Advanced Application 9

Single Span Composite Precast Beams & Deck Bridge

Civil

CONTENTS

Summary

This example shows the analysis of a 120-ft single span AASHTO bulb-tee beam bridge with no skew, according to the AASHTO LRFD specifications. The structure consists of six precast, pretensioned beams spaced at 9'-0" centers. Beams are designed to act compositely with the 8-in. cast-in-place concrete deck to resist all superimposed dead loads, live loads and impact. A ½ in. wearing surface is considered to be an integral part of the 8-in. deck. A 2 in. wearing surface will be installed in the future.

This example is similar to the one presented in the PCI Bridge Design Manual.

Bridge Dimensions:

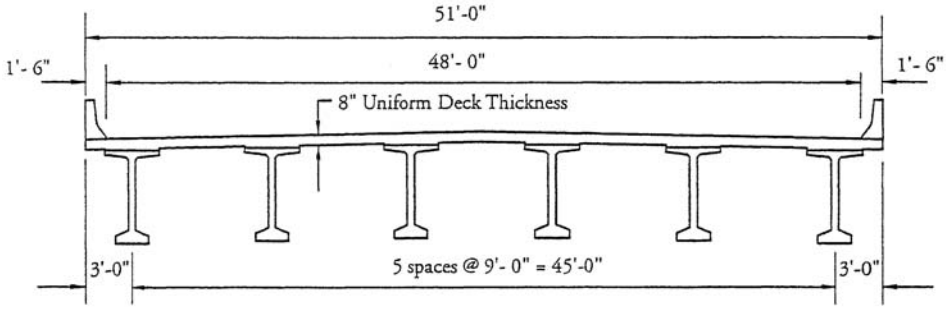


Figure 1a Bridge Cross-Section

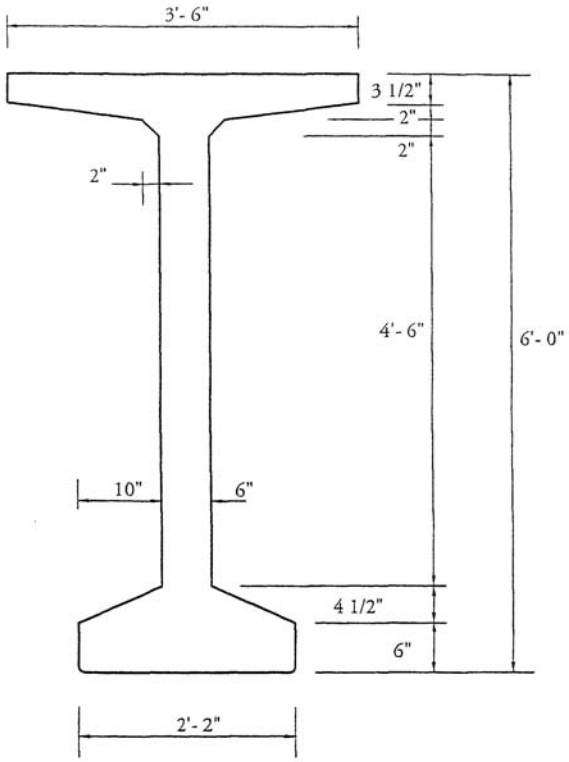


Figure 1b Precast Beam Dimensions

Materials:

Concrete – Deck and Cross Beams

ASTM C4000	
Modulus of elasticity:	3834 ksi
Poisson's ratio:	0.2
Density:	0.150 kcf
Concrete strength (28-day)	4.0 ksi
Deck total thickness	8.0 in
Deck structural thickness	7.5 in
Cross beam depth	7.5 in (same as thickness of deck)
Interior cross beam width	10 ft (c/c distance between adjacent cross-beams)
End cross beam width	5 ft

Concrete – Precast Beams

ASTM C6500	
Modulus of elasticity:	4888 ksi
Poisson's ratio:	0.2
Density:	0.150 kcf
Concrete strength (28-day)	6.5 ksi

Steel – Prestress Tendons: ½ in. dia., seven-wire, low-relaxation

Modulus of elasticity:	28500 ksi
Ultimate strength:	270.0 ksi
Yield strength:	243.0 ksi

