

Three- Span Continuous Horizontally Curved Composite Steel TUB Girder Bridge

WIZARD, ANALYSIS AND DESIGN



Bridge Information

Material and Section Properties

Wizard

Modelling Tweaks

Live Load and Analysis Control

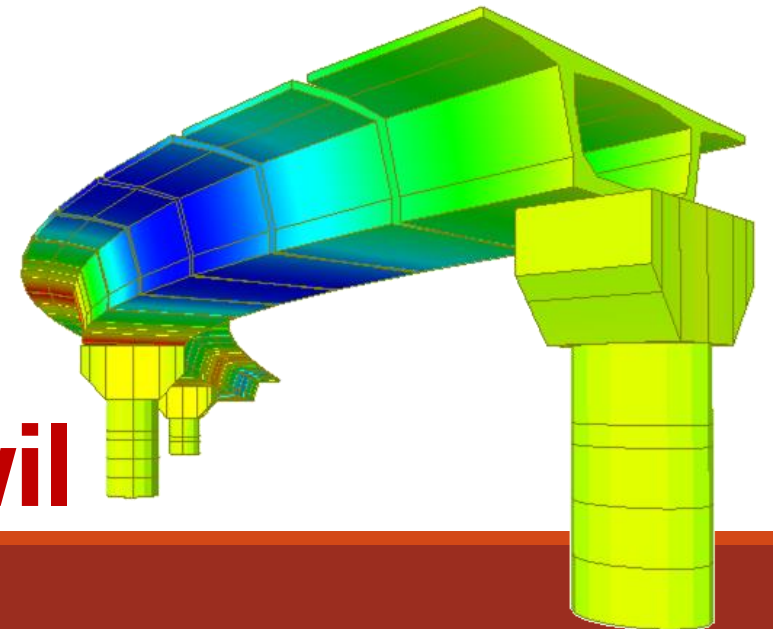
Results

Load Combinations

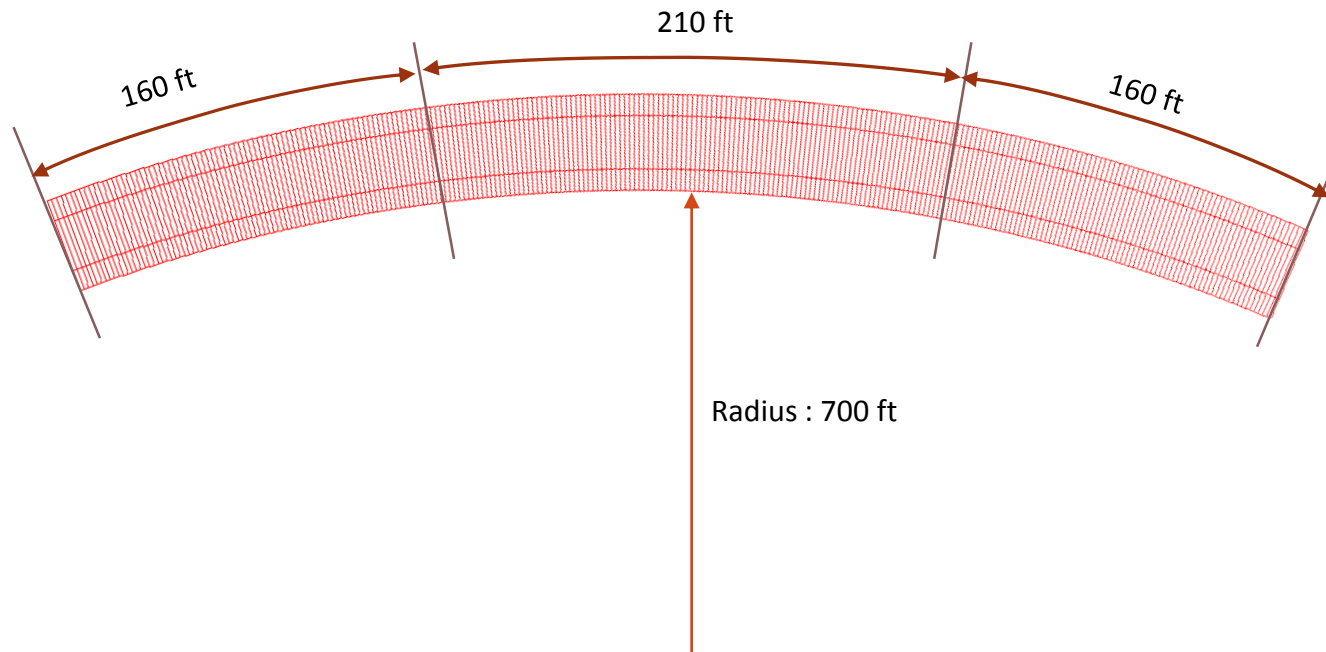
Design

Question & Answer

midas **Civil**



Bridge Information

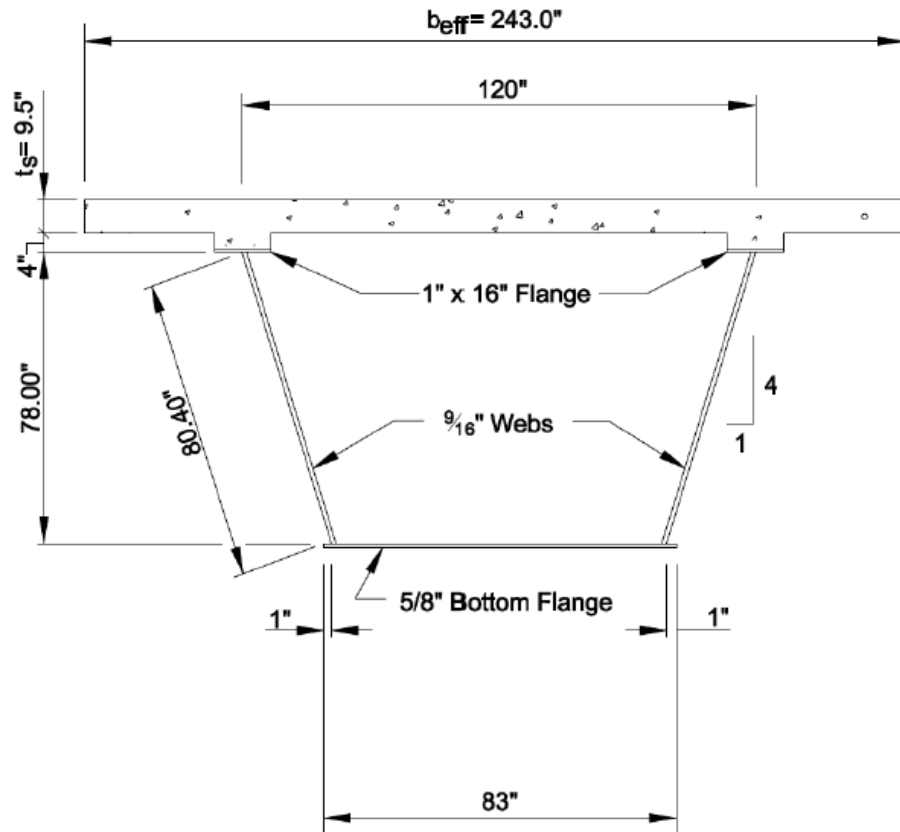


Material Properties

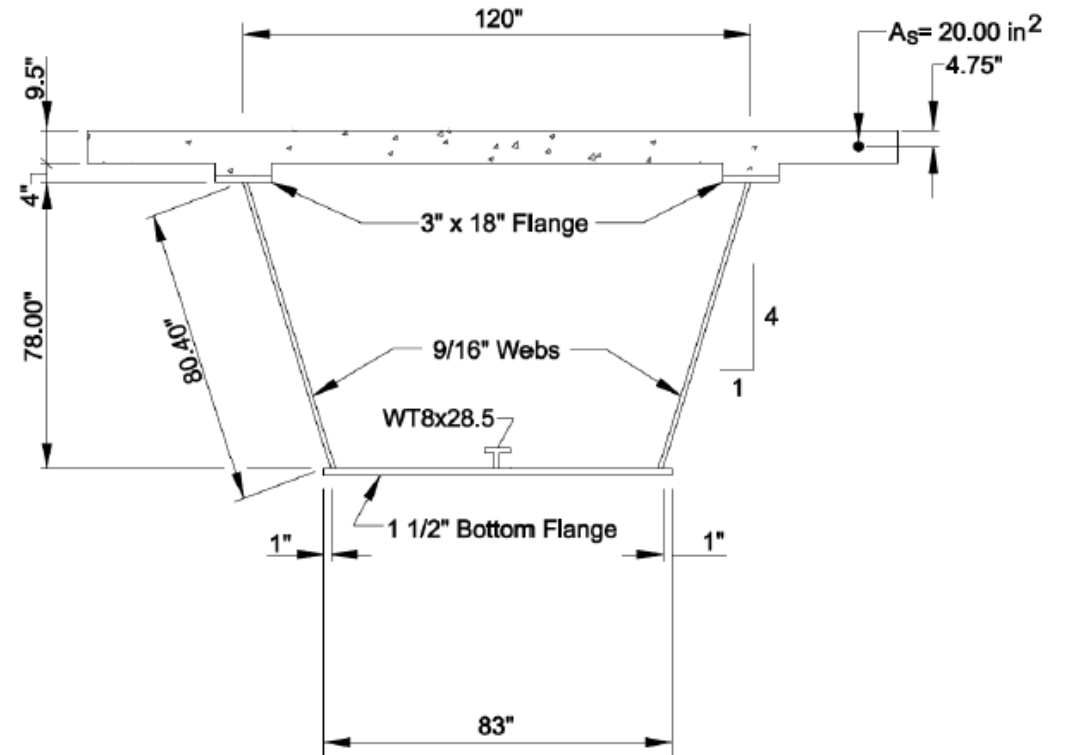
