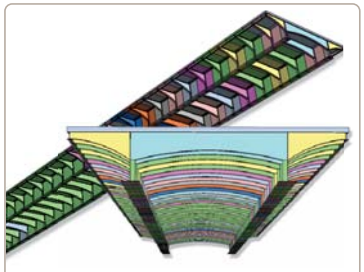
- Perform 3D FEM to verify the stability of a long span FSM bridge (over 70m long) with varying geometry dimensions at cross-section and top deck width
- Determine stress variations and tendon prestress distribution using construction stage analysis at critical sections
- Obtain accurate reaction distribution results at bearing points compared to frame analysis

Detailed Behavior for 1 pylon Extradosed Bridge

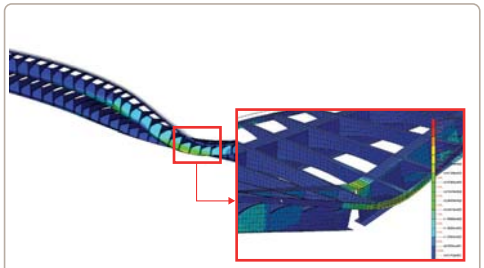


- Verify transversely distributed forces of one pylon extradosed bridge
- Verify detailed behavior of the FCM bridge using construction stage analysis during installation of key segment parts

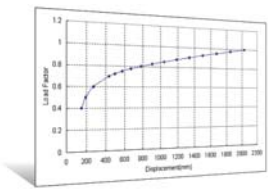
Reviewing the Redundancy for Plate Girder System



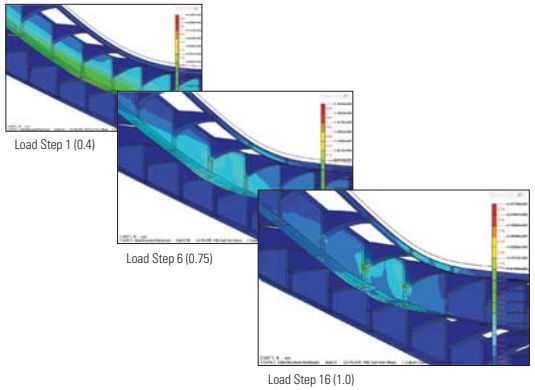
Detail model of Plate Girder system



von Mises Stress Plasticity



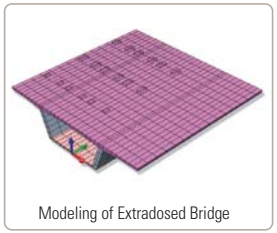
Max Deformation vs. Load Increment Curve



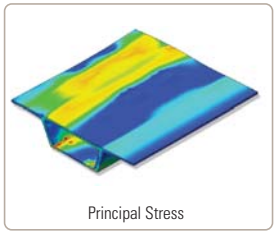
von Mises Stress Shape with Load Increment

- Model the redundancy effects on a plate girder bridge
- Simulate nonlinear behavior of the model using von-Mises constitutive model and Newton-Raphson iterative method
- Monitor failure mechanism based on ultimate state of nonlinear material models

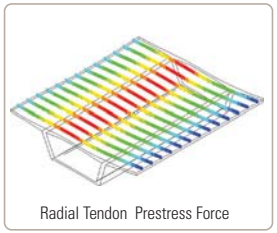
Load Carrying Capacity Evaluation for PSC Box Bridge