Loads:

Dead Load

Concrete deck

Exterior PC Beam	(8/12ft) (7.5ft) (0.150kcf) = 0.750 kip/ft
Interior PC Beam	(8/12ft) (9.0ft) (0.150kcf) = 0.900 kip/ft
Haunch above beam	(0.5/12ft) (3.5ft) (0.150kcf) = 0.022 kip/ft

Barrier

(2 barriers)(0.300 kip/ft)/(6 beams) = 0.100 kip/ft

Future wearing surface

(2/12ft)(0.15kcf)(48ft)/(6 beams) = 0.200 kip/ft

Prestress Load

Stress in tendon before transfer = 202.50 ksi (75% of ultimate strength)
Assumed initial loss due to elastic shortening = 9.2 % (18.6 ksi)
Therefore: Stress in tendon after transfer = 183.90 ksi

Moving Loads (AASHTO LRFD)

HL-93

File Opening and Preferences Setting

File / ***New Project***
File / ***Save* (PSC Single Span)**

Tools / ***Unit System***
Length>**in**; Force (Mass)>**kips** ↵

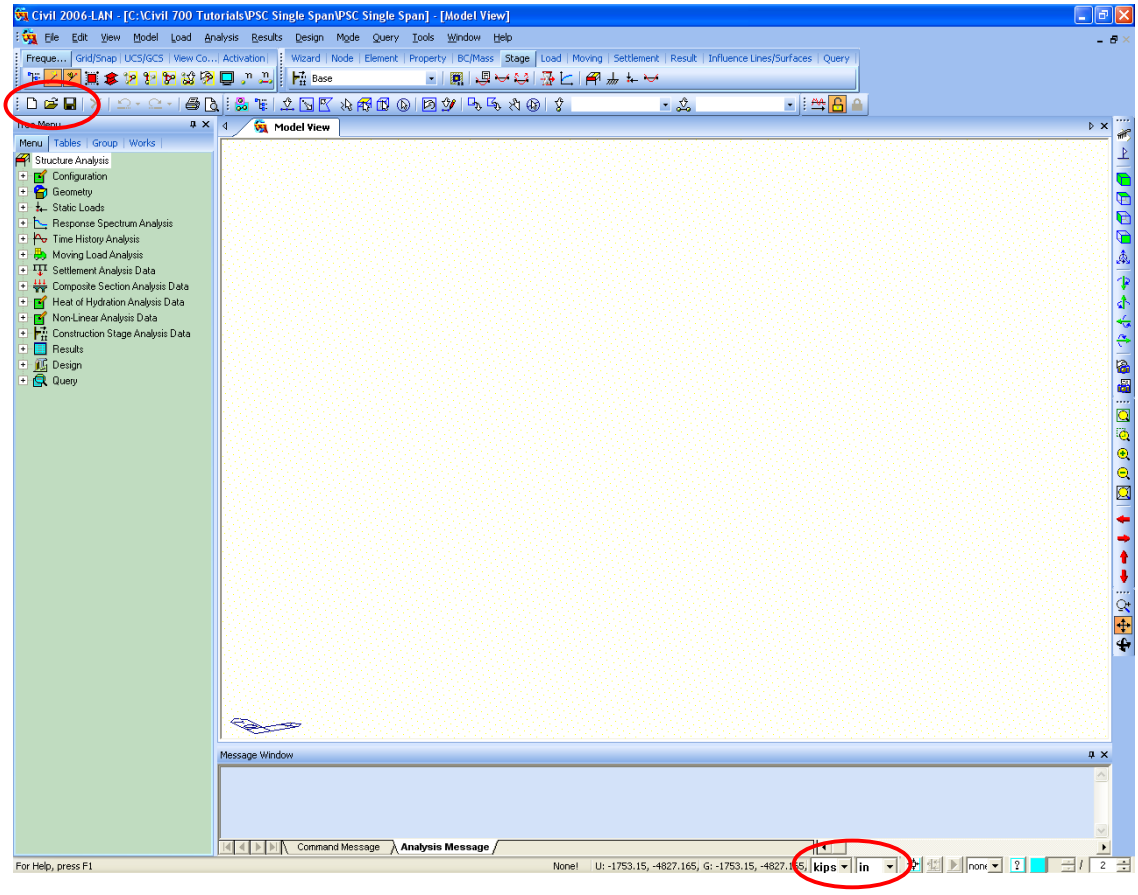


Figure 2 File Opening and References Setting

Material and Section Properties

In this section the materials and sections used to model the structure are defined.

