PSC Bridge Wizard
Two Span Prestressed Concrete Composite Bridge
Overview

Bridge Overview:
- 2 span continuous PSC composite girder bridge
- Span length: 21m
- Width: 15m
- Dual carriageway: 6.7m
- System of units: kN, m

Tutorial overview:
- Prestressed Composite Bridge Wizard
  - Geometry, materials, sections, boundaries, static loads, moving loads, construction staging
- PSC Design
Outline

Pre Processing
Material Properties...5
Section Properties...6
PSC Bridge Wizard – Layout...12
PSC Bridge Wizard – Section...13
PSC Bridge Wizard – Tendon...14
PSC Bridge Wizard – Load...15
PSC Wizard Bridge – Construction Stage...18
Model Generation....19

Time Dependent Material Properties...20
Reinforcement...25
Moving Load Cases...28
Composite Section for C.S....30
Load Combinations...31

Post Processing
Results...32

Design
PSC Design...34
1. Tools>Preferences
2. Select Design/Load Code
3. Steel: AASHTO-LRFD12
4. RC: AASHTO-LRFD12
5. PSC: AASHTO-LRFD12
6. Rebar Material: ASTM(RC)
7. Material DB: Grade 60
8. Rebar Material: ASTM(RC)
9. Material DB: Grade 60
10. Click OK
Definition of Material Properties

1. Properties > Material Properties
2. Click Add
3. Type of Design: Concrete
4. Standard: ASTM(RC)
5. DB: Grade C6000
6. Click Apply
7. DB: Grade C5000
8. Click Apply
9. Type of Design: Steel
10. Standard: A416-270(Low)
11. DB: Tendon
12. Click OK
Definition of Section Properties

1. Select **Section** tab
2. Click **Add**
3. **DB/User**
4. **Solid Round**
5. Name: Pier
6. Tick **User**
7. D: 1.5m
8. Click **Apply**

Continued on next page…
Definition of Section Properties

1. **Solid Rectangle**
2. Name: Pier Cap
3. H:1.5m, B:1.6m
4. **Change Offset**
5. **Center-Top**
6. Click **OK**
7. Click **Apply**
8. Name: Diaphragm
9. H:1.7m, B:0.4m
10. **Change Offset**
11. Offset: **Center-Top**
12. Click **OK**
13. **Click OK**
14. **Close** Properties window

Close the Properties tab
1. Press PS and click Enter on the work window. It will open the Section Input Again.
2. Click Add.
3. Go to PSC Tab.
4. Choose PSC-I.
5. From Section Name choose AASHTO.
6. From Type Choose AASHTO TYPE6.
7. Name: Precast.
8. Check on Symmetry.
9. Change the offset to Center-Top.
10. Click Apply.

Continued on next page…
Definition of Section Properties

1. Go To **Composite Tab**
2. Name: Composite
3. Section Type : Composite-I
4. **Slab:**
   - Bc : 2.5, tc: .2, Hh: 0
5. In **Girder Information**, check on **Symmetry**.
6. Drag down: Besides **Size-I**, click on **Import**.
7. Select section **Precast** and click **Import**
8. **Material** :
9. Change the offset to **Center-Top**
10. Click **OK**
11. Click **Close**
Definition of Tendon Properties

1. Load.Temp./Prestress>Tendon Property
2. Click Add
3. Tendon Name: tendon
4. Tendon Type: Internal(Pre-Tension)
5. Material: Tendon
6. Click ... for Total Tendon Area
7. Strand Diameter: 15.2mm(1x7)
8. Number of Strands: 1
9. Click OK
10. Relaxation Coefficient: CEB-FIP 2010, 5%, Class 2 : Mean
11. Ultimate Strength: 1.86326e6kN/m²
12. Yield Strength: 1.56906e6kN/m²
13. Click OK
14. Close
1. **Structure**> Prestressed composite Bridge
2. Span Information: 2@21m
3. Deck Width: 15m
4. Spacing(a): 0.4m
5. Spacing(b): 0.2m
6. Elastic Link Length: 0.05m
7. Pier Cap Section: 2: Pier Cap
8. Length: 14.2m
9. Column Section: 1: Pier
10. Height: 5.5m
11. Spacing: 5m
12. **Save As** – the wizard file can be saved from within any tab of the wizard for later access or modifications