Structural Steel :	Grade 50W (ASTM A709)
Concrete :	$f'_c = 4.0$ ksi
Slab Reinforcing Steel :	Grade 60 with $F_y = 60$ ksi

Loadings on the Bridge

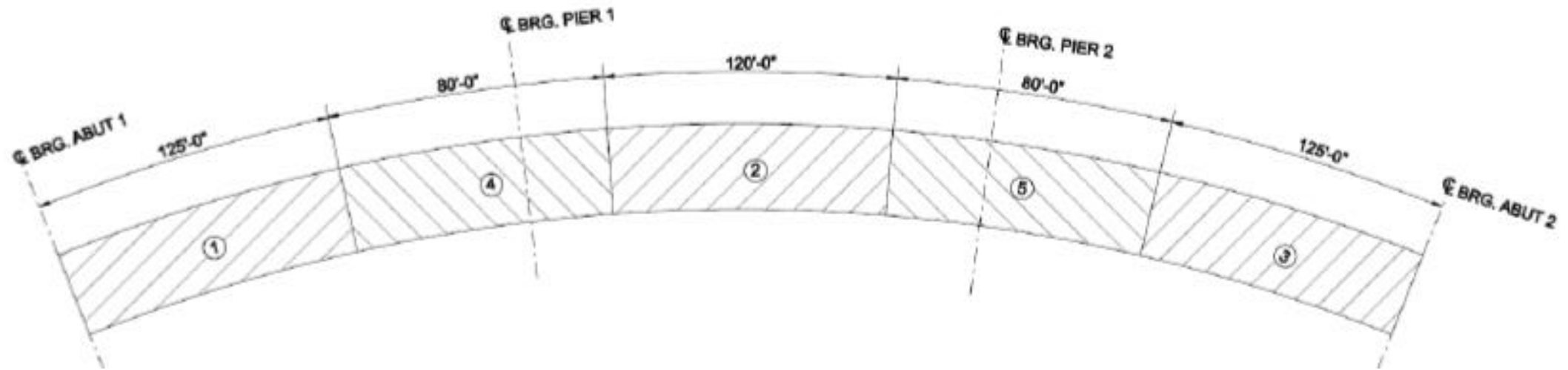
Self Weight of the Bridge
Live Load : Three 12ft Traffic Lanes HL-93
Wearing Surface : 25 psf
Parapets/Barriers : 495 lb/ft



Section 1 : For Span 1 and 3

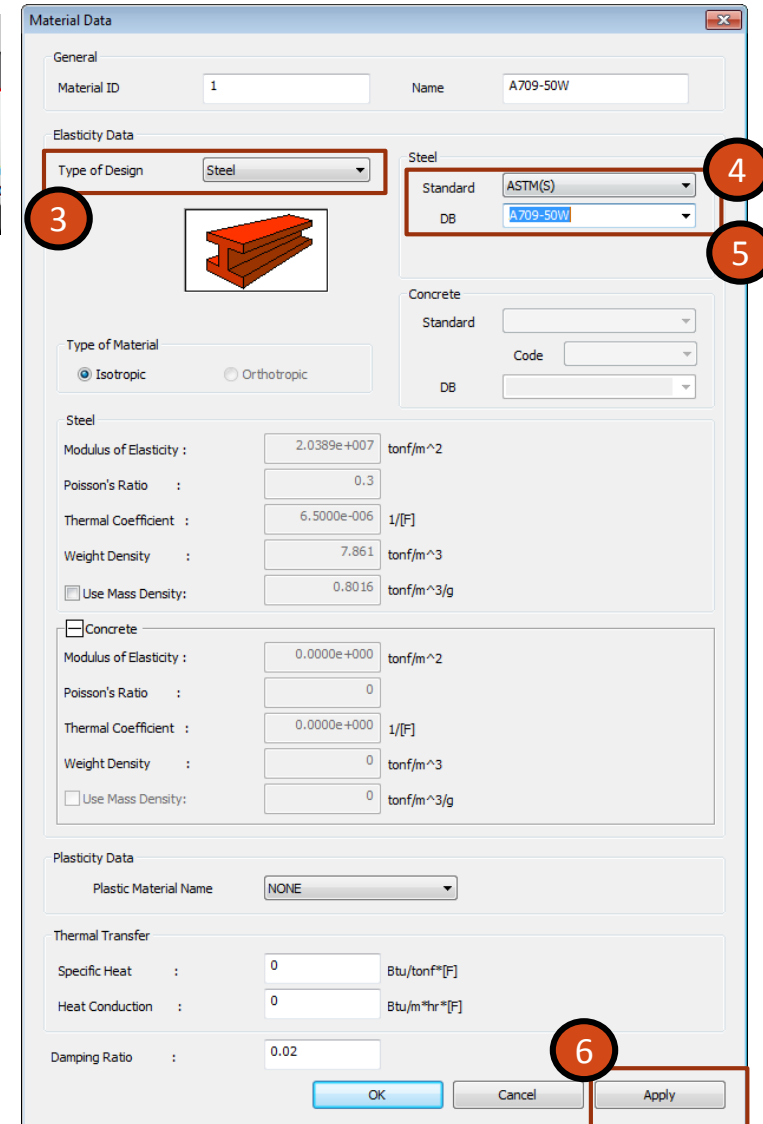
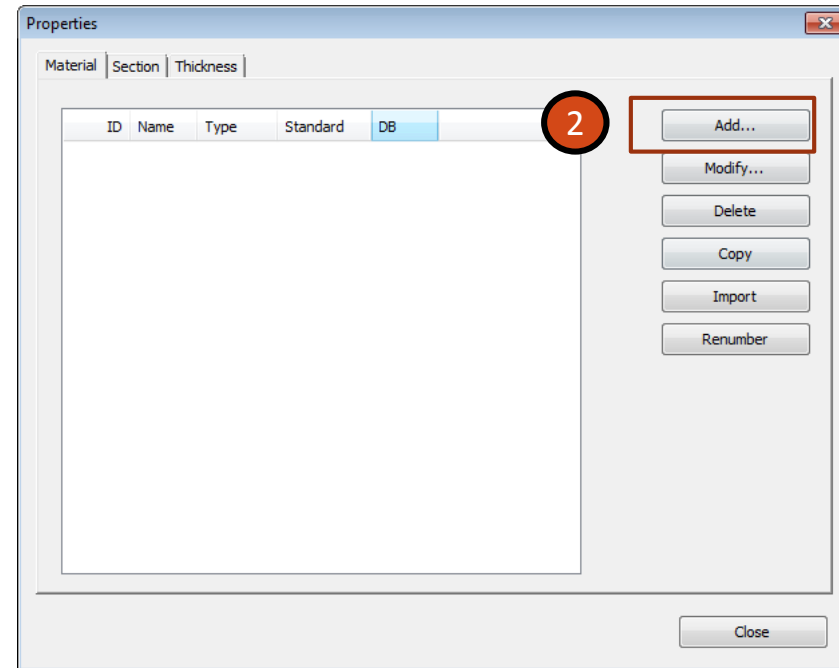
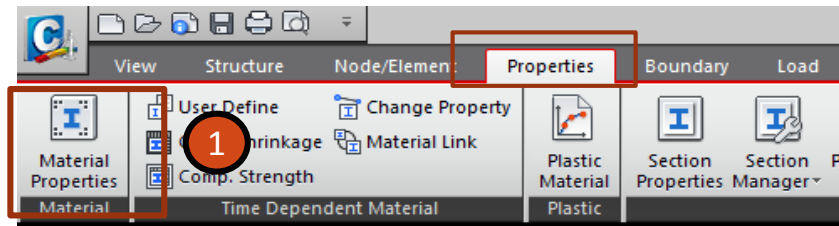