Material Properties:

The following materials are defined:
Deck
Precast Beams
Tendons
Cross Beams.

In the **Tree Menu:** Geometry / Properties / **Material** ↵
Properties dialog box>***Material*** tab> Click Add
Name>**Deck**
Type of Design>**Concrete**
Standard>**None**
Modulus of Elasticity>**3834** (calculated for Grade C4000 as per AASHTO formula)
Poisson's Ratio>**0.2**
Weight Density>**0** ↵
Click Apply

Note:
Deck weight density is assigned a value of zero, because we want to treat deck weight as beam load, as opposed to self weight calculated automatically by the program after Composite Section for Construction Stage is created (refer page 40).

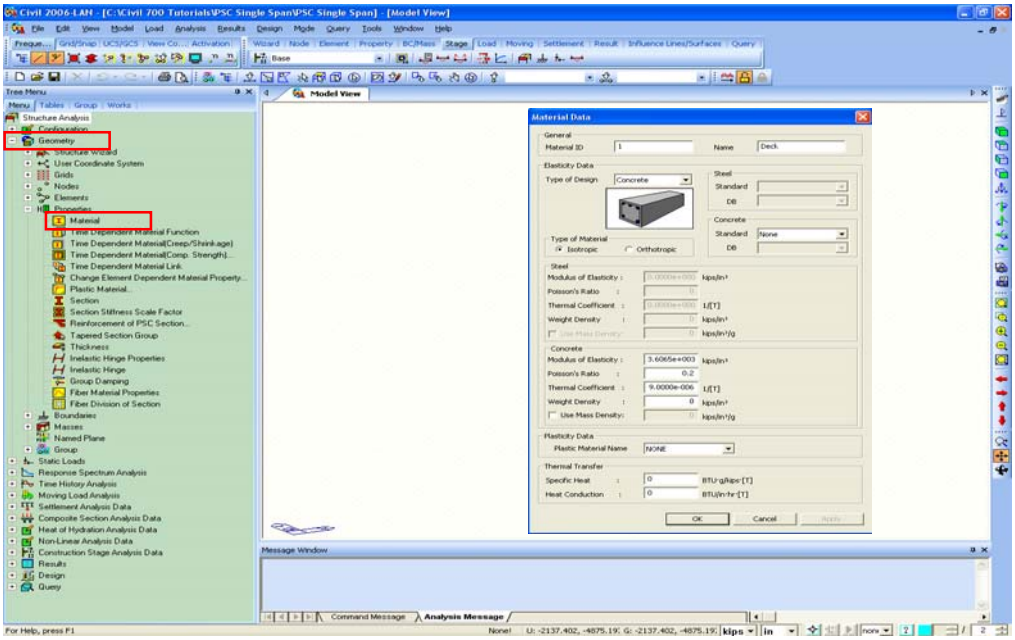


Figure 3 Material Data Window

Name> **Precast Beams**
Type of Design>**Concrete**
Standard>**None**
Modulus of Elasticity>**4888** (calculated for Grade C6500 as per AASHTO formula)
Poisson's Ratio **(0.2)**
Weight Density **(8.681e-005)** kips/in^3 (i.e., 0.150 kcf) ↵
Click Apply

Name> **Tendon**
Type of Design>**User Defined**
Standard>**None**
Modulus of Elasticity>**28500**
Poisson's Ratio **(0.3)**
Weight Density **(8.681e-005)** kips/in^3 (i.e., 0.150 kcf) ↵
Click Apply

Note:
In this tutorial the density of tendons is considered to be the same as the density of concrete, since it will be easier to compare results with the example presented in the PCI Bridge Design Manual. The example in the PCI Bridge Design Manual does not consider separate density for tendons.)