Continued on the next page...
1. **Section** tab
2. Deck Thickness: 0.25m
3. Girder Material: 2:C50/60
4. Girder Offset (m): -6.25, -3.75, -1.25, 1.25, 3.75, 6.25
5. End Support, Pier Support, Intermediate: **3:Diaphragm**
6. Transverse Deck Spacing: 1.5m
7. Span 1> Girder Section: **5:Composite**
8. Span 2> Girder Section: **5:Composite**
9. Tick off **Generate 10th points elements**

Continued on the next page...
1. **Tendon** tab
2. Input Tendon information as in screenshot
3. Jacking Stress: 1300000kN/m²
4. Click **Modify**
5. Select: **Span2**
6. Jacking Stress : 1300000kN/m²
7. Click **Modify**

Continued on the next page...
1. **Load** tab
2. b1: 0.5m
3. b2: 6.7m
4. b3: 0.6m
5. b4: 6.7m
6. b5: 0.5m
7. Weight Density: 25kN/m³
8. Thickness: 0.25m
9. Barrier: 2kN/m
10. Median Strip: 2.6kN/m
11. Wearing surface Weight Density: 23kN/m³
12. Define Moving Load Case
13. AASHTO LRFD
14. Click OK
15. Tick on **Live Load**
16. Define Traffic Lanes

Continued on the next page…
1. No. of lanes: 4
2. Distance (m): 2.6, 5.6, 9.4, 12.4
3. Click **OK**
4. Define Vehicles
5. Add Standard

Continued on the next page...
Define Standard Vehicular Load

1. Standard Name: AASHTO LRFD Load
2. Vehicular Load Type: HL-93TDM
3. Dynamic Load Allowance: 33%
4. Click Apply
5. Vehicular Load Type: HL-93TRK
6. Click OK
7. Click Close

Continued on the next page...
1. **Construction Stage** tab
2. Click **OK**

Continued on the next page...
The model generation is completed.
### Time Dependent Material Properties

1. **Properties** > **Creep/Shrinkage**
2. Click **Add**
3. **Code:** CEB-FIP(2010)
4. **Name:** C6000 C&S
5. **fck:** 40000
6. **Notional member size:** 1
7. **Show Results**
8. **Close**
9. Click **Apply**

Continued on the next page…
Time Dependent Material Properties

1. Name: C5000 C&S
2. $f_{ck}$: 34000
3. Click OK
4. Close

Continued on the next page...
Time Dependent Material Properties

1. Properties > Comp. Strength
2. Click Add
4. Name: C6000
5. $f_{ck}$: 48000
6. Redraw Graph
7. Click OK

Continued on the next page...
Time Dependent Material Properties

1. Click Add
3. Name: C5000
4. fck: 42000
5. Redraw Graph
6. Click OK
7. Close
1. Properties>Change Property
2. Click Select All
3. Click Apply
4. Properties>Material Link
5. Creep/Shrinkage: C6000 c&s
6. Comp. Strength: C6000
7. C6000 > Selected Materials
8. Click Add/Modify
9. Creep/Shrinkage: C5000 c&s
10. Comp. Strength: C5000
11. C5000 > Selected Materials
12. Click Add/Modify
13. Click Close
1. Properties>Section Manager>Reinforcements
2. Click 5: Composite
3. Guide Line: 0.05m
4. Tick Input Method B
5. Click in Starting Point box, then on the top left corner of the guide line
6. Click in the End Point box, then on the top right corner of the guide line
7. Num.: 15
8. Tick on Edge Bar
9. Dia: #14
10. Part: Part 2
11. Click Add

Continued on next page…
1. Similarly add reinforcement for the bottom part of the slab

