Bridging Your Innovations to Realities

midas Civil
Prestressed Box Girder Bridge (FCM, FSM)
Contents:

Introduction
Modeling Features
Analysis Control
Results
Design Check
1. Introduction

Balanced Cantilever (FCM) Bridge
2. Modeling Features

Modeling of Bridge can be done in Midas Civil by:

- General Modeling Procedure
- Using wizards

Modeling of the bridge will be done by:
- General Procedure
- FSM wizard
- FCM wizard (Balanced Cantilever Bridge)

**Wizard can be applied to the bridge which is constructed through the typical construction process. Other types of bridges should be modeled by general modeling process.**
3. FCM Bridge Wizard

Construction Sequence of FCM Bridge:
Model -> Structure Wizard -> FCM Bridge

**Pier Section:** Define the section for the Pier

**Stage Duration:** stage duration for the construction of the balanced cantilever bridge

**Method:** Define the method of construction as Cast – in Situ or precast sections
3. FCM Bridge Wizard

Model -> Structure Wizard -> FCM Bridge

**Number of Piers:** Define the number of piers

**Radius:** Define the radius of the circle if the bridge is horizontally curved
Model -> Structure Wizard -> FCM Bridge
Model -> Structure Wizard -> FCM Bridge

**Pier Table:** Advanced option:

In the advanced option of the Pier table you can define the type of diaphragm as Single Diaphragm or double diaphragm and various parameters associated with them.
3. FCM Bridge Wizard

Model -> Structure Wizard -> FCM Bridge

**Key Segment:** The length of the key segment can be defined.

**Advanced** option: Here the length of the key segment as well as the elements straight (constant depth) to the left and right can be defined.
3. FCM Bridge Wizard

Model -> Structure Wizard -> FCM Bridge

FSM:

Here the FSM which is generally constructed as a part of balanced cantilever construction can be defined on the left and the right of the balanced cantilever construction.
3. FCM Bridge Wizard

**Model -> Structure Wizard -> FCM Bridge**

**Zones:**

Here the zones of the balanced cantilever bridge i.e. the profile of the bridge on the left and the right side of the pier can be defined.

**Advanced option:**

If the balanced cantilever bridge has different zone profile on different piers the it can be defined in the advanced option.
Pier Table Placing:

Here the construction delay between the piers can be defined, which would be responsible for varying time dependent effects on the structure.

If we define the time delay between the construction of pier 2 as 60 days, it means that pier 2 and the Corresponding elements (zones) are constructed 60 days after.
3. FCM Bridge Wizard

Model -> Structure Wizard -> FCM Bridge

Member Age:

Here you can define the activation age of the elements i.e. the age when the elements would become the structural Component of the bridge.
Model -> Structure Wizard -> FCM Bridge

Type 1 and Type 2:

If the default section is to be used for the modeling then Type 1 can be used but if the database section is to be used for modeling then Type 2 can be used.

When type 2 is selected the user must define the section as the PSC section in the Section definition of midas Civil.
3. FCM Bridge Wizard

*Model -> Structure Wizard -> FCM Bridge*

Include Wet concrete Load

If the wet concrete load (in the case of Cast-in-situ) is to be considered for the analysis.
3. FCM Bridge Wizard

Model -> Structure Wizard -> FCM Bridge

Parameters for Tendon Placement:

Define the geometric parameters of the location of the Tendons.
3. FCM Bridge Wizard

Model -> Structure Wizard -> FCM Bridge

Web Tendon:
Check on this option to define the web tendons.

Tendon Number:
Select the number of Tendons for each location
Model -> Structure Wizard -> FCM Bridge

Tendon Anchorage Number:

Define the anchorage position of the tendons.
4. FSM Bridge Wizard

Construction Sequence of FSM Bridge
4. FSM Bridge Wizard

*Model -> Structure Wizard -> FSM Bridge*

**Bridge Model Data Type:**

Select Type 1 to use the default section and tendon Parameters for the bridge.