Name> **Cross Beams**
Type of Design>**Concrete**
Standard>**None**
Modulus of Elasticity>**3834** (calculated for Grade C4000 as per AASHTO formula)
Poisson's Ratio **(0.2)**
Weight Density **(0)** ↵
Click OK

Note:
Weight density of cross beams has been assigned a value of zero because these are fictitious beams used only for generating moving loads (refer to page 32).

Click Close

Time Dependent Material Properties:

In the **Tree Menu:** Geometry / Properties / **Time Dependent Material (Creep / Shrinkage)** ↵

Time Dependent Material (Creep / Shrinkage) dialog box > Click Add
Name>**CEB-FIP**
Code>**CEB-FIP**
Compressive strength of concrete at the age of 28 days>**6.5**
Relative Humidity of ambient environment (40–99)>**70**
Notational size of member>**10** (This is a provisional value that will be replaced later after calculation by the program).
Type of cement>**Normal or rapid hardening cement (N, R)** ↵
Age of concrete at the beginning of shrinkage>3
Click Show Result...
Click Close
Click OK
Click Close

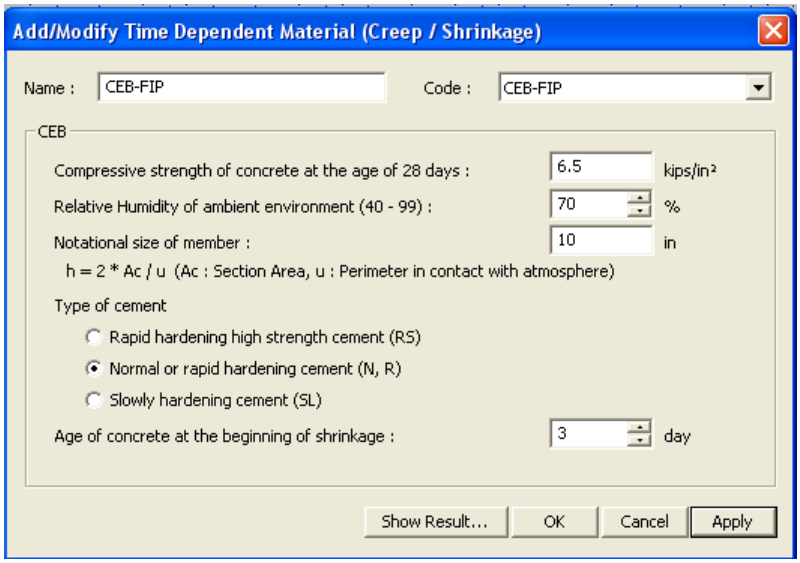


Figure 4 Creep and Shrinkage Data

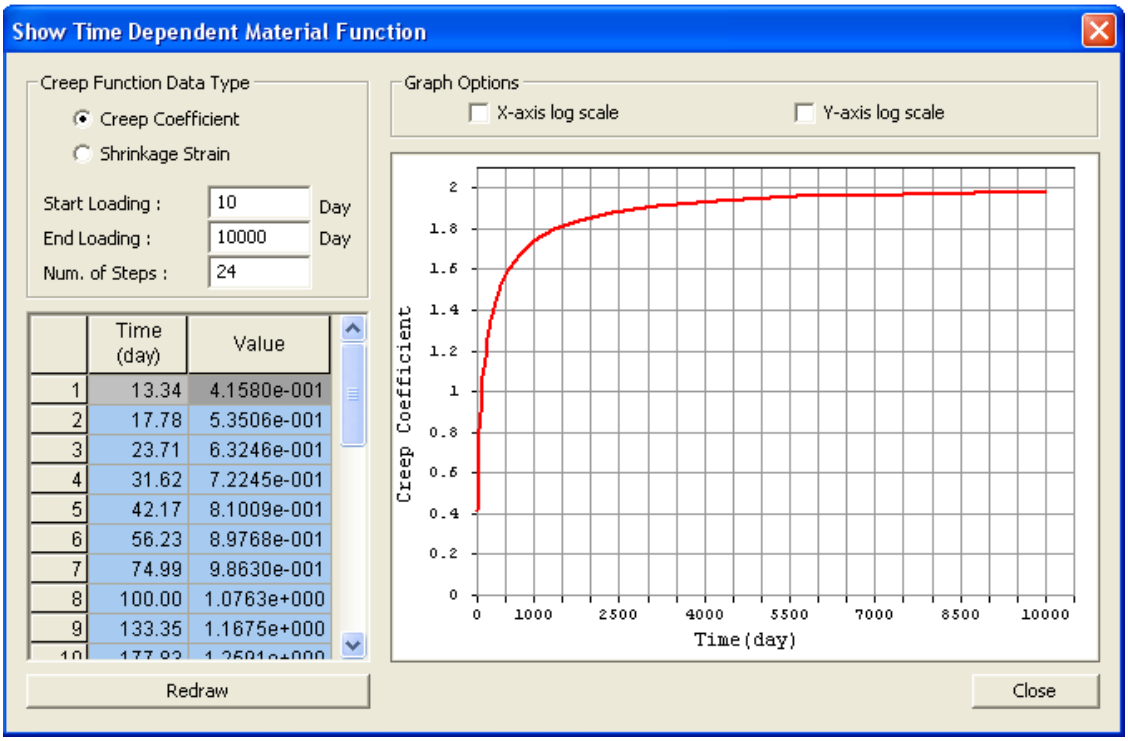


Figure 5 Creep Coefficient

In the **Tree Menu:** Geometry / Properties / **Time Dependent Material (Comp. Strength)** ↵

Time Dependent Material (Comp. Strength) dialog box > Click Add
Name>**C6500**
Type>**Code**
Development of Strength>Code>**CEB-FIP**
Concrete Compressive Strength at 28 Days>**6.5**
Cement Type(s)> **N, R: 0.25**
Click Redraw Graph
Click OK
Click Close

