MIDAS Academy

Transverse Analysis

Part 1 : Model Summary

TaeYong Yu
Technical Engineer
MIDASIT
1. Introduction
   • Bridge Type
   • Analysis Type

2. Geometry
   • Overview of Target
   • Overview of Modelling method
   • Overview of Modelling plan
   • Modelling steps & used options

3. Loads
   • Design code & Load combination
     • Loads

4. Boundary Conditions

5. Post Processing
1. Introduction

Bridge Type

- Structure Type: Prestressed Concrete Girder in Cable Stayed Bridge
- Bridge Information
  - Span: 175 + 420 + 175 = 770m
  - Superstructure: PSC girder
  - Pylon & Pier: Concrete structure

Fig 1. Longitudinal & Section View
1. Introduction

**Analysis Type**

- **Analysis Type**: Transverse Analysis of Girder in the preliminary design step
- **Purpose**:
  1. Check the capability of a girder in the transverse direction.
  2. Check reinforcements, tendons and the shape of the girder.

---

**Fig 2. Plan & Cross-Section View**
2. Geometry

- **Analysis Type**: Transverse Analysis of Girder in the preliminary design step.
- **Check List**: 1) Stresses on the upper slab under SLS load combination.
  2) Bending moment, shear on the upper & bottom slab and the web under ULS load combination.

Fig 3. Stress & Section Check

Stress Distribution
- Top of Slab
- Bottom of Slab

PSC
- Check M, V in PSC member

RC
- Check M, V in RC member
2. Geometry

Overview of Modelling method

- Girder Modelling; 2D (Frame element) vs 3D (Plate element)

Case 1) By using beam elements

Case 2) By using plate or solid elements
2. Geometry

Overview of Modelling method

- Girder Modelling; 3D model was used.

**Reasons**

1) In the simplified method, the vehicle load can be considered as the uniform distribution load at the 2D model that is generated with frame elements. However, the vehicle load can be overestimated in this method. It makes differences between the analysis model and real model.

2) We can model the box girder as the real model under the 3D model. It allows us to consider complex behavior in 3D. Especially the box girder shows 3D behavior strongly.

3) In order to do effective design, the 3D model method was selected.

---

<table>
<thead>
<tr>
<th>In the beam analysis</th>
<th>At The Inner Slab</th>
<th>At The Cantilever</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>( M_1 )</td>
<td>( M_2 )</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>In the plate analysis</th>
<th>At The Inner Slab</th>
<th>At The Cantilever</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>( M_{11} )</td>
<td>( M_{22} )</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Calculation of UDL width(( \text{Type1, 2} ))</th>
<th>( E_1' = \frac{M_1}{M_1'} )</th>
<th>( E_2' = \frac{M_2}{M_2'} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( E_1 = \frac{9.6L}{4(L+0.6)} )</td>
<td>( E_2 = 0.8X+1.14 )</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Selecting the width</th>
<th>( \min(E_1', E_1') )</th>
<th>( \min(E_2', E_2') )</th>
</tr>
</thead>
</table>

**a) Simplified Method for Vehicle Load**

**b) 3D model with Vehicle load**

---

Fig 5. Vehicle load on 2D&3D model
2. Geometry

Overview of Target Structure

- The girder contains a normal section, rib and cable anchorage.
- Total 15 segments were created to reduce the poisson’s effect.
- Length of the model: Total 55.0m (3.75m per one segment)

Fig 6. Longitudinal View of Target Model
2. Geometry

Overview of Target Structure

- Three typical sections were used for checking.

a) Detailed longitudinal view

b) Section A-A (Normal section)

c) Section B-B (At Rib)

d) Section C-C (At Cable Anchorage)

Fig 7. Overview of Cross Sections

- Three typical sections were used for checking.
2. Geometry

Overview of Modeling Plan

- Plan for mesh generation

  1) Cross-Section View
  - The cross-section of a girder was divided with several nodes.
  - The location of nodes is planned to consider where results should be checked.

Fig 8. The location for checking results
2. Geometry

Overview of Modeling Plan

- Plan for mesh generation
  1) Cross-Section View
     For creating nodes. Two cases are available.
     a) Case 1; Using the offset length from the standard line to create nodes.
     b) Case 2; Using the center of divided section to create nodes.