Modeling of Extradosed Bridge

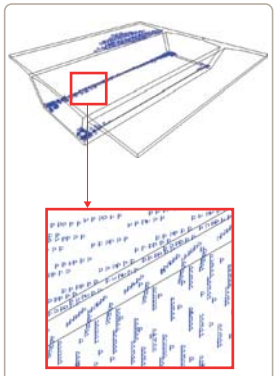


Principal Stress

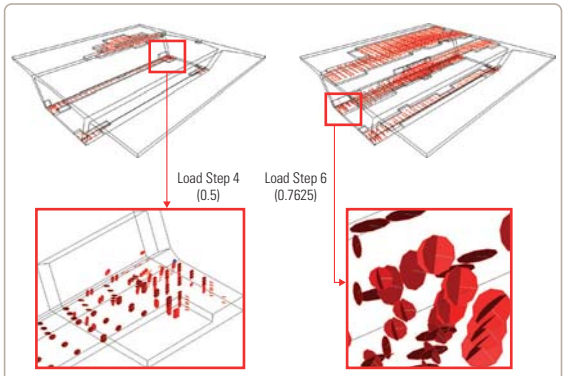


Radial Tendon Prestress Force

Crack Analysis



Crack Status

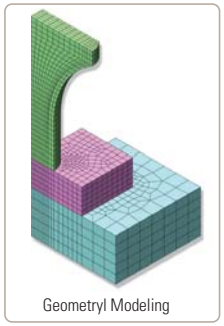


Crack pattern per load step

- Behaviorial characteristic results different from elastic analysis can be caused by material nonlinearity in the case of actual structure in the extreme conditions
- Evaluating stress condition considering the nonlinearity of the ultimate condition
- Determining crack pattern and status for concrete crack analysis applying TSC (Total Strain Crack) model

Application for Heat of Hydration Analysis Using Parametric Analaysis Feature

Selecting Hydration Analysis Model and Parameter

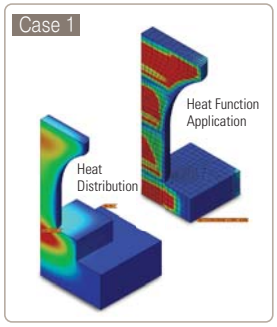

Geometryl Modeling

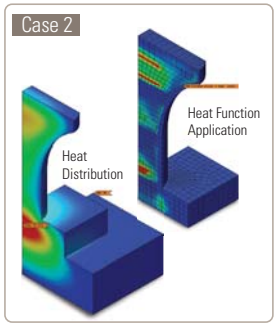
Case	Construction Stage	Heat Function	Convection Coefficient
1	Casting in 4 stages	Normal Portland	Steel
2	Casting in 9 stages	Normal Portland	Steel
3	Casting in 4 stages	Normal → low temperature	Steel

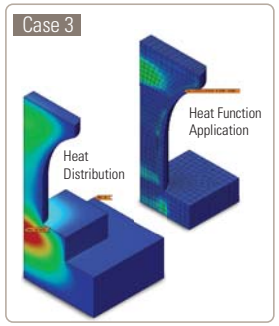
Result

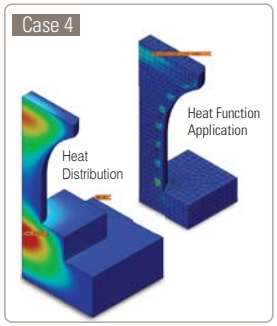
Case	Temperature crack ratio	Heat time function
1	0.2575	390hr
2	0.2982	490hr
3	0.8102	320hr

