Three-Span Continuous Horizontally Curved Composite Steel TUB Girder Bridge
Bridge Information
Material and Section Properties
Wizard
Modelling Tweaks
Live Load and Analysis Control
Results
Load Combinations
Design
Question & Answer
Bridge Information
Material Properties

Structural Steel: Grade 50W (ASTM A709)
Concrete: $f'c = 4.0$ ksi
Slab Reinforcing Steel: Grade 60 with $F_y = 60$ ksi

Loadings on the Bridge

Self Weight of the Bridge
Live Load: Three 12ft Traffic Lanes HL-93
Wearing Surface: 25 psf
Parapets/Barriers: 495 lb/ft
Section 1: For Span 1 and 3

Section 2: For Span 2
2. Material and Section Properties
1. From the Ribbon Menu Go To Properties > Material Properties
2. Click on Add
3. From Type of Design choose Steel
4. Standard: ASTM(S)
5. DB: A709-50W
6. Click Apply
1. Type of Design: **Concrete**
2. Standard: **ASTM(RC)**
3. **DB**: **C4000**
4. Standard: **None**
5. Edit Modulus of Elasticity: 3.8340e+003 kips/in²
6. Name: **4 ksi**
7. Click OK
1. Go To: Section Tab > Add
2. Go To: Composite Tab
3. Section Type: Steel-Tub (Type-1)
4. Slab Information
   Bc: 243 in
   tc: 9.5 in
   Hh: 4 in
5. Girder Information
   Hw: 78, B1: 104, BF1: 16, TF1: 1, BF3: 8
   TW: .5625, B2: 81, BF2: 1, TF2: .625
6. Material > Select Material from DB...
7. Concrete Material:
   DB: ASTM (RC), Name: Grade C4000
8. Steel Material:
   DB: ASTM (S), Name: A709-50W
   Click OK
9. Modify Es/Ec: 7.56
10. Check on Multiple Modulus of Elasticity
11. Es/Ec (Creep): 22.68
12. Click on Change Offset...
13. Offset: Center-Top
14. Click OK
15. Name: Section-1
16. Click Apply
1. Name: Section-2

2. Girder Information
   Hw: 78, B1: 104, Bf1: 18, tf1: 3, Bf3: 9
   tw: .5625, B2: 81, Bf2: 1, tf2: 1.5

3. Click on Stiffener...
4. Name: WT8x25
5. Type: Tee
6. H: 8.13, B: 7.07, tw: 0.38, tf: 0.63
7. Click Add
8. Give NBottom as 1
9. Check on C
10. d(in): 41.5
11. Click OK
12. Click Apply
1. Shift to DB/User
2. Name: Diaphragm
3. Choose Solid Rectangle from Type
4. Shift to User
5. H: 78 in, B: 12 in
6. Offset: Center-Top
7. Click Apply
8. Choose T-Section from Type
9. Shift to DB
10. Choose Code: AISC10 (US)
11. From Sect. Name Choose WT9x48.5
12. Offset: Center-Center
13. Click OK
14. Click Close to exit the Properties Window
3. Wizard
1. From Ribbon Menu Go to Structure > Steel Composite Bridge

This will open the LAYOUT Tab

2. Girder Type: Composite Steel Tub
3. Modeling Type: All Frame
4. Span Information: 1920, 2520, 1920 (in)
5. Deck Width: 486 in
6. Radius: 8400 in, Type: Concave
7. Boundary: Bearing Type
8. Support Type: Supports
9. Fixed Support: Pier 1
10. Direction: Tangential
11. Click Save As...
12. Save the Wizard file as Main Wizard
1. Go to the Sections Tab
2. Deck Thickness : 9.6 in
3. Material:
   Deck: 4 ksi
   Girder: A709-50W
   Bracing: 4 ksi
4. Number of Girders: 2, Apply
5. Girder Offset: For 2 Girders. -135, 135
6. Transverse Deck Element Spacing: Distances: 24 in

**Bracing Information**
7. Click on Bracing Details
8. Name: Diaphragms
9. Type: Single Beam
10. Element Type: Beam
11. Choose Diaphragm
12. Click Add
13. Click Close
14. Spacing: Divisions per Span
15. Choose Diaphragms in Name and Divisions per Span as 10, 14, 10
   Angle Type: Perpendicular