Section 2 : For Span 2



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**

Material Data

General
Material ID: 2 Name: 4 ksi

Elasticity Data
Type of Design: Concrete (1)
Standard: ASTM(RC) (2)
DB: Grade C4000 (3)

Type of Material
 Isotropic Orthotropic

Steel
Modulus of Elasticity: 0.0000e+000 kips/in²
Poisson's Ratio: 0
Thermal Coefficient: 0.0000e+000 1/[F]
Weight Density: 0 kips/in³
 Use Mass Density: 0 kips/in³/g

Concrete
Modulus of Elasticity: 3.6441e+003 kips/in²
Poisson's Ratio: 0.2
Thermal Coefficient: 5.0000e-006 1/[F]
Weight Density: 8.681e-005 kips/in³
 Use Mass Density: 2.248e-007 kips/in³/g

Plasticity Data
Plastic Material Name: NONE

Thermal Transfer
Specific Heat: 0 Btu/kips*[F]
Heat Conduction: 0 Btu/in*hr*[F]
Damping Ratio: 0.05

Buttons: OK, Cancel, Apply

Material Data

General
Material ID: 2 Name: 4 ksi (6)

Elasticity Data
Type of Design: Concrete
Standard: None (4)
DB:

Type of Material
 Isotropic Orthotropic

Steel
Modulus of Elasticity: 0.0000e+000 kips/in²
Poisson's Ratio: 0
Thermal Coefficient: 0.0000e+000 1/[F]
Weight Density: 0 kips/in³
 Use Mass Density: 0 kips/in³/g

Concrete
Modulus of Elasticity: 3.8340e+003 kips/in² (5)
Poisson's Ratio: 0.2
Thermal Coefficient: 5.0000e-006 1/[F]
Weight Density: 8.681e-005 kips/in³
 Use Mass Density: 2.248e-007 kips/in³/g

Plasticity Data
Plastic Material Name: NONE

Thermal Transfer
Specific Heat: 0 Btu/kips*[F]
Heat Conduction: 0 Btu/in*hr*[F]
Damping Ratio: 0.05

Buttons: OK (7), Cancel, Apply

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**

The image shows the 'Section Data' dialog box in Midas Civil, with the 'Composite' tab selected. The 'Section ID' is 1 and the 'Name' is 'Section-1'. The 'Section Type' is 'Steel-Tub (Type1)'. The 'Slab' section has Bc = 243 in, tc = 9.5 in, and Hh = 4 in. The 'Girder' section has Hw = 78 in, tw = 0.5625 in, B1 = 104 in, B2 = 81 in, Bf1 = 16 in, Bf2 = 1 in, tf1 = 1 in, and tf2 = 0.625 in. The 'Material' section shows 'Es / Ec' set to 7.56, 'Ds / Dc' to 3.27168, 'Ps' to 0.3, 'Pc' to 0.2, 'Ts / Tc' to 1.3, and 'Multiple Modulus of Elasticity' checked. The 'Steel Material' is set to 'ASTM (S)' with name 'A709-50W'. The 'Change Offset' dialog is open with 'Offset' set to 'Center-Top' and 'Center Loc.' set to 'Centroid'. The 'Select Material of Concrete and Steel' dialog is also open with 'Concrete Material' set to 'ASTM (RC)' and 'Grade C4000', and 'Steel Material' set to 'ASTM (S)' and 'A709-50W'. The 'Apply' button is highlighted in the bottom right corner.

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**

Section Data

DB/User | Value | SRC | Combined | PSC | Tapered | Composite | Steel Girder

Section ID: 2 Name: Section-2

Section Type: Steel-Tub (Type1)

Slab Width: 0 in

Girder : Num: 0 CTC: 0 in

Slab

Bc	243	in
tc	9.5	in
Hh	4	in

Girder

Hw	78	in	tw	0.5625	in
B1	104	in	B2	81	in
Bf1	18	in	Bf2	1	in
tf1	3	in	tf2	1.5	in
Bf3	9	in			

Stiffener...

Material

Select Material from DB ...

Es / Ec	7.56	Ds / Dc	3.27168
Ps	0.3	Pc	0.2
Ts / Tc	1.3		

Multiple Modulus of Elasticity

Es/Ec (Creep)	22.68
Es/Ec (Shrinkage)	0

Consider Shear Deformation.
 Consider Warping Effect(7th DOF)

Offset : Center-Top

Change Offset ...

