midas Civil
Advanced Webinar

- Date: December 6th, 2011
- Topic: General Use of midas Civil
- Presenter: Nithil Malguri

Bridging Your Innovations to Realities

midas Civil
Contents:

Overview
Modeling
Boundary Conditions
Loading
Analysis Results
Design and Misc.
midas Civil is 3D structural analysis and design software based on FEM analysis.

### Structure Types

<table>
<thead>
<tr>
<th>Application field</th>
<th>Sub-category</th>
<th>Remark</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Conventional bridge</strong></td>
<td>Slab bridge</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Culvert</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Frame bridge</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Arch bridge</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Integral bridge</td>
<td></td>
</tr>
<tr>
<td><strong>Girder bridge</strong></td>
<td>Composite precast girder bridge</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Composite steel plate girder bridge</td>
<td></td>
</tr>
<tr>
<td><strong>Segmental post-tensioned box girder bridge</strong></td>
<td>Staged construction considering creep &amp; shrinkage</td>
<td></td>
</tr>
<tr>
<td></td>
<td>FSM (Single cell, multi-cell)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>FCM or BCM (Free Cantilever Method or Balanced Cantilever Method)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>MSS (= Movable Scaffolding System)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>ILM (= Incremental Launching Method)</td>
<td></td>
</tr>
<tr>
<td><strong>Cable bridge</strong></td>
<td>Cable-stayed bridge</td>
<td></td>
</tr>
<tr>
<td></td>
<td>- Cable installation force optimization</td>
<td></td>
</tr>
<tr>
<td></td>
<td>- Concrete cable-stayed bridge</td>
<td></td>
</tr>
<tr>
<td></td>
<td>- Steel cable-stayed bridge</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Suspension bridge</td>
<td></td>
</tr>
<tr>
<td></td>
<td>- Initial configuration analysis</td>
<td></td>
</tr>
<tr>
<td></td>
<td>- Earth-anchored suspension bridge</td>
<td></td>
</tr>
<tr>
<td></td>
<td>- Self-anchored suspension bridge</td>
<td></td>
</tr>
<tr>
<td><strong>Railway bridge</strong></td>
<td>Rail track-structure interaction</td>
<td></td>
</tr>
<tr>
<td></td>
<td>High speed train - bridge dynamic interaction</td>
<td>Not available</td>
</tr>
</tbody>
</table>
## Structure Types

<table>
<thead>
<tr>
<th>Underground structure</th>
<th>Subway station</th>
<th>Purification plant</th>
</tr>
</thead>
<tbody>
<tr>
<td>Soil-structure interaction</td>
<td>Nonlinear static</td>
<td>Nonlinear time history analysis</td>
</tr>
<tr>
<td>Offshore structure</td>
<td>Wave load</td>
<td>Not available</td>
</tr>
<tr>
<td><strong>Seismic analysis</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Response spectrum analysis</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pushover analysis</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Inelastic time history analysis</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Nonlinear time history analysis with damper or isolator</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Multiple support excitation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Linear local buckling analysis of steel member</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Plastic analysis of connection (Mohr-Coulomb, von-Mises)</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Detailed analysis</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Heat of hydration analysis</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Prestressed box girder analysis with shell / solid elements</td>
<td>FEA</td>
<td></td>
</tr>
<tr>
<td>Cracking analysis</td>
<td>FEA</td>
<td></td>
</tr>
<tr>
<td>CFD analysis (Turbulence model)</td>
<td>FEA</td>
<td></td>
</tr>
<tr>
<td>Fatigue analysis</td>
<td>FEA</td>
<td></td>
</tr>
<tr>
<td>Interface nonlinear analysis</td>
<td>FEA</td>
<td></td>
</tr>
<tr>
<td>Contact analysis</td>
<td>FEA</td>
<td></td>
</tr>
</tbody>
</table>
2. Analysis Types

- Moving Load Analysis
- Construction Stage Analysis
- P-Delta Analysis
- Buckling Analysis
- Eigen value analysis
- Heat of Hydration analysis
- Geometry Non-Linear Analysis
- Material Non-Linear Analysis
- Pushover Analysis
- Time History Analysis
- Response Spectrum Analysis
3. Element Types

1- D Elements

- Truss
- Tension only
- Hook
- Cable
- Compression only
- Gap

2-D Elements

- Plate – 3 Node, 4 Node
- General/Tapered Beam
- Plane Stress
- Plane Strain
- Axisymmetric

3-D Elements

- Solid – 4 Node, 6 Node, 8 Node
midas civil provides the user with extensive graphic features which can be used for modeling and obtaining results very simply.
Model View
1. Bridge Wizards

- RC Bridge
- Grillage Model Wizard
- FCM Bridge
- FSM Bridge
- ILM Bridge
- MSS Bridge
- Suspension Bridge
- Cable Stayed Bridge
2. dxf import

Import dxf model
Analyze in midas civil
2. Dxf import

Important considerations:
- Polyline in dxf file will be imported as a beam element.
- Surface in dxf file will be imported as a plate element.
- Solid cannot be imported.
- Unit system must be consistent.