**Girder Information**
16. No. of Divisions: 3, Apply
17. 1. Section-1, End: 1920 in
2. Section-2, End: 4440 in
3. Section-1, End: 6360 in
18. Click on Copy Current Girder Data to Other Girders
Select Girder 2 and Click OK
19. Check On Generate 10th points elements
1. Go to the **Loads** Tab
2. Pavement and Barrier:
   - b1: 18 in, b2: 225 in, b3: 0 in
   - b4: 225 in, b5: 18 in
3. Dead Loads:
   - Self Weight: **Check ON**
   - Wet Concrete: **Check ON**
     - Weight Density: 8.68
     - Thickness: 9.6
   - Barrier: **0.04125**
   - Wearing Surface: **Check ON**
     - Weight Density: 0.003472
     - Thickness: 0.05 in
4. Live Loads: **Check ON**
5. Click on **Define Moving Load Case...**
   - Choose: AASHTO LRFD
6. Click **Define Traffic Lanes...**
   - Number of Lanes: 3
     - D1: 99
     - D2: 243
     - D3: 387
   - Click **OK**
7. Click on **Define Vehicles...**
8. Click on **Add Standard**
9. Vehicle Load Type: HL-93TRK
10. Dynamic Load Allowance: 33%
11. Click **Apply**
12. Vehicle Load Type: HL-93TDM
13. Click **OK**
14. Click **Close**
1. Go to **Construction Stage** Tab
2. Construction Stage : Check ON
3. Deck Pouring Sequence : Check ON
4. Click on **Deck Split Construction**
5. Deck Stage : 1
6. Duration : 10
7. Select D1, D3 & D5 to Selected Deck List
8. Click **Add**
9. Similarly Deck Stage: 2
10. Select D2 & D4 and Click **Add**
11. Click **OK**
12. Click on Advanced
13. For **Support1**:
   - Negative-moment zone length:
     - S1: 420 in
     - S2: 540 in
14. Click **Modify**
15. Click **Close**
16. Check on **Long Term Boundary Group...**
17. Check on **Creep 1** and **Creep 2**
18. Click **Save As...** save the wizard file again.
19. Click **OK**
4. Modeling Tweaks
Modeling
Different View Shortcuts

Wireframe View: Ctrl+H
Top View: Ctrl+Shift+T
Front View: Ctrl+Shift+F
Isometric View: Ctrl + Shift + I
Right View: Ctrl + Shift + R
Left View: Ctrl + Shift + L

To Display Boundaries / Other Entities
1. Switch to Works Tree
2. Maximize Boundaries
3. Maximize Supports
4. Right Click on Type 1 > Display
1. In the Works Tree, under Sections, **Double Click on Diaphragms**, to select all the Diaphragms.
2. Click on **Activate** to activate all the Diaphragms
3. Click on **Select Single** and Select the **Interior Diaphragms**
4. Clear all the nodes in the **Node Selection Bar**, to unselect the nodes
5. **Click Delete from Keyboard.**
1. Go to File > Merge Data File
2. Click on Browse...
3. Look for file Only Bracings.mcb and Open it.
4. Uncheck Material, Section, Thickness
5. Check on Make Group and Group Name Only Bracings
6. Check ON Intersect Frame Elements.
7. Click on OK
1. Go to Groups > Double Click on Only Bracings
2. Click Activate to Activate the Bracings
3. Go to Boundary > Rigid Link
4. Scroll your Middle Mouse Button, to zoom into first Bracing, Click on Master Node Box and click on the topmost node.
5. Select Single and select all the other nodes (Slave Nodes)
6. Click on Rigid Body
7. Click on Apply
8. Repeat the same for all the bracings.
1. Press Ctrl+A to Activate All
2. Go to Group and Double Click on Only Braces
3. From Boundary Groups Drag and Drop > Substructure
4. Click OK
5. Press Ctrl+Q (Select Previous) From Works Tree, Drag and Drop Material > A709-50W
6. Press Ctrl+Q and Drag and Drop Section > WT9x48.5
1. Go to Loads > Construction Stage Loads
2. Go to Define Construction Stages
3. Double Click on CS1
4. From Element Group List Select Only Bracings
5. Click on Add under Activation
6. Click OK
7. Click Close to Close the Construction Stage Definition
1. Go To Works Tree
2. Drag Down to Moving Load Analysis
3. Maximize Traffic Line Lanes
4. Right Click on Traffic Line Lane 1 and Click Properties
5. Check on Traffic Lane Optimization.
6. Click on OK
7. Similarly do the same for all other Lanes.
8. Click Close on Traffic Line Lanes
9. Right Click on Traffic Line Lanes and click Display
10. Right Click and Undisplay to Undisplay.
1. Go To Load > Moving Load > Moving Load Cases
2. Click on Add.
3. Load Case Name: MVL
4. Loading Effect: Independent
5. Click Add
6. Vehicle Class : VL: HL-93TRK
7. Scale Factor: 1
8. Min. Number of Loaded Lanes : 1
9. Max. Number of Loaded Lanes : 3
10. Select All the lanes from the List of Lanes to Selected Lanes.
11. Click OK.
12. Similarly, repeat the same for VL-HL93 TDM
13. Click OK.
14. Click Close.
1. Go To Load > Moving Load > Lane Support Negative Moment
2. Inside Girder Group choose Girder
3. Click Add
4. In the Works Tree, Press Ctrl and Double Click on the first Support and Press Shift and Double Click on the last. (It will select all the Boundary Nodes)
5. Go To Load > Moving Load > Lane Support Reaction
6. Click on Apply
5. Analysis Control
1. Go To Analysis > Moving Load
2. Analysis Method: Exact
3. Load Point Selection: Influence Line Dependent Point
4. Number/Line Element: 3
5. Analysis Results:
   - Frame: Normal + Concurrent Force/Stress
   - Check On Combined Stress Calculation
6. Click OK
7. Click Perform Analysis
Please join for the upcoming webinars

For further queries and doubts write a mail to: techsupport@midasit.com