Show Calculation Results... OK Cancel Apply

Section Stiffener

Stiffener Properties

Name: WT8x25

Type: Tee

H: 8.13 in

B: 7.07 in

tw: 0.38 in

tf: 0.63 in

Add Modify Delete

Name: WT8x25 Type: Tee

Position: Both Left Right

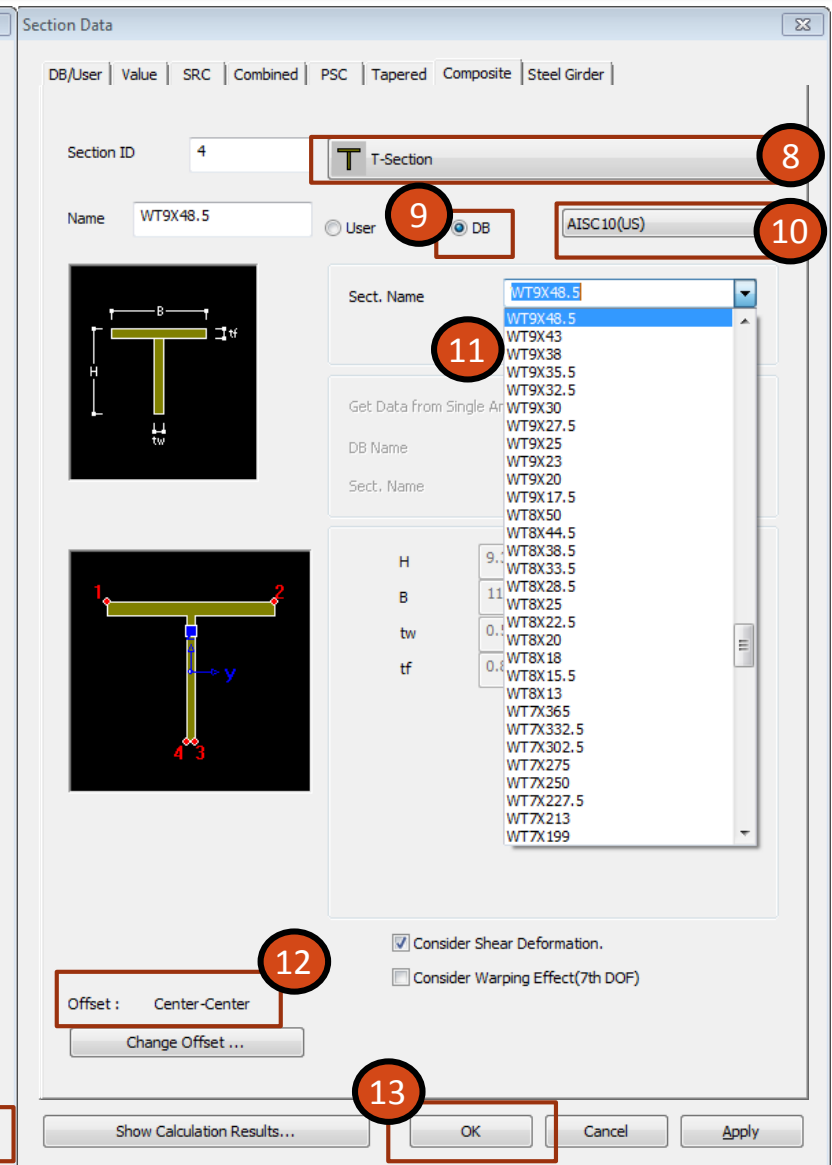
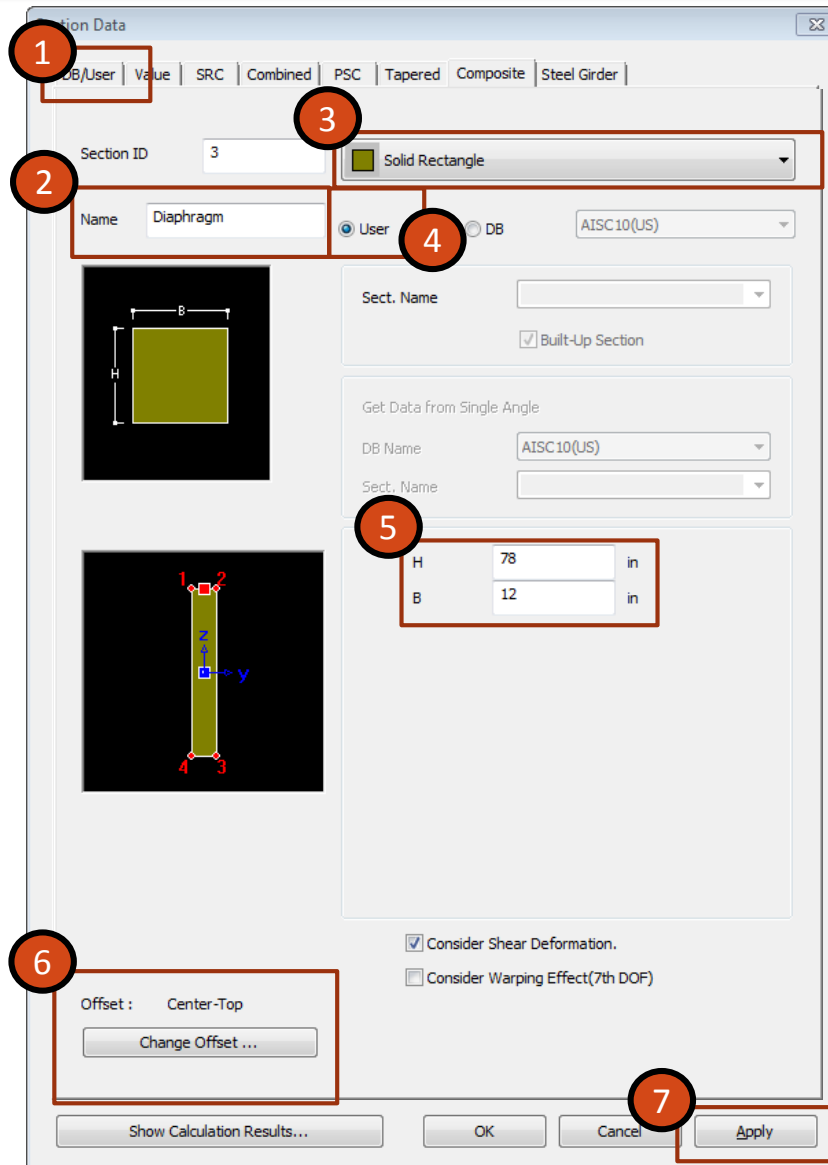
N Left: 0 N Right: 0 N Bottom: 1 N Top: 0