- **Line**
  - Import result: Beam Elements

- **Polyline Rectangle**
  - Import result: Plate Element

- **3-D Face**
  - Import result: Plate Element
3. Material Properties

The material properties from several codes can be defined:
- ACI
- Eurocode
- Canadian Code
- Indian Code
- BS code

- The Time Dependent Material for Creep, Shrinkage and Compressive Strength can be defined as per:
  - CEB-FIP (1990)
  - CEB-FIP (1978)
  - ACI
  - Combined ACI&PCA
  - AASHTO
  - INDIA (IRC:18-2000)
  - European
4. Section Properties

Midas civil equips the users with the following sections:

- Standard Sections
- Composite sections
- Combined sections
- SRC sections
- PSC Sections
- User Defined Section
4. Section Properties

Determination of Torsional moment of inertia and effective shear area

Fixed Support

Meshed Element (Length – 20m)
5. Create Elements

- Node to Node
- Grid
The Bridge having curved profile can be modeled using this feature:
The curve can be defined using the following:

- Arc by 3 points
- Arc by Centre and 2 Points
- Circle by 3 Points
- Circle by Centre and 2 Points
- Ellipse by Centre and 2 Points
- Parabolic Curve by 3 Points
- Cubic Curve by 4 Points
7. Extrude Elements

The extrusion can be done in the following three manner:
- Node -> Element
- Element -> Plate
- Plate -> Solid
8. Change Element Parameters

- Change Element parameters
- Element Table
- Works Tree
- Elements -> Change Element Parameters
9. Mass Data

**Structure Mass**
- Automated Program determines the mass of the structure considering the material density
- Additional Mass can be defined using Model -> Masses
  - Load To Masses
  - Nodal Masses
**1. Supports**

**USE:** To provide the support to the structures. Can be used to simulate pin and fix supports.

**TYPE:** The supports can be provided to X Y and Z directions. Also the rotations about these axes can be restrained.
2. Point Spring Supports

**USE:** Generally used to simulate the soil conditions (to simulate the non-stiff support provided by soil). Soil support to the footings can be modeled by this function.

**TYPE:** Four types of springs can be defined:
- Linear
- Comp – Only
- Tens – Only
- Multi-Linear
3. Surface Spring Supports

**USE:** Generally used to simulate the soil conditions (to simulate the non-stiff support provided by soil) underneath a footing. The subgrade modulus of soil obtained by geotechnical experiments can be directly entered and midas civil determines the equivalent spring constants to simulate the soil support.

**TYPE:** Four types of springs can be defined:
- **Nodal Spring** – Program automatically calculates the spring simulating the soil conditions and applies that support to nodes.
- **Distributed Spring** – The soil support is applied as a planar support.

Further type of element – beam, planar or solid can be selected.
4. Elastic Links

**USE:** Can be used to connect two member via springs. Especially used to simulate the bearing characteristics. The equivalent spring constants of bearings can be specified.
Can be used to simulate the ballast characteristics.

**TYPE:** Four types of springs can be defined:
- General
- Rigid
- Tension Only
- Compression Only
- Multi Linear
4. Elastic Link

Simulation of Ballast Characteristics

---

Elastic Link Data

- **Type**: Multi-Linear

| 1 | 0 | 0 |
| 2 | 0.002 | 60 |
| 3 | 0.01 | 60 |

- **Direction**: Dx

- **Shear Spring Location**: 

- **Dist. Ratio From End**: 0.5

- **Beta Angle**: 0°
5. Beam End Releases

**USE:** To release specific degree of freedom for a specific point of time. The released degree of freedom can be assigned to a boundary group and can be activated and deactivated at users will. Especially useful in composite bridge construction analysis where the beam becomes continuous after the laying of deck.

**TYPE:** Fz, Fy, Fx, Mx, My, Mz can be released.
6. Rigid Link

**USE:** To connect the nodes which cannot move relative to each other. Especially used to connect the nodes representing the same solid.

**TYPE:** The rigid link can be provided to restrict DX, DY, DZ, RX, RY, RZ.
7. Node Local Axis

**USE:** To define a local coordinate system for the definition of boundary conditions or produce output for reactions

**TYPE:** The local Axis can be defined for x, y and z (local) axis separately.
8. Effective Width Scale Factor

**USE:** A scale factor for moment of inertia ($I_{yy}$) about $y$-axis can be applied to check sectional stresses reflecting an effective width. The scale factors are applied to stress calculations which can be further used to check the maximum stress limit from code.

**TYPE:** The scale factor can be applied separately to $I_y$ and distance of top and bottom fiber from the neutral axis.