Select Type 2 to use the section defined in the PSC tab Of the section definition.
4. FSM Bridge Wizard

Model -> Structure Wizard -> FSM Bridge

Apply Tapered Section:

![Diagram of FSM Bridge Model](image)
4. FSM Bridge Wizard

*Model -> Structure Wizard -> FSM Bridge*

**Bridge Geometry:**

Define the Bridge Material, span, radius if the bridge is horizontally curved, segment division per span and tapered section properties.
4. FSM Bridge Wizard

Model -> Structure Wizard -> FSM Bridge

Cold Joint:

Define the location of the cold joint i.e. the construction Joint and the anchorage for the cold joint.

A cold joint is the intersection between the end of one concrete pour and the beginning of a new pour.
4. FSM Bridge Wizard

*Model -> Structure Wizard -> FSM Bridge*

**Stage Duration:** Define the stage duration for the Construction.
4. FSM Bridge Wizard

Model -> Structure Wizard -> FSM Bridge

Section: Define the section properties for the Centre Of the span, Joint and the Diaphragm.
**4. FSM Bridge Wizard**

*Model -> Structure Wizard -> FSM Bridge*

**Tendon**: Define the tendon parameters.
4. FSM Bridge Wizard

Model -> Properties -> Section -> PSC section

The following types of PSC sections can be defined:

- n cell sections
- PSC I sections
- PSC T sections

Database sections:
- AASHTO Type I to Type VI
- Caltrans sections
- Italy
- UK
5. PSC Sections

Model -> Properties -> Section -> PSC section
5. PSC Sections

Determination of Torsional moment of inertia and effective shear area

Fixed Support

Meshed Element (Length – 20m)
5. PSC Sections - User Defined

**User defined section:**

The section can be imported from Autocad or can be modeled in the special module of midas Civil known as Section Property Calculator or SPC.

The following are the general steps:
1. Tools -> Section Property Calculator
2. Define the geometry of the section (or import from Autocad)
3. Generate the section in SPC
4. Export the section from SPC to midas section file.
5. Import the section in midas Civil
5. PSC Sections - User Defined

Important Considerations:

1. The units must be consistent with the Autocad File
2. For the definition of Tapered Sections, Number of Points must be same
5. PSC Sections – Tapered Sections

Tapered Sections for PSC Bridges:

There are 2 ways in which the tapered sections can be modeled in midas civil:

1. **Tapered section tab**: We can define the tapered section in the tapered tab of the section definition.

2. **PSC Bridge Wizard**: The PSC bridge wizard is a robust and easy method of making the tapered sections in midas civil.
5. PSC Sections – Tapered Sections

Tapered Sections for PSC Bridges:

1. Tapered Section Tab:

Model-> Properties -> Section -> Tapered Tab

The sections can be defined in tapered section tab or the Sections can be defined in the PSC tab and imported as I and J end of the tapered section.
5. PSC Sections – Tapered Sections

Tapered Sections for PSC Bridges:

1. Tapered Section Tab:

   Model -> Properties -> Tapered Section Group
5. PSC Sections – Tapered Sections

Tapered Sections for PSC Bridges:

2. PSC Bridge Wizard

_Model-> Structure Wizard -> PSC Bridge -> Section & Reinforcement_

This feature obviates the use of the tapered section and the depth of the section be directly defined in the longitudinal direction of the bridge.
5. PSC Sections – Tapered Sections

Tapered Sections for PSC Bridges:

2. PSC Bridge Wizard

With PSC bridge wizard not only the depth but the width, web thickness and flange thickness can be varied.

![Diagram of PSC Bridge Wizard](image-url)
If the curve type is a curved line, then the symmetrical plane must be wisely selected otherwise the profile may not be what is desired.
5. PSC Sections – Tapered Sections

<table>
<thead>
<tr>
<th>Ref. Line</th>
<th>Distance (mm)</th>
<th>Dimension (mm)</th>
<th>Curve Type</th>
<th>Exp.</th>
<th>Sym. Plane Distance (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 S1</td>
<td>0.00</td>
<td>2000.00</td>
<td>Curved line</td>
<td>2.0</td>
<td>0.00</td>
</tr>
<tr>
<td>2 S2</td>
<td>0.00</td>
<td>4000.00</td>
<td>Curved line</td>
<td>2.0</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
6. Reinforcement to PSC Sections

Reinforcement in the PSC sections:

The reinforcements in the PSC sections can be defined in the section manager:

Model -> Properties -> Section Manager-> Reinforcements
6. Reinforcement to PSC Sections

Reinforcement in the PSC sections:

The reinforcements defined in the PSC sections can be used to calculate the stiffness of the Section.