C	d (in)	Stiffener
<input checked="" type="checkbox"/>	41.5	WT8x2

OK Cancel

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

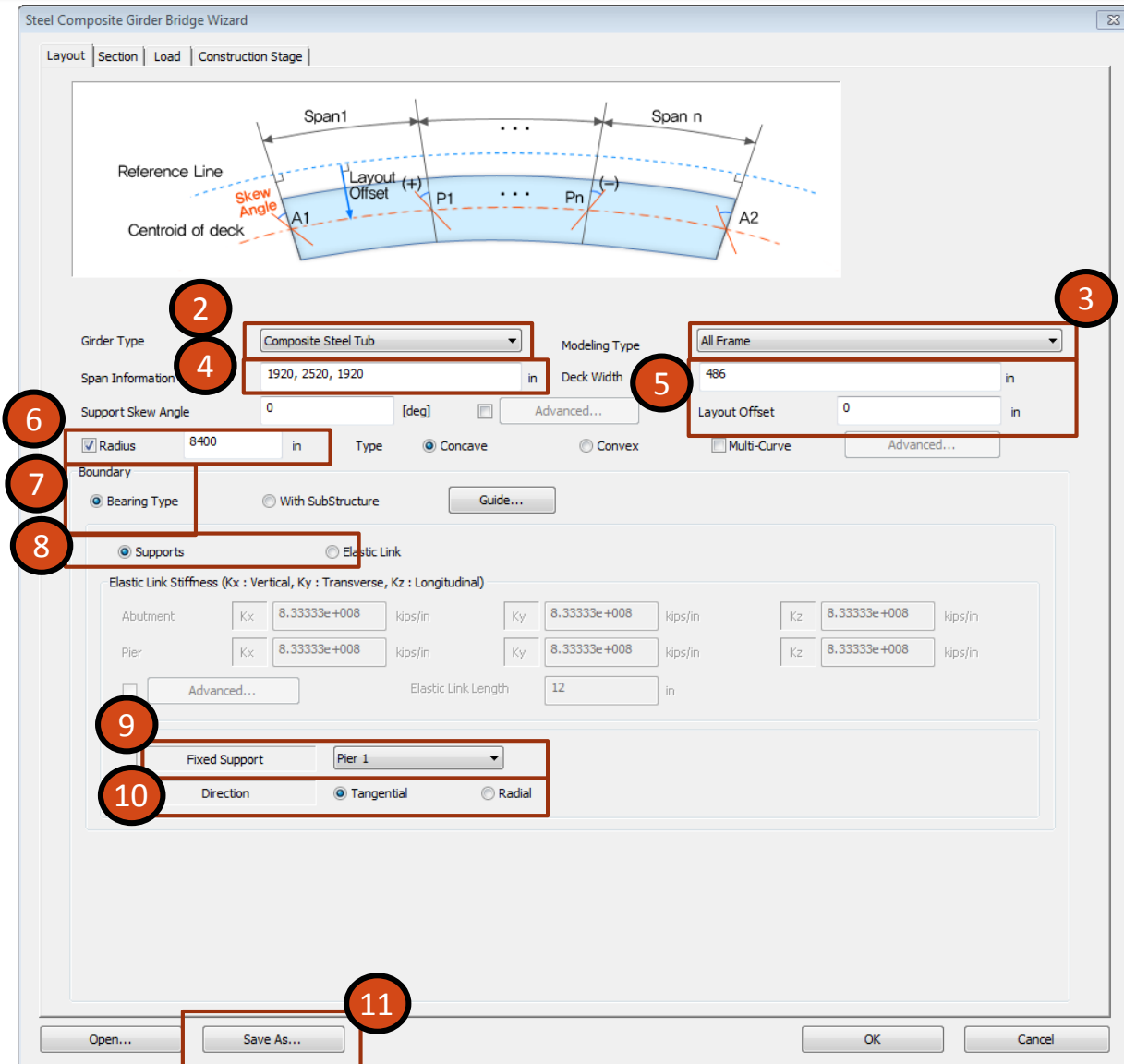
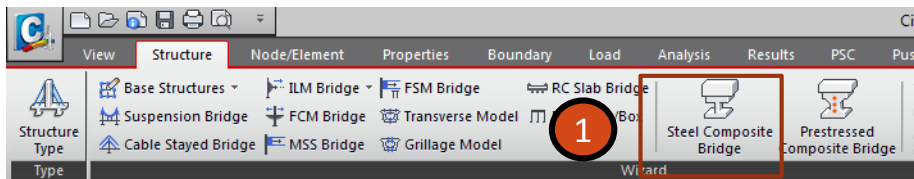


3. Wizard

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**

The screenshot displays the 'Composite Girder Bridge Wizard' software interface. It is divided into several sections:

- Section Tab:** Contains input fields for Deck Thickness (9.6 in), Launch Height (3 in), Number of Girders (2), and Material (Deck: 4 ksi, Girder: A709-50W, Bracing: 4 ksi).
- Transverse Deck Element Spacing:** Set to Distances with a value of 24 in.
- Bracing Information:** Includes a 'Bracing Details...' button, Spacing (Divisions per Span), and a table for Girder Name, Divisions per Span, Angle Type, and Angle.
- Girder Information:** Shows a table with columns for No., Name, Start (in), and End (in) for three sections.
- Copy Girder Data to Other Girders:** A dialog box with a list of girders and a 'Selected List' containing 'Girder 2'.
- Define Bracing Details:** A separate dialog box with fields for ID, Name (Diaphragms), Type (Single Beam), Element Type (Beam), and Single Beam (3: Diaphragm).

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**

The screenshot shows the 'Composite Girder Bridge Wizard' software interface. The 'Loads' tab is active. The 'Pavement and Barrier' section shows a deck width diagram with dimensions b1 through b5. The 'Dead Loads' section has checkboxes for 'Self Weight' and 'Wet Con'c' checked, with input fields for weight density, thickness, and barrier. The 'Live Loads' section has 'Live Loads' checked. The 'Define Moving Load Case...' dialog is open, showing 'AASHTO LRFD' selected. The 'Define Traffic Lanes...' dialog is open, showing 3 lanes and distance values D1=99, D2=243, D3=387. The 'Define Standard Vehicular Load' dialog is open, showing 'HL-93TRK' selected and a dynamic load allowance of 33%. The 'Vehicles' dialog is open, showing 'Add Standard' selected. The 'Select Moving Load Code' dialog is open, showing 'AASHTO LRFD' selected.

1 Composite Girder Bridge Wizard

2 Deck width diagram with dimensions b1, b2, b3, b4, b5 and positions P1, P2.

3 Dead Loads section with checkboxes for Self Weight and Wet Con'c, and input fields for Weight Density, Thickness, Barrier, Median Strip, Additional Load, and Utilities.

4 Live Loads section with the Live Loads checkbox checked.

5 Define Moving Load Case... dialog with AASHTO LRFD selected.

6 Define Traffic Lanes... dialog with 3 lanes and distance values D1=99, D2=243, D3=387.

7 Define Vehicles... dialog with Add Standard selected.

8 Add Standard button in the Define Vehicles... dialog.

9 Define Standard Vehicular Load dialog with HL-93TRK selected.

10 Dynamic Load Allowance: 33% in the Define Standard Vehicular Load dialog.

11 Apply button in the Define Standard Vehicular Load dialog.

12 Define Standard Vehicular Load dialog with HL-93TDM selected.

13 OK button in the Define Standard Vehicular Load dialog.

14 Close button in the Define Vehicles... dialog.

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**

The screenshot displays the 'Steel Composite Girder Wizard' interface. The 'Construction Stage' tab is active. The main wizard window shows a table of construction stages and a 'Deck Split Construction' button. A 'Modify Deck Split Construction Stage' dialog is open, showing a sequence of deck stages (D1, D2, D3, D4, Dn-1, Dn) and a 'Selected Deck List' containing D1, D3, and D5. A 'Negative-moment zone length' dialog is also open, showing a diagram of a pier with supports S1 and S2, and a table of support lengths. A 'Long Term Boundary Group' dialog is open, showing a list of boundary groups with 'Creep 1' and 'Creep 2' checked. The interface includes various buttons like 'Advanced...', 'Add', 'Modify', 'Delete', 'OK', 'Cancel', 'Save As...', and 'Close'.

Stage	Stage Description	Load Condition	Duration (Day)
1	Girder(Part1), Bracing is Activated	Form Work Load Activated	10
2-1	Deck(D1, D3, D5) is Activated as a load	Wet concrete Load(D1, D3, D5) is Activated	10
2-2	Deck(D1, D3, D5) is in composite stage	Wet concrete Load(D1, D3, D5) is Deactivated	0
2-3	Deck(D2, D4) is Activated as a load	Wet concrete Load(D2, D4) is Activated	10
2-4	Deck(D2, D4) is in composite stage	Wet concrete Load(D2, D4) is Deactivated	0
3	After-Composite Load is Activated	DC2, DW Load Activated	10
4	Long Term Effect is considered	-	10000

Support	S1	S2
Support1	420	540
Support2	540	420

Boundary Group List
<input checked="" type="checkbox"/> Creep 1
<input checked="" type="checkbox"/> Creep 2