Analysis Result for each Parameter





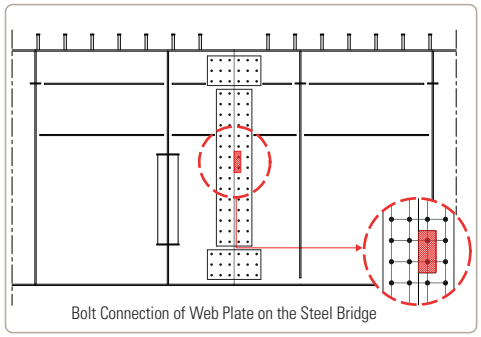


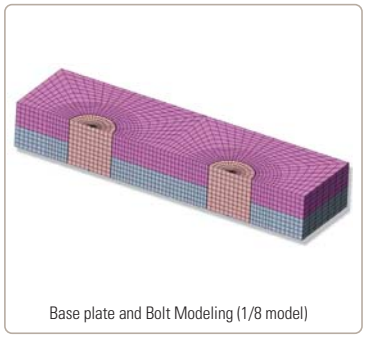


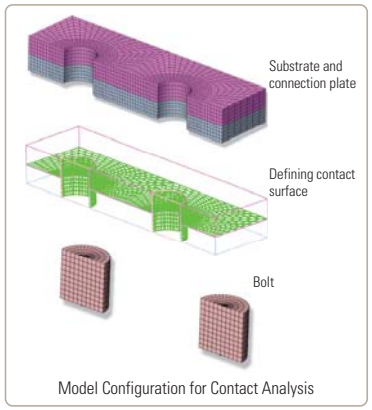
- Perform analysis consisting the cases for various factors (Casting, Convection boundary, material. Thermal characteristics, Self-weight) for controlling heat of hydration effect in a model
- Select casting height and material parameters through parametric analysis for finding crack index range

Contact Analysis of Bolt Connection

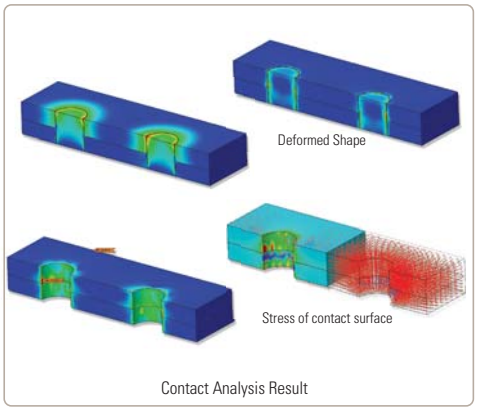
Overview of Analysis model


Bolt Connection of Web Plate on the Steel Bridge


Base plate and Bolt Modeling (1/8 model)



Model Configuration for Contact Analysis

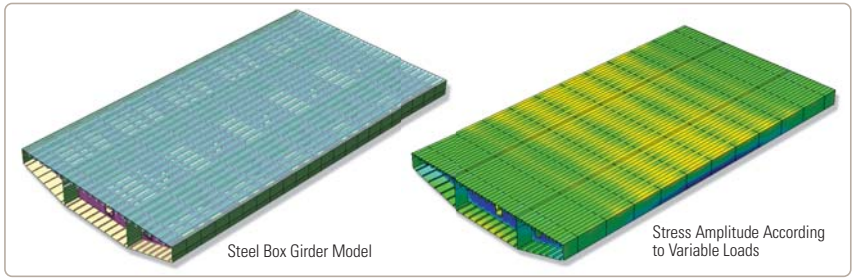


Contact Analysis Result

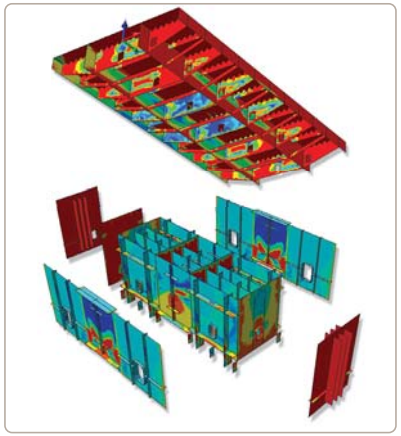
- Define contact surface between base plate, gusset plate and bolts to transfer contact forces
- Perform nonlinear analysis considering contact condition between members
- Penalty method is applied in order to intercept the interaction of the contact surface based on spring elements