2. Chose Starting Point and End Point from the bottom left corner of the slab guide line to the to the bottom right corner of the slab guideline

3. Num.: 15

4. Click Add

Continued on next page…
1. **Shear Reinforcement** tab
2. Tick on **Diagonal Reinforcement**
3. Pitch: 0.15m
4. Angle: 90
5. Click on...
6. Dia: \#4
7. Num.: 4
8. Click OK
9. Click **Apply**
10. **Copy Reinforcements to**
11. Select **Composite_1**
12. Pass to **Selected Section List**
13. Click OK
14. Click **Apply**
15. Click **Close**
Moving Load Cases

1. Load>Moving Load>Moving Load Cases
2. Add
3. Load Case Name: MLC
4. Check on Independent
5. Add

Continued on the next page…
Moving Load Cases

1. Vehicle Class: VL:HL-93TDM
2. Min. Number of Loaded Lanes: 1
3. Max. Number of Loaded Lanes: 4
4. Select all the lanes into Selected Lanes
5. Click OK
6. Click Add again
7. Same procedure for VL:HL93TRK
8. Click OK
1. Click on Lane Support Negative Moment in Load> Moving Load
2. In the Dialog Box, Select the Girder Group > Girder
3. Click Add
4. Click on Lane Support Reaction
5. Select the nodes on the Girders where Boundaries have been defined (end and continuous ones)
6. Click Apply
7. Click Close
Composite Section for C.S.

1. Load>Construction Stage>Composite Section For C.S.
2. Select item
3. Click Modify
4. Part1 Age: 28 days
5. Part1 h: 0.29m
6. Part2 Age: 10 days
7. Part2 h: 0.277m
8. Click OK
9. Close
Load Combinations

1. Results>Load Combination
2. Concrete Design tab
3. Auto Generation
4. Design Code: AASHTO-LRFD12
5. Click OK
6. Close

Run Analysis
Or Click on Analysis > Perform Analysis
1. Go to PostCS
2. Results>Deformations
3. Tick on Legend
4. Click Apply
5. For Load Case/Combinations: MVmax: MLC check forces
   Forces>Beam Diagrams
6. Tick on Solid Fill
7. Tick off Deform
8. Click Apply
Results

1. Now check the maximum Negative Moments
2. Load Case/Combinations: MVmin: MLC
3. Tick on Solid Fill
4. Tick off Deform
5. Click Apply
6. Results>Moving Tracer>Beam Forces/Moments
7. Key element: 270
8. Click Apply
9. Close
1. **PSC>Parameters**
2. Design Code: **AASHTO-LRFD12**
3. **Select All**
4. Click **OK**
5. **PSC>PSC Design Material**
6. Select Grade C6000
7. Rebar Code: **ASTM(RC)**
8. Grade of Main/Sub-Rebar: **Grade 60**
9. Slab Code: **ASTM(RC)**
10. Slab Grade: **Grade C5000**
11. Rebar Code: **ASTM(RC)**
12. Grade of Main/Sub-Rebar: **Grade 60**
13. Click **Modify**
14. **Close**

Continued on next page…
1. Right click **Section:5: Composite** and choose **Activate** or **Press F2** from **KeyBoard**

2. **Initial View**

3. **Top View**

4. **Hidden View** to come back to wireframe view.

5. **PSC>Design/Output Position>Design Position**

6. **Select Single**

7. **Select Girders**

8. Click **Apply**

9. **Close**

Continued on next page…
1. PSC>Design/Output Position>Output Position
2. Left click anywhere in the Model Window, then press Esc
3. Select Single
4. Select elements 185 and 193
5. Click Apply
6. Click Close
1. PSC>Perform Design
2. PSC>Results Tables>Check Flexure Strength
3. Click OK
4. Close window from X
5. Activate All
6. Iso View
7. PSC>PSC Results Diagram
8. Fill Type: Solid
9. Click Apply
10. Click on Excel Report to generate a detailed report for design.