*Analysis -> Main Control Data*
7. Tendon Profile Definition

Tendon Profile can be defined in 2D and 3D as per the following reference axis:

- **Straight**: the x-direction (the reference line from which tendon coordinates are defined) of TCS for tendon placement is considered as a straight line.

- **Curve**: the x-direction (the reference line from which tendon coordinates are defined) of TCS for tendon placement is considered as a curved line.

- **Element**: Tendon location is converted into Element Coordinate System and applied.
7. Tendon Profile Definition

X axis rotation angle:

With the help of this function a tendon defined at the centroid of the Section can be assigned to the web with rotation and offset.
7. Tendon Profile Definition

Element Reference Axis:

- The Profile is define with respect to the element. So if the element profile is curved, this option is used as:

Then:
1. The total length of the tendon in red (shown by midas in tendon weight table) would be more than actual and tendon in yellow will be less than actual.

2. The ‘Element’ option does not consider friction loss due to horizontal curvature. So, if you need to consider the friction loss due to horizontal curvature, you should use the ‘Curve’ option.
8. Tendon Profile Export/Import with dxf

Tendon Export/Import with Autocad (*.dxf):

The tendon can be exported/imported with Autocad in midas civil

1. **Export**: For exporting the tendon, select the tendon in the Tendon definition dialogue box and then hit export to dxf.
8. Tendon Profile Export/Import with dxf

Tendon Export/Import with Autocad (*.dxf):

The tendon can be exported/imported with Autocad in midas civil

2. Import: For importing the tendons from the autocad, the tendon profile generator is used.

The following steps must be followed:
1. Tools-> Tendon Profile Generator
2. Open the dxf file in which the tendon profile is defined
3. Select the tendon profile and define the required parameters
4. Click -> Add
5. Click -> Ok, the text file will be generated
6. Copy the contents of the text file and paste to MCT and then hit run.
9. Effective Width Scale Factor

**Effective Width Scale Factor:**

Effective Width Scale Factor can be used to consider the effect of shear lag.

The effective width scale factor can be defined by:

1. **Automatic definition:** The automatic definition of the Effective width scale factor is supported for AASHTO LRFD code and can be defined in PSC Bridge wizard.

2. **Manual definition:** The manual definition of the effective width scale factor can be done in effective width scale factor as the boundary.
10. Loading

Load Data:

The following are the loads that are supposed to act on the bridge during the construction:

- Self weight
- Weight of the wet concrete
- Form Traveler weight
- Prestress Loads

Additional loads include:

- Settlement Effects
- Creep and Shrinkage effects
- Temperature Loads
10. Loading

**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.
10. Loading

How to Model Wet Concrete Load:

1. Elements Load must be activated in the construction stage in which it is casted by nodal loads or moments if needed.

2. When the scaffolding is removed, nodal load defined in 1 must be deactivated and element must be activated. The self weight function will now automatically consider the weight of the element.

3. Self Weight must be activated at day 1 of the 1st constructions stage.

CS 8 – Elements casted on day 7

CS 9
10. Loading

CS 8: Activating the wet concrete and scaffolding weight on day 7

CS 9: Activating the Element
10. Loading

**Form Traveler Load:** Form traveler load can be defined as the nodal loads.

**Temperature Loads:** Can be defined as system temperature or the beam section temperature.

Beam Section Temperature can be defined in

\[ \text{Loads} \rightarrow \text{Temperature Loads} \rightarrow \text{Beam Section Temperature} \]
10. Loading

**Time Load:**

In midas Civil you can apply the time load to a structure to impose time load to specific members to reflect the time dependent material properties relative to the contiguous members.