Fig 9. 2 Ways for element lines

a) Case 1; Offset

b) Case 2; Center
2. Geometry

Overview of Modeling Plan

- Plan for mesh generation
  1) Cross-Section View
     - In order to get the same level of elements on the upper slab, I chose the case 1.

Fig 10. Comparison of 2 ways

a) Case 1; Normal Section & At Rib

Not same level

a) Case 2; Normal Section & At Rib

Fig 10. Comparison of 2 ways
2. Geometry

Overview of Modeling Plan

- Plan for mesh generation

  1) Cross-Section View
      - 3 Types of the section have the same element lines.

Fig 11. Plan for mesh generation in cross-section view
2. Geometry

Overview of Modeling Plan

- Plan for mesh generation

2) Plan View

- In order to make a plan of elements, some factors are considered as shown below.
- The location of tendons, ribs and cable anchorages.

Fig 12. Plan view of the girder
2. Geometry

Overview of Modeling Plan

- Plan for mesh generation

2) Plan View
   - The length of elements is planned considering divided position in the longitudinal direction.
2. Geometry

Modeling steps & Used options

- Modeling Steps

Step 1. Prepare *.dxf.

Step 2. Import *.dxf into midasCivil.

Step 3. Extrude beam elements to plate elements.

Step 4. Copy one segment of the girder.

Fig 14. Modeling Steps
2. Geometry

Modeling steps & Used options

• Additional modeling in midasCivil

1) Offset option for plate elements
   - All plate elements have the thickness property to input their stiffness.
   - Offset option in the thickness property was used to consider various thickness of the girder section.

Fig 15. Offset option for plate elements
2. Geometry

Modeling steps & Used options

- Additional modeling in midasCivil

2) Dummy beam elements
   - The dummy beam elements were used to apply tendon profiles.
   - It was restrained with nodes on plate elements.
   - The section of dummy beam was assumed as the small rectangle.

Fig 16. Dummy beam elements
2. Geometry

Modeling steps & Used options

- Additional modeling in midasCivil

3) Elements for cable

- Cable & Truss element are available in midasCivil for the cable structure.
- The truss element was used. So, cable forces were only considered.
- The coordinates of cable anchorages in the drawing was used for nodes of truss elements.

Fig 17. Elements for Cable
3. Loads

Overview of Loads

- Considered Conditions
  - Girder body
  - Barriers, Pavement and Additional attachment.
  - Prestress
  - Wind
  - Temperature
  - Vehicles
  - Cable Force

Fig 18. Load Conditions
3. Loads

Overview of Loads

- Load Cases
  1) Girder Body > Self Weight
     - **Self Weight function** was used.
  2) Barriers, Pavement & Attachment > DC & DW
     - **Plane Loads function** was used.
     - Plane Loads can be applied with coordinates of loads regardless the position of nodes.

Fig 19. Self Weight & Superimposed Loads
3. Loads

Overview of Loads

- Load Cases

3) Wind
- The wind load on the barrier was considered with Nodal Load function because barriers weren’t modeled.
- The wind load on the side web of the girder was considered with Pressure Load function.
- The wind load was considered on the left and right side of a girder.

Fig 20. Wind Loads
3. Loads

Overview of Loads

- Load Cases

4) Temperature
   - Element Temperature function was used.
   - Uniform temperature was considered.

Fig 21. Temperature Loads

a. Heating(+20)  b. Cooling(-30)
3. Loads

Overview of Loads

- **Load Cases**

  5) Prestress
  - **Tendon Prestress Load function** was used.
  - **Tendon Profile function** was used to check the effect of profile.
  - Tendon Profile can’t be created on the plate element. Therefore, dummy beam elements were added on the plate elements and these shared nodes.

Fig 22. Tendon Prestress Load
3. Loads

Overview of Loads

- Load Cases

6) Vehicle Load
   - **Plane loads function** was used.
   - The truck load was used.
   - All of wheels in a truck was considered. The concentrated load of wheel was replaced as the distribution load of wheel.