Fatigue Analysis for Steel Bridge

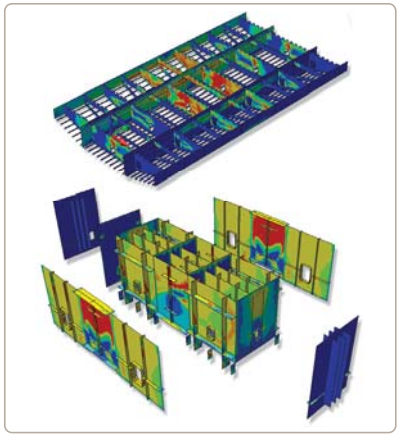
Fatigue Analysis



Life Cycle



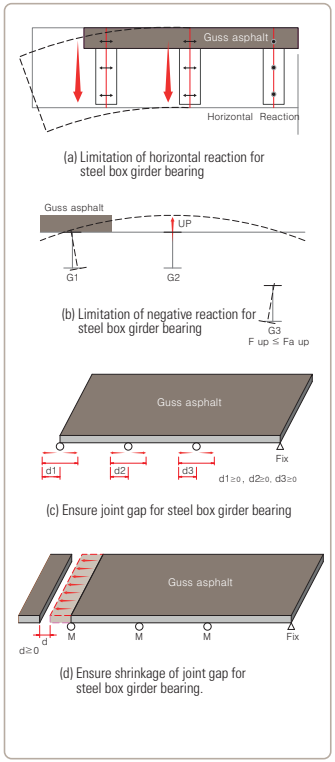
Damage



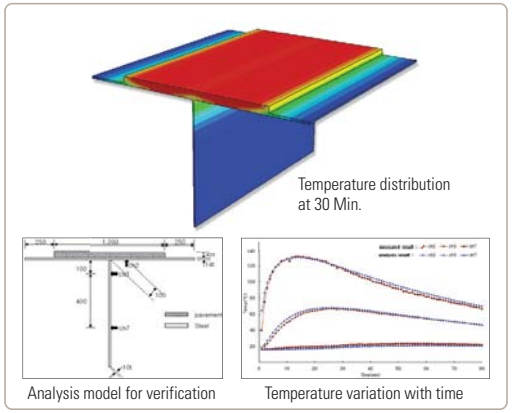
- Perform static analysis on structures with applied static loads and live loads to calculate the amplitude stress
- Setting maximum amplitude stress for S-N Curve of steel box and limitation of amplitude stress
- Calculate fatigue life(fatigue iteration) and failure considering the influence by Mean stress correction

Thermal Stress Examination According to the Guss Asphalt Construction

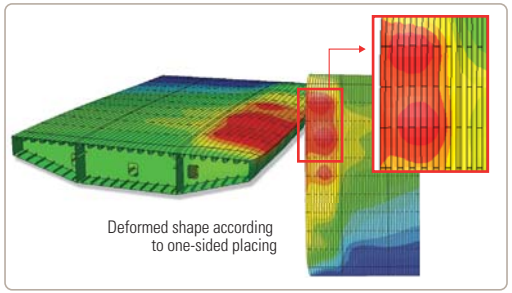
Heat Deformation of Steel Box Girder



Heat Transfer Analysis



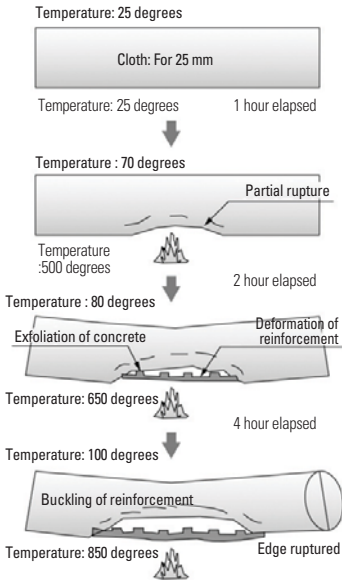
Heat Stress Analysis



- When constructing Guss asphalt, the section of steel box temporarily generates high temperature difference
- When constructing the previous Guss asphalt, Problems arising at expansion joints or damage cases for the cross clamps depending on the vertical direction of the displacement
- Preparation for effective construction plan through the thermal stress examination before the Guss asphalt construction

Subway station and fire analysis of tunnel

Fire Analysis Concept



Fire Analysis Process

Basic section design

Section design of general external forces by strength design method

Heat transfer analysis

Heat transfer analysis and calculation of temperature distribution from the demanded fire time