Deck Stage	Duration	Decks
1	10	D1, D3, D5
2	10	D2, D4

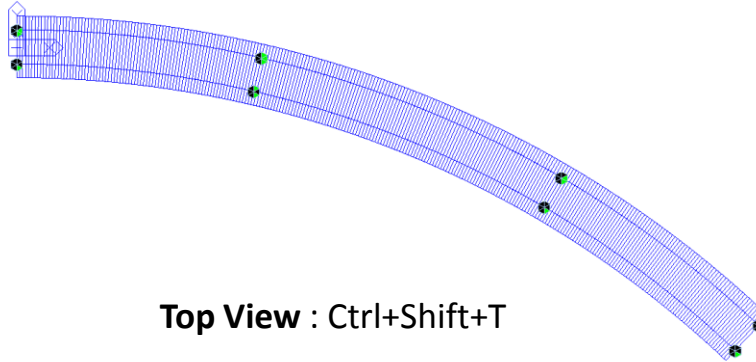
Base



4. Modeling Tweaks



Wireframe View : Ctrl+H



Top View : Ctrl+Shift+T

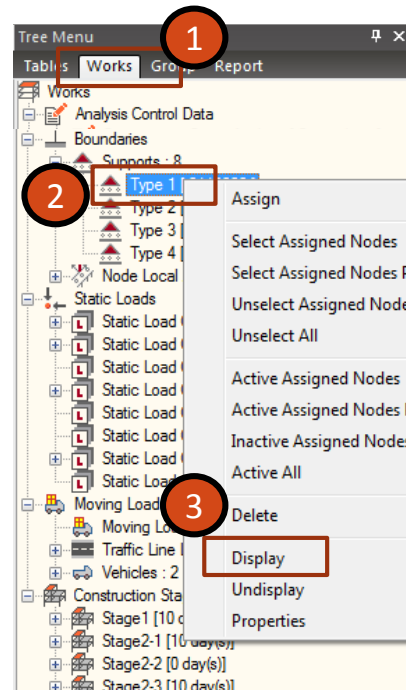


Front View : Ctrl+Shift+F

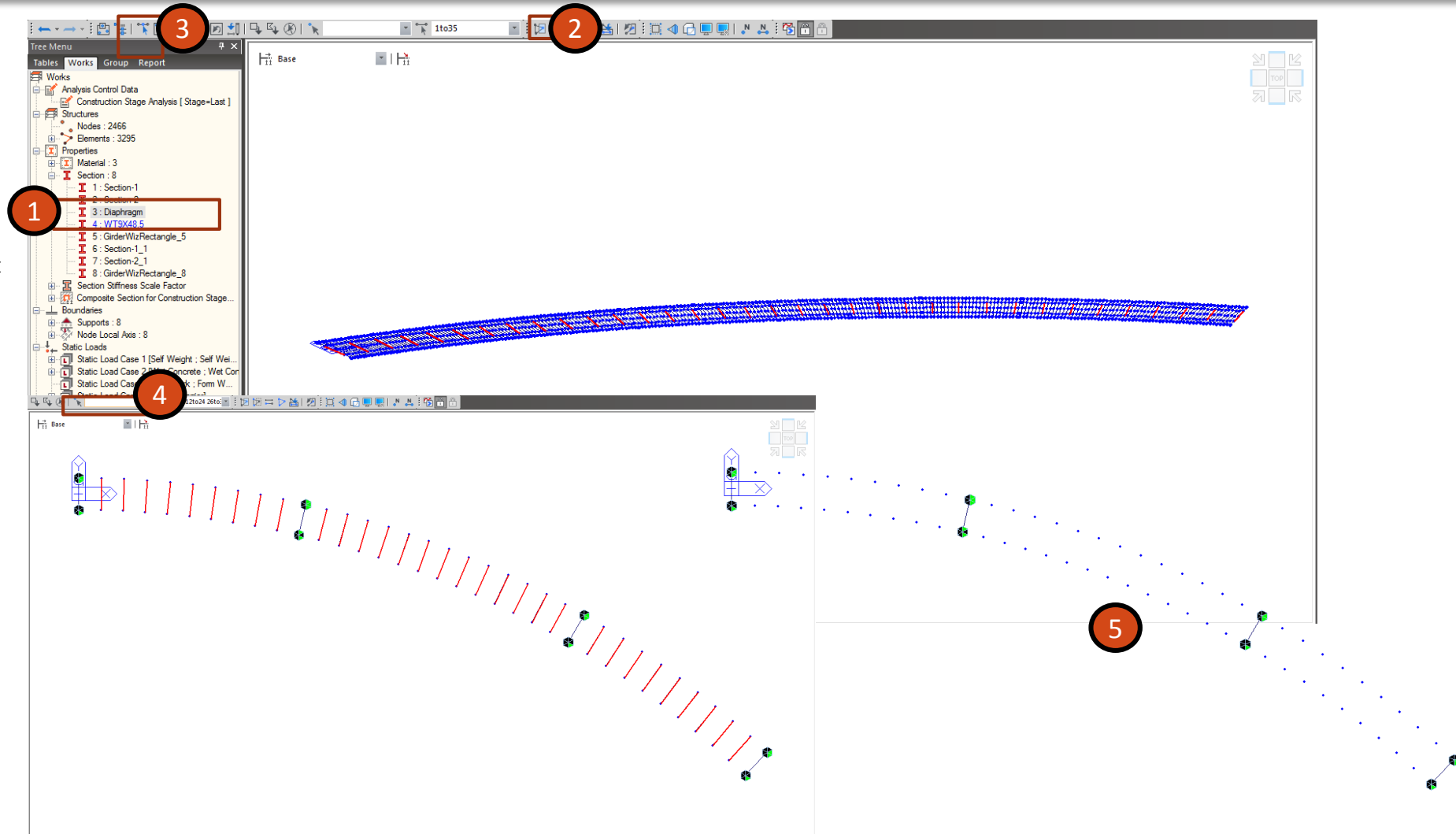
Isometric View : Ctrl + Shift + I
 Right View : Ctrl + Shift + R
 Left View : Ctrl + Shift + L

To Display Boundaries / Other Entities

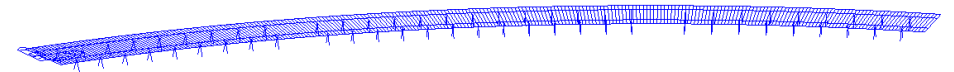
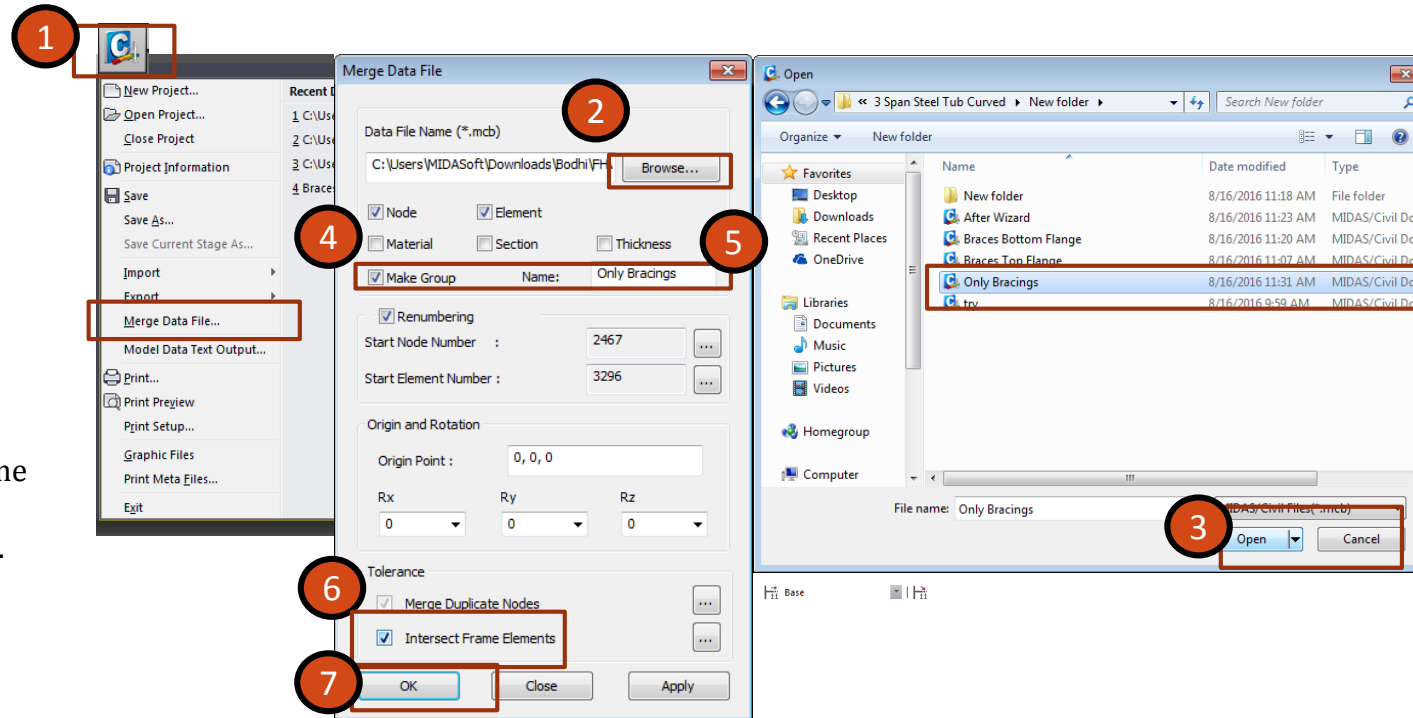
1. Switch to Works Tree
2. Maximize Boundaries
3. Maximize Supports
4. Right Click on Type 1 > Display