Construction Schedule of Pier 1: \(T = 0\) days to \(T = 180\) days.
Construction Schedule of Pier 2: \(T = 60\) days to \(T = 240\) days.
10. Loading

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

**TYPE:** The specified displacements can be provided.
CS 9 : Deactivating the load
1. Tendon Tension Loss Effect

Check on to reflect the effect of prestressing tension loss of tendons due to creep and shrinkage.
2. Rebar Confinement Effect

Check on to reflect the effect of rebar confinement for creep and shrinkage.
3. Stress Decrease at Lead Length Zone

Select a method of computing stresses over a transfer length in a post-tension model. This feature is applicable only when a Transfer Length is entered in the Tendon Profile dialog.

If Composite section for Construction Stages is used this function cannot be applied.
4. Change of Beam Section Properties with Tendons

Select whether to consider the presence of tendons for calculating section properties.
1. Secondary effects of Prestressing, Creep & Shrinkage

Tendon Primary represents member forces caused by tendon prestress forces.

Tendon Secondary represents member forces resulting from tendon prestress forces acting in an indeterminate structure
2. PSC Stresses

PSC stresses can be obtained at 10 locations:

Components

- Axial stress
- Bending stress
- Shear stress
- Principal stress
### 3. Tendon Time Dependent Loss Table

<table>
<thead>
<tr>
<th>Tendon Group</th>
<th>Top-P 1-1</th>
<th>Stage</th>
<th>CS13</th>
<th>Apply</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>17</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>18</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>19</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>22</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### 3. Tendon Time Dependent Loss Table

<table>
<thead>
<tr>
<th>Elem</th>
<th>Part</th>
<th>Stress (After Immediate Loss) : A (tonf/m²)</th>
<th>Elastic Deform. Loss : B (tonf/m²)</th>
<th>Stress(Elastic Loss)/Stress(Immediate Loss)</th>
<th>Creep/Shrinkage Loss (tonf/m²)</th>
<th>Relaxation Loss (tonf/m²)</th>
<th>Stress(All Loss)/Stress(Immediate Loss)</th>
<th>Effective Num.</th>
</tr>
</thead>
</table>

Stress in the tendon after instantaneous loss.

**Post – Tensioned Beams:**
Instantaneous loss includes:
- Friction Loss
- Anchorage Loss

**Elastic Shortening loss is considered to have taken place already during prestressing**

**Pre – Tensioned Beams:**
- Elastic Shortening Loss only due to prestressing (not due to self weight)
3. Tendon Time Dependent Loss Table

Anchorage Loss

\[ \Delta P = 2p l_{set} \]

\[ l_{set} = \sqrt{\frac{A_p \cdot E_p \cdot \Delta l}{p}} \]

Where: \( p \) is the frictional resistance per unit length determined the friction loss between the anchorage and quarter length of the beam and dividing it by \( L/4 \)

Friction Loss:

\[ P_x = P_0 e^{-(\mu \alpha + kl)} \]
Other types of elastic shortening losses, which are caused by subsequent loadings (self weight, live loads, creep, shrinkage, etc.) after the prestressing force is applied, are included.

Elastic Shortening due to prestressing of other tendons is included:
### 3. Tendon Time Dependent Loss Table

<table>
<thead>
<tr>
<th>Elem</th>
<th>Part</th>
<th>Stress (After Immediate Loss) : A (tonf/m²)</th>
<th>Elastic Deform. Loss : B (tonf/m²)</th>
<th>Stress(Elastic Loss) Stress(Immediate Loss)</th>
<th>Creep/Shrinkage Loss (tonf/m²)</th>
<th>Relaxation Loss (tonf/m²)</th>
<th>Stress(After All Loss)/ Stress(After Immediate Loss)</th>
<th>Effective Num.</th>
</tr>
</thead>
</table>