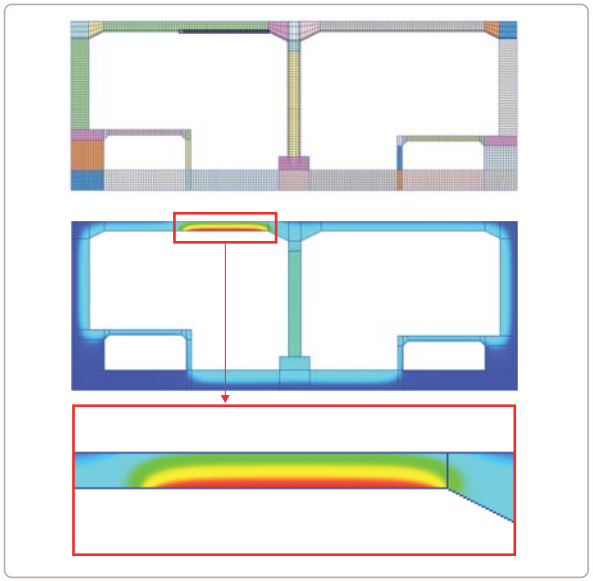
Strength reduction computation

Temperature dependent strength computation (fckΘ, fyΘ)

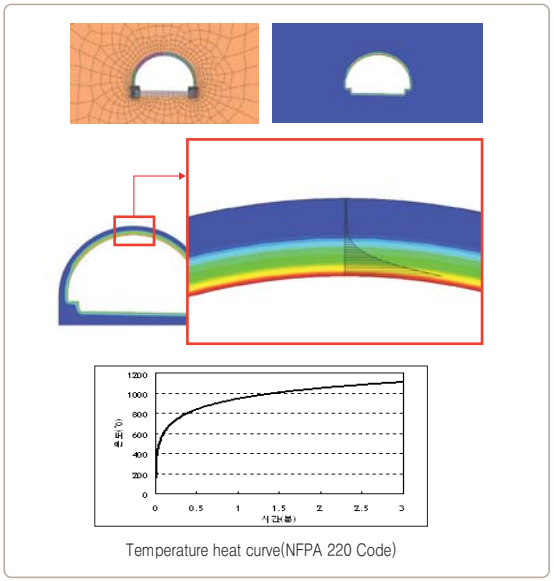
Section reexamination

MnΘ calculation & examination in demanded fire time
(+)MnΘ ≥ (+)Mmax (service load)

Heat Transfer Analysis of Subway Station



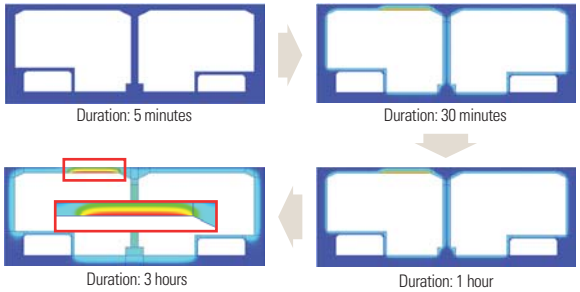
Heat Transfer Analysis of Tunnel Fire



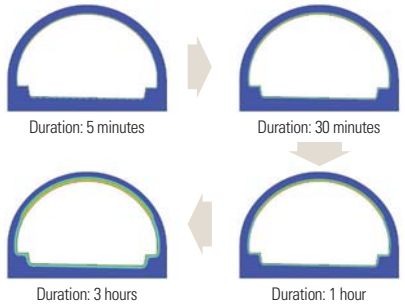
Temperature heat curve(NFPA 220 Code)

- Simulate and analyze fire exposure for underground structure based on reference code
- Perform heat transfer analysis using temperature dependent material properties
- Estimating the safety factor for damaged sections exposed to fire

Temperature distribution of subway station by time frame



Temperature distribution of tunnel by time frame



midas FEA

Advanced Nonlinear and Detail Analysis System

www.MidasUser.com