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.

The image is a composite of four screenshots from the Midas Civil software interface, illustrating the steps to link cross beams to girders:

- Top Left:** A screenshot of the 'Tree Menu' showing the 'Group' tab. The 'Only Bracings' group is selected and highlighted with a red box and a circled '1'.
- Top Right:** A screenshot of the 3D model showing a long bridge structure with blue nodes and elements. A red box and a circled '2' highlight the 'Activate' button in the top toolbar.
- Bottom Left:** A screenshot of the 'Boundary' tab in the properties dialog. The 'Rigid Link' option is selected and highlighted with a red box and a circled '3'.
- Bottom Center:** A screenshot of the 'Rigid Link' dialog box. The 'Master Node' field is set to '84' and highlighted with a red box and a circled '4'. The 'Rigid Body' option is selected under 'All Types' and highlighted with a red box and a circled '6'. The 'Apply' button is highlighted with a red box and a circled '7'.
- Bottom Right:** A zoomed-in screenshot of the bridge model showing a single bracing element. The topmost node is highlighted with a red box and a circled '4', and the other nodes connected to it are highlighted with red boxes and circled '5's.

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)
6. From Works Tree, Drag and Drop Material > A709-50W
7. Press Ctrl+Q and Drag and Drop Section > WT9x48.5

The image displays two screenshots of the Midas Civil software interface, illustrating the steps for assigning material and section properties to a bridge model.

Top Screenshot: Shows the 'Select Boundary Type' dialog box. The 'SubstructureSupport' option is selected in the 'Works' tree. The 'OK' button is highlighted with a red circle and the number 4. A red arrow points from the 'SubstructureSupport' option in the 'Works' tree to the 'OK' button. The text 'Drag and Drop' is written next to the 'OK' button.

Bottom Screenshot: Shows the 'Works' tree with the following properties assigned:

- Material: 1: A709-50W
- Section: 4: WT9X48.5

Red arrows and red circles with numbers 5 and 6 indicate the selection of these properties from the 'Works' tree.

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**

Construction Stage

Name	Duration	Date	Step	Result
Stage1	10	10	0	Stage
Stage2-1	10	20	0	Stage
Stage2-2	0	20	0	Stage
Stage2-3	10	30	0	Stage
Stage2-4	0	30	0	Stage
Stage3	10	40	0	Stage
Stage4	10000	10040	0	Stage

Compose Construction Stage

Stage: Stage1
 Name: Stage1
 Duration: 10 day(s)

Additional Steps
 Day: 0
 (Example: 1, 3, 7, 14)

Auto Generation
 Step Number: 0
 Generate Steps

Element | Boundary | Load

Group List
 Dummy Beam
 Dummy Beam2
 Dummy Beam-D1
 Dummy Beam-D2
 10th Point Girder-14
 10th Point Girder-14
 10th Point Girder-24
 10th Point Girder-24

Activation
 Age: 0 day(s)
 Group List
 Substructure 0
 Girder 0
 Bracing 0
Only Bracings 0

Deactivation
 Element Force
 Redistribution: 100 %
 Group List
 Name Redist.

Buttons: Add, Modify, Delete, OK, Cancel, Apply

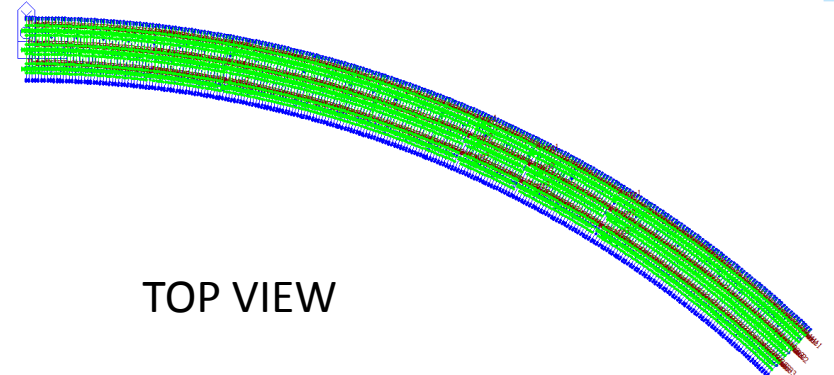
1. Go To **Works Tree**
2. Drag Down to **Moving Load Analysis**
3. Maximize **Traffic Line Lanes**
4. Right Click on **Traffic Line Lane 1** and Click **Properties**
5. Check on **Traffic Lane Optimization**.
6. Click on **OK**
7. Similarly do the same for all other Lanes.
8. Click **Close** on Traffic Line Lanes
9. Right Click on **Traffic Line Lanes** and click **Display**
10. Right Click and **Undisplay** to Undisplay.

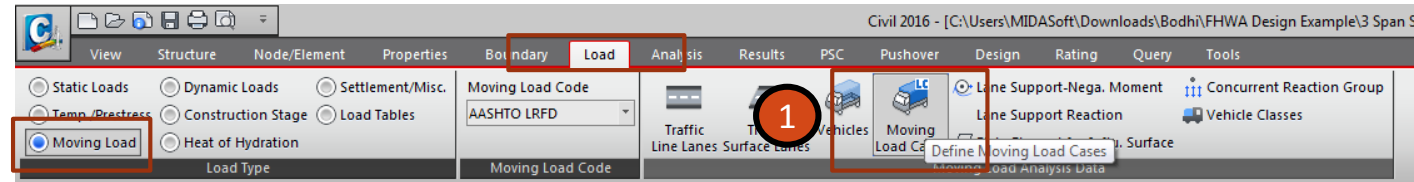
The screenshot shows the 'Traffic Line Lanes' dialog box with the following settings:

- Lane Name: Lane 1
- Lane Width: 120 in
- Eccentricity: 0 in
- Wheel Spacing: 72 in
- Traffic Lane Optimization
- Vehicular Load Distribution: 2 Points
- Operations: Add, Insert, Delete