![Vehicle Type & Wheel Load](image)

Fig 23. Vehicle Type & Wheel Load
3. Loads

Overview of Loads

- Load Cases

6) Vehicle Load
   - The location of a vehicle in the longitudinal direction is selected by considering the influence diagram.
   - Cable anchorages were considered as the support in the longitudinal direction.

Fig 24. The influence diagram in the longitudinal direction
3. Loads

Overview of Loads

- Load Cases

6) Vehicle Load
   - The location of a vehicle in the transverse direction was selected by considering the influence diagrams for three cases.
   - The top of webs was considered as the support in the transverse direction.

Fig 25. The influence diagram of a girder

a) Maximum member forces at the web
b) Maximum member forces at the middle of the slab
c) Maximum member forces at the cantilever

Check Point

Fig 25. The influence diagram of a girder
3. Loads

Overview of Loads

- Load Cases

6) Vehicle Load

Fig 26. Location of Vehicles
3. Loads

Overview of Loads

- Load Cases
  
  6) Vehicle Load

Fig 27. Applied Plane Loads
3. Loads

Overview of Loads

- Load Cases

7) Cable Force
   - Pretension Loads function was used.
   - The data for cable forces was from the global analysis.

Fig 28. Cable Force : Pretension Loads
### Overview of Loads

- **Load Cases**

  7) **Cable Force** *(Another approach)*
  - Cable anchorages in the girder can be considered as **Point Spring** boundaries.

---

**Fig 29. Cable Force: Point Spring (Another approach)**

- Displacement results against the unit load
- Use Point Spring with the stiffness calculated
- Calculate the stiffness of springs = 1 / Displacement

---
4. Boundary Condition

Boundary Conditions

- Boundary Condition

1) At the girder
   - Define Supports function was used.
   - Displacement of all direction was fixed.

Fig 30. Boundary Conditions
Boundary Conditions

2) At the anchorages
   - **Define Supports function** and **Rigid Link function** were used.
   - At the pylon; 6DOF were fixed.
   - At the girder; Nodes for anchorages were linked to nodes in the upper slab of the girder.

![Diagram of boundary conditions](image)

a) Cable Anchorage at the pylon

b) Cable Anchorage at the girder

Fig 31. Boundary Conditions
5. Post Processing

- Overview of Checking results

  • Post processing

    1) Define load combinations

      - **Load Combination function** was used.

![Fig 32. Post Processing](image-url)
5. Post Processing

Overview of Checking results

- Post processing
  2) Extract stress & member forces results.
     - **Results function** were used.
  3) Check lists
     - Deformations, Reactions, Forces & Stresses.

Fig 33. Post Processing
5. Post Processing

Overview of Checking results

- Post processing
  4) Modify the input & plan.
    - Stresses on the upper slab were controlled through prestress loads and tendon profile.
    - Tendon profiles were modified to manage the stress range of the slab.

![Graphs showing stress distribution before and after modification.](attachment:image1.png)

- Initial condition of the tendon profile at the cable anchorage
- Modified condition of the tendon profile at the cable anchorage

**Fig 34. Check & Modification of Tendon Profile**
5. Post Processing

Overview of Checking results

- The conclusion

<table>
<thead>
<tr>
<th>Modified Condition</th>
<th>Normal Section</th>
<th>At The Rib</th>
<th>At The Cable Anchoragae</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1.Rebar size at the inner web.</td>
<td>1.Rebar size at the inner web. 2.Rebar size at the bottom slab.</td>
<td>1.Rebar size at the inner web. 2.The shape of the rib in the inner box.</td>
</tr>
<tr>
<td></td>
<td>2.Rebar size at the bottom slab.</td>
<td>3.Inner ribs were deleted.</td>
<td></td>
</tr>
</tbody>
</table>

Fig 35. Initial & Modified Condition of the girder
6. Epilogue

- **Detail Design Step**

  ![midasFEA/NX](image1)

  ![midasCivil](image2)
Thank you for watching the webinar.

MIDAS Academy: https://www.midasbridge.com/midascivil-seminar

Global Technical Center: http://globalsupport.midasuser.com