Tendon Loss due to creep and Shrinkage
### 3. Tendon Time Dependent Loss Table

<table>
<thead>
<tr>
<th>Elm</th>
<th>Part</th>
<th>Stress (After Immediate Loss): A (tonf/m²)</th>
<th>Elastic Deform. Loss: B (tonf/m²)</th>
<th>Stress(Elastic Loss)/Stress(Immediate Loss)</th>
<th>Creep/Shrinkage Loss (tonf/m²)</th>
<th>Relaxation Loss (tonf/m²)</th>
<th>Stress(After All Loss)/Stress(After Immediate Loss)</th>
<th>Effective Num.</th>
</tr>
</thead>
</table>

Tendon Loss due to Relaxation of Tendon

When CEB-FIP is selected

Enter the final loss ratio due to steel relaxation. Prestress loss due to steel relaxation is determined from the following equation:

\[
\Delta f_s = f_{si} \cdot \gamma \cdot (K_i - K_{i-1})
\]

where,

- \( f_{si} \): initial stress
- \( \gamma \): final loss ratio due to steel relaxation
- \( K_i \): progress of steel relaxation at the last time step
4. Tendon Elongation

Tendon Elongation:

Results -> Result Tables -> Tendon -> Tendon Elongation

Tendon Elongation required in the field for obtaining a desired level of prestressing can be obtained from the program. The total elongation which the tendon has to achieve during stressing should be calculated as:

\[ \Delta l_{tot} = \Delta l_p + \Delta l_c + \Delta l_{sl} + \Delta l_e \]

- \( \Delta l_p \): elongation of the tendon
- \( \Delta l_c \): elastic deformation of the concrete (shortening must be treated as a positive value)
- \( \Delta l_{sl} \): sum of anchor plates impressions and dead end wedge slip according anchorage type applied
- \( \Delta l_e \): elongation of the prestressing steel in the jack and seating device (if applicable)

Tendon elongation calculated by the program consists of \( \Delta l_p, \Delta l_c, \) and \( \Delta l_{sl} \). The summation of \( \Delta l_p \) and \( \Delta l_{sl} \) is shown in the tendon elongation. And, \( \Delta l_c \) is separately shown in the element elongation.
5. FCM Camber

FCM Camber:

In structural design, camber refers to a pre-fabricated / pre-designed shape which enables the members maintain a stress-free state in order for the member to reach a targeted shape at a specified time.

<table>
<thead>
<tr>
<th>Formula</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\text{Camber} = \delta_{\text{camber}} = \text{alignment} + \sum \delta_i \times (-1) + \delta_a$</td>
<td>- Discontinuous point of saw tooth shape exists since each segment upon creation has only the load influence accumulated (the load influence has not been accumulated before the creation of the segments).</td>
</tr>
<tr>
<td>$\delta_{\text{camber}}$</td>
<td>- Turning the thoughts, for creation of each segment, it means camber that has to be constructed by segment for a target shape.</td>
</tr>
<tr>
<td>$\delta_a$</td>
<td>- If alignment (plane, vertical section) during modeling is included, the ‘alignment’ is excluded.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Formula</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\text{Pre-Camber} = \delta_{\text{N,pre-camber}} = \delta_c + \sum \delta_i = 1 \ldots N$</td>
<td>- Real displacement in which load influence is accumulated from specified stage (N) to camber and specified stage (N).</td>
</tr>
<tr>
<td>$\delta_{\text{N,pre-camber}}$</td>
<td>- The stage time inconsistency of the created segments has been cleared since the stage accumulative displacement has been deducted from camber to specified stage (N). As a result, displacement of stage state occurred.</td>
</tr>
<tr>
<td>$\delta_c$</td>
<td>- Calculated as [camber + accumulated influence of load at specified stage (N)].</td>
</tr>
</tbody>
</table>
5. FCM Camber

Using the analysis results, each stage of the camber graph are drawn below according to 2 camber distribution tables.