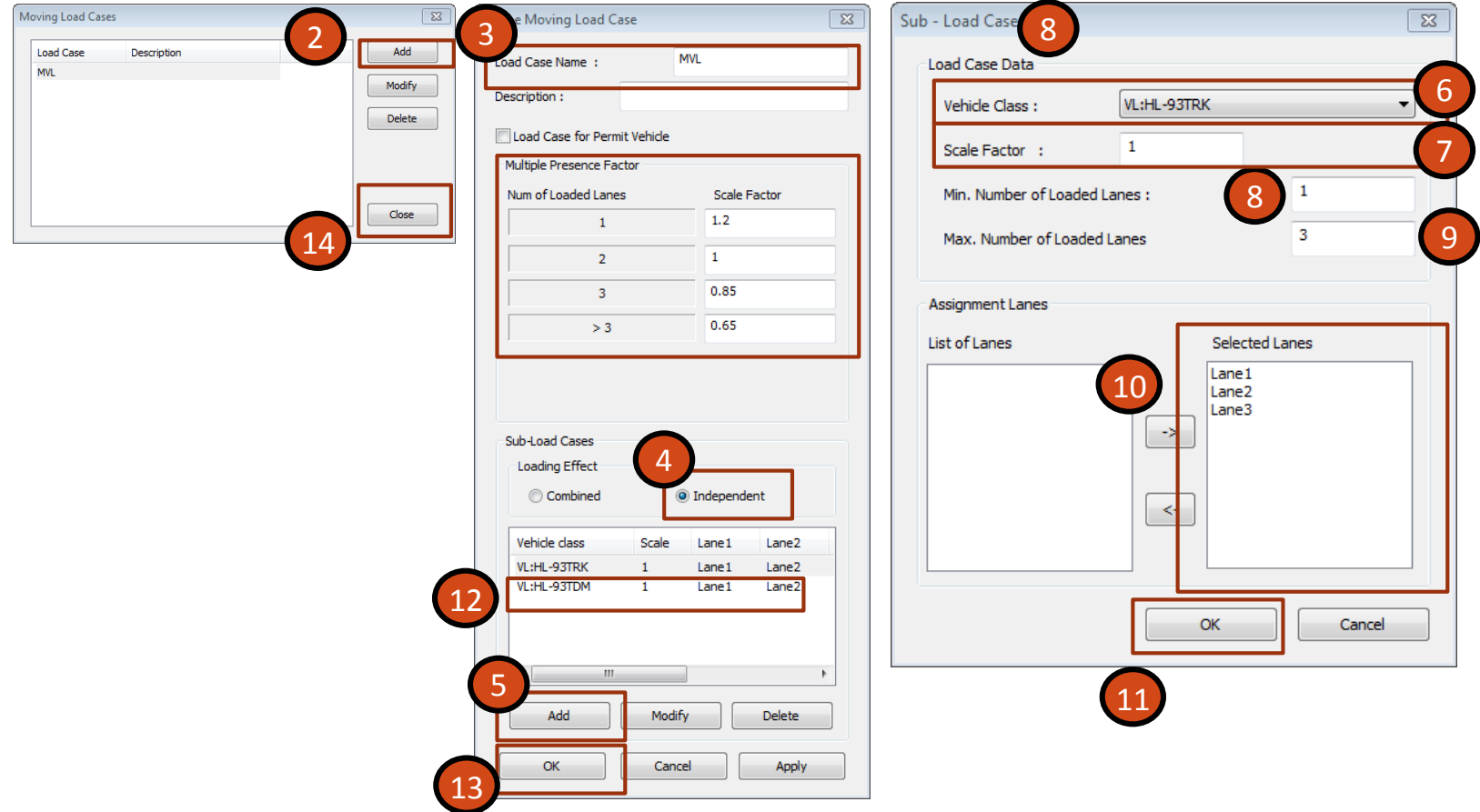
The 'Works Tree' on the left shows the project hierarchy with 'Traffic Line Lanes' selected under 'Moving Load Analysis'. The 'Traffic Line Lanes' list in the dialog shows:

No	Elem	Eccen. (in)	Span Start
1	5	99	<input type="checkbox"/>
2	6	99	<input type="checkbox"/>
	7	99	<input type="checkbox"/>
	8	99	<input type="checkbox"/>





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**.



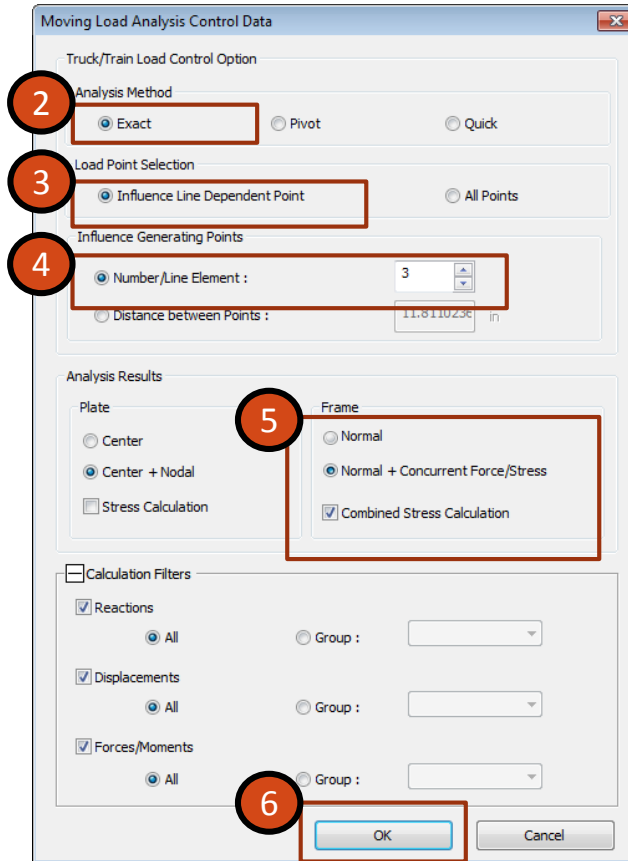
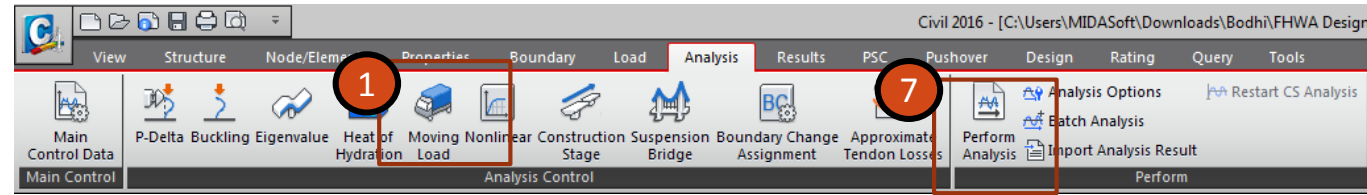
1. Go To **Load > Moving Load > Lane Support Negative Moment**
2. Inside **Girder Group** choose **Girder**
3. Click **Add**
4. In the Works Tree, Press Ctrl and Double Click on the first Support and Press Shift and Double Click on the last. (It will select all the Boundary Nodes)
5. Go To **Load > Moving Load > Lane Support Reaction**
6. Click on **Apply**

The screenshot shows the Midas Civil software interface with the following elements:

- Menu Bar:** View, Structure, Node/Element, Properties, Boundary, Load, Analysis, Results, PSC, Design, Rating, Query, Tools.
- Toolbar:** Static Loads, Dynamic Loads, Settlement/Misc., Temp./Prestress, Construction Stage, Load Tables, Heat of Hydration, Moving Load, Lane Support-Nega. Moment, Lane Support Reaction, Concurrent Reaction Group.
- Works Tree:** Boundaries, Supports: 8 (Type 1 [0110000], Type 2 [0010000], Type 3 [1110000], Type 4 [1010000]), Rigid Link: 62.
- Dialog Boxes:**
 - Lane Supports(Negative Moments):** Shows 'Girder' selected in the 'Girder Group' dropdown and the 'Add' button highlighted.
 - Lane Supports(Reactions at Interic):** Shows a table of node numbers and an 'Apply' button highlighted.

No	Node Number
1	3
2	4
3	5
4	6
5	731
6	732
7	1676
8	1677

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