Effective Width scale factors for **AASHTO code** can be *automatically* specified. (Model -> Structure Wizard -> PSC Bridge)
1. Static Load Cases

**USE:** The load case can be specified so that the program can automatically recognize what factor has to be applied to the loads for generating the load combinations for AASHTO, Eurocode etc.

**TYPE:** The following type of static load cases are frequently used:

- Dead Load
- Dead Load of Components and Attachments
- Dead Load of Wearing Surface
- Live Load
- Prestress
- Creep
- Shrinkage
- Temperature
- Temperature Gradient
2. Self Weight

**USE:** The self weight of the structure is automatically considered when the element is activated.

**TYPE:** The self weight can be applied to X, Y and Z direction.
3. Specified Displacement of Supports

**USE:** The support settlement can be specified with the help of this function. Particularly useful when the structures are built on soils that can settle. Also differential settlements of piers can be simulated.

**TYPE:** The specified displacements can be provided in x, y and z directions. Also the rotations can also be specified.

\[ D_z = 1 \text{ mm} \quad \quad D_z = 2 \text{ mm} \]
3. Specified Displacement of Supports

Specified displacement = 10mm
3. Specified Displacement of Supports
4. Settlement Analysis Data

**USE:** Midas Civil automatically generates the combination of settlements.

**TYPE:** The specified displacements can be provided in x, y and z directions. Also the rotations can also be specified.

Dz = 5mm

Dz = 8mm

Dz = 10 mm

Dz = 10 mm
5. Beam Load

**USE:** For specifying the regular beam forces such as concentrated forces, uniformly distributed forces etc.

**TYPE:** The following types of beam loads can be specified:
- Element beam Loads – for each element
- Line beam Loads – for a set of elements
- Typical Beam Loads – Special Distribution
5. Beam Loads – Uniform Pressure (Element)

**USE:** For specifying the lateral load (wind) on the beam elements. It obviates the need of calculating the force on individual elements. Also used to determine the wind load effect on the noise barriers.

**TYPE:** Rectangular and Trapezoidal pressure distribution can be applied.
6. Plane Load

**USE:** For specifying the pressure load which doesn’t cover the entire plate.

**TYPE:** Rectangular and Trapezoidal pressure distribution can be applied.
**7. Temperature Load**

**USE:** To simulate the stresses obtained due to the change in temperature.

**TYPE:** The following type of temperature loads can be specified:
- System Temperature – for applying the change in temperature to whole structure.
- Nodal Temperate – for applying change in temperature to certain nodes.
- Element Temperature – for applying change in temperature to certain elements.
- Temperature Gradient – for applying change in temperature to beam and plate sections.
- Beam Section Temperature – for applying temperature gradient to beam sections.
The width of the section is automatically considered for the calculation of thermal stresses.
Midas Civil automatically considers the self restraint moment obtained in the section due to differential temperature.

Even if the resultant force or bending moment is 0, thermal stress may be expected.
8. Moving Load

**USE:** For obtaining the critical force effects due to movement of vehicles on the bridge.

**TYPE:** Midas Civil has the database for the following codes:
- AASHTO Standard
- AASHTO LRFD
- Eurocode
- INDIAN code
- Canadian Code
- PENNDOT
- BS Code

User defined vehicles can also be specified and used for moving load analysis.
** 8. Moving Load

AASHTO LRFD Section 3.6.1.3.1

- For both negative moment between points of contraflexure under a uniform load on all spans, and reaction at interior piers only, 90 percent of the effect of two design trucks spaced a minimum of 15,000 mm between the lead axle of one truck and the rear axle of the other truck, combined with 90 percent of the effect of the design lane load. The distance between the 145,000-N axles of each truck shall be taken as 4300 mm.
8. Moving Load

Lane Element

Cross Beam

Reference element
9. Hydrostatic Load

**USE:** To simulate the hydrostatic or earth pressure load in the lateral direction.

**TYPE:** The user can select the linear or curved type of variation. The user can also specify the constant intensity load.
8. Moving Load

Train Lane

Vehicles Lane
USE: To see the force effect due to a combination of loads. Also for defining combinations for strength and service limit states.

TYPE: The load combinations can be made considering the loads Activated in the stages and specifying factors for the same. The load combinations can be automatically generated for the following Codes:

- AASHTO LRFD 07
- AASHTO Standard 2000
- AASHTO LRFD 02
- Indian Code
- Eurocode
- Taiwan Code
The load combination can be generated considering the construction stage loads also.
9. Buckling Analysis

Using this option the user can
Select the range of loads to be used
For the determination of mode shape.

**Positive:** Force in direction of gravity
will be considered

**Search:** The force can be applied
parallel to gravity direction or anti parallel to it

The loads can be applied as constant or varying
Particularly useful to define the self weight as
constant and other forces as varying

The user can select whether
the Lateral Torsional Buckling
load has to be considered or not
Thank You!

For more information, please visit us at http://en.midasuser.com