<table>
<thead>
<tr>
<th>terminology</th>
<th>Analysis result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stage</td>
<td>Stage</td>
</tr>
</tbody>
</table>

Accumulative Displacement

Final Displacement
- Same as above

Additional Camber
- will be calculated and added according to Geometry Engineer’s (erection engineer) experience and judgment

Camber
- Seg.1 camber gives step 1 support height (fabrication shape=0, Add. camber=0 assumption)

Pre-Camber
-
1. Design Parameters

Midas Civil can design the PSC sections as per the AASHRO LRFD code and EUROCODE. The following steps must be performed to obtain the design results:

Steps:

1. Result -> Combinations -> define load combinations under the Concrete Tab for design (by defining the strength and serviceability combinations)
2. Design -> PSC Design -> PSC Design Parameters and select the requisite parameters for design.
3. Design -> PSC Design -> PSC Design Materials and specify the material
4. Design -> PSC Design -> position for PSC design and select the elements for design
5. Design -> PSC Design -> Position for PSC output and select the elements for output.
9. Change of Beam Section Properties with Tendons

PSC Design Parameters:

*Design - > PSC Design - > PSC Design parameters.*

**Tendon Type:**
Define the tendon type for the calculation of stress.

<table>
<thead>
<tr>
<th>Condition</th>
<th>Stress-Relieved Strand and Plain High-Strength Bars</th>
<th>Low Relaxation Strand</th>
<th>Deformed High-Strength Bars</th>
</tr>
</thead>
<tbody>
<tr>
<td>Immediately prior to transfer</td>
<td>0.70 (f_{psu})</td>
<td>0.75 (f_{psu})</td>
<td>—</td>
</tr>
<tr>
<td>At service limit state after all losses (f_{psu})</td>
<td>0.80 (f_{psu})</td>
<td>0.80 (f_{psu})</td>
<td>0.80 (f_{psu})</td>
</tr>
<tr>
<td>Post-Tensioning</td>
<td>0.90 (f_{psu})</td>
<td>0.90 (f_{psu})</td>
<td>0.90 (f_{psu})</td>
</tr>
<tr>
<td>Prior to seating—short-term (f_{psu}) may be allowed</td>
<td>0.70 (f_{psu})</td>
<td>0.70 (f_{psu})</td>
<td>0.70 (f_{psu})</td>
</tr>
<tr>
<td>At anchorages and couplers immediately after anchor set</td>
<td>0.70 (f_{psu})</td>
<td>0.74 (f_{psu})</td>
<td>0.70 (f_{psu})</td>
</tr>
<tr>
<td>Elsewhere along length of member away from anchorages and couplers immediately after anchor set</td>
<td>0.80 (f_{psu})</td>
<td>0.80 (f_{psu})</td>
<td>0.80 (f_{psu})</td>
</tr>
</tbody>
</table>
2. Design Parameters

PSC Design Parameters:

*Design - > PSC Design - > PSC Design parameters.*

Bridge Type:
Define whether the bridge is fully or partially PSC as Defined in the AASHTO LRFD code.

1. Fully PSC: Resistance only from prestressing tendons and concrete considered. No cracking is permitted.

2. Partially PSC: Resistance from prestressing tendons, reinforcement and concrete considered. Cracks within a control limit are allowed in this type of bridge.
1. Design Parameters

PSC Design Parameters:

*Design - > PSC Design - > PSC Design parameters.*

**Construction Type:**

Select the construction type as Segmental or Non Segmental
1. Design Parameters

PSC Design Parameters:

Design - > PSC Design - > PSC Design parameters.

Flexural Strength:

Code: Clause 5.7.3.2 of AASHTO LRFD-07

Strain Compatibility: For more precise calculations moment resistance is calculated iteratively using strain compatibility approach. (clause 5.7.3.2.5 of AASHTO LRFD-07)
Transverse Moving load analysis can be performed in midas Civil.

Transverse analysis wizard can be used to perform the transverse analysis by selecting a section from the bridge.

The following steps must be performed for the same:
1. Selecting the required element for generating the transverse analysis model
2. Defining the parameters such as loads and reinforcement.
3. Defining the moving load parameters
4. Opening the generated model
Full 3D modeling in midas FEA

Warping Effect Check and 3D Finite Element Modeling in FEA

Midas Civil and Midas FEA can be used in tandem with each other for performing global and detailed analysis.

Midas Civil model can be exported to FEA for generating the solid model and meshing and Detailed analysis can be performed.