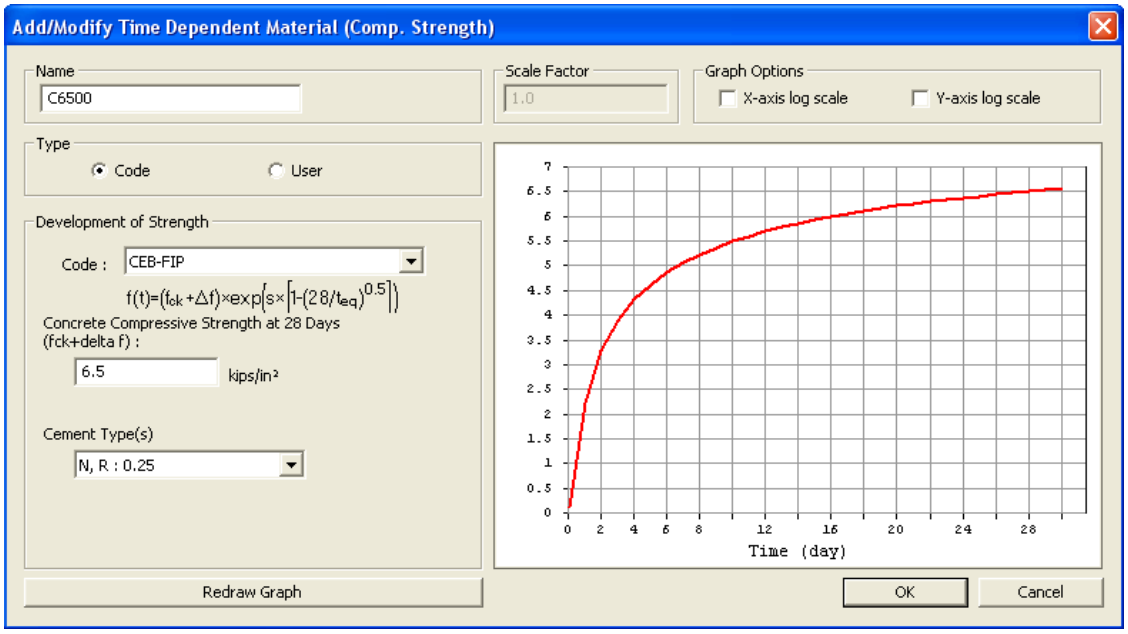


Figure 6 Compressive Strength Data

In the **Tree Menu:** Geometry / Properties / **Time Dependent Material Link** ↵
Time Dependent Material Type>Creep/Shrinkage>**CEB-FIP**
Time Dependent Material Type>Comp. Strength>**C6500**
Select Material to Assign>Materials>**2:Precast Beams**
Click >
Click Add / Modify
Click Close

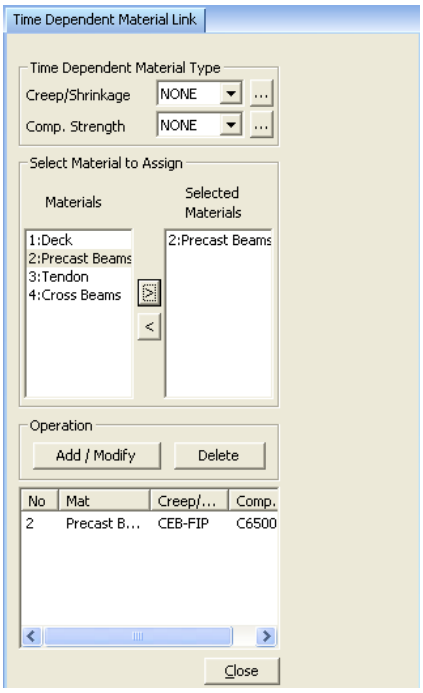


Figure 7 Time Dependent Material Link Window

Section Properties:

The following sections are defined:
Interior Precast Beams
Exterior Precast Beams
End Cross Beams
Interior Cross Beams

Interior and exterior precast beams differ from each other in their effective deck width.
Interior and end cross beams differ from each other in their width.

In the **Tree Menu:** Geometry / Properties / **Section** ↵
Properties dialog box>***Section*** tab> Click Add
Click ***Composite*** tab
Name>**Interior Precast Beams**
Section Type**>Composite-I**
Slab Width**>612**
Girder>Num>**6**
Girder>CTC>**108** (center-to-center beam spacing)
Slab>Bc>**108**
Slab>tc>**7.5**
Slab>Hh>**0.5**

By comparing the section shown in the **PSC Viewer** with the cross section of the Interior PC Beams, determine the points (J1, JL1…JL4, JR1…JR4) that are required to define the section.

Girder>**J1, JL1** (on)
Girder>**Symmetry** (on)

Scroll down the **Girder** window and enter the following section geometry data:

H1	72.0
HL1	3.5
HL2	4.0
HL2-1	2.0
HL3	54.0
HL4	4.5
HL5	6.0
BL1	3.0
BL2	21.0
BL2-1	16.0
BL4	13.0

Table 1.1 Girder Section Geometry Data

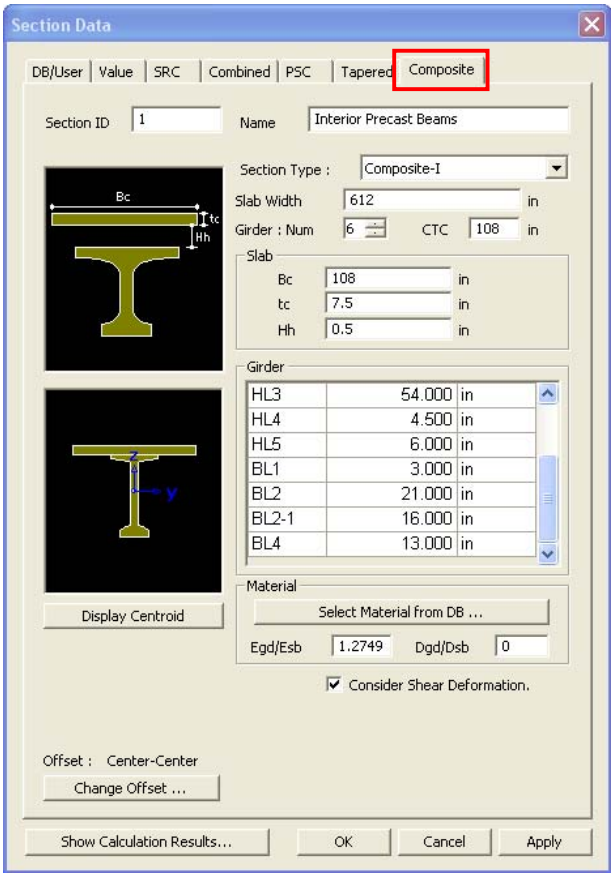


Figure 8 Section Data Window

Egd/Esb>**1.2749**
Dgd/Dsb >**0**
Consider Shear Deformation>(on)
Click OK

Note:
Egd/Esb represents the ratio of Modulus of Elasticity for both types of concrete – girder and slab. Therefore, Egd/Esb = 4888/3834 = 1.2749.

Dgd/Dsb is the ratio of unit weight for both types of concrete – girder and slab. It has been assigned a value of zero because we want to treat deck weight as beam loads as opposed to self weight calculated automatically by the program.

Properties dialog box >***Section*** tab>**ID 1 (Interior Precast Beams)**
Click Copy
Click **ID 2 (Interior Precast Beams)**
Click Modify...
Name>**Exterior Precast Beams**
Slab>Bc>**90** (this is the only difference between the two sections).
Click OK

Properties dialog box **>*Section*** tab>Click Add
Click ***DB/User*** tab
Name>**End Cross Beams**
Click **User**
Select **Solid Rectangle** (

)
H>**7.5**
B>**60**
Click OK

Properties dialog box **>*Section*** tab>Click Add
Click ***DB/User*** tab
Name>**Interior Cross Beams**
Click **User**
Select **Solid Rectangle** (

)
H>**7.5**
B>**120**
Click OK
Click Close

Note:
The depth of the Cross Beams is taken as the thickness of deck slab and width of the Cross Beams is taken as the center-to-center distance between the Cross Beams.

Structural Modeling Using Nodes and Elements

Tools / ***Unit System***
Length>**ft**; Force (Mass)>**kips** ↵

Click **Top View**

In the **Tree Menu:** Geometry / Nodes / **Create** ↵
Coordinates (x,y,z)>**0,0,0**
Copy>Number of Times>**5**
Copy>Distances (dx,dy,dz)**>0,9,0**
Click Apply
Click Close

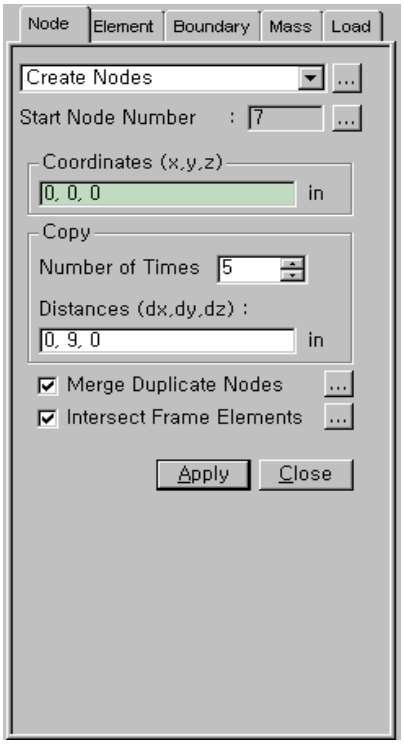


Figure 9 Create Nodes Window

Precast Beams:

Click **Auto Fitting**
Click **Node Number**

In the **Tree Menu:** Geometry / Elements / **Extrude** ↵
Select Window >Nodes 1 and 6
Extrude Type>**Node→ Line Element**
Element Attribute>Element Type>**Beam**
Material>**2: Precast Beams**
Section>**2: Exterior Precast Beams**
Generation Type>**Translate**
Translation>dx,dy,dz>**10, 0, 0**
Number of Times>**12**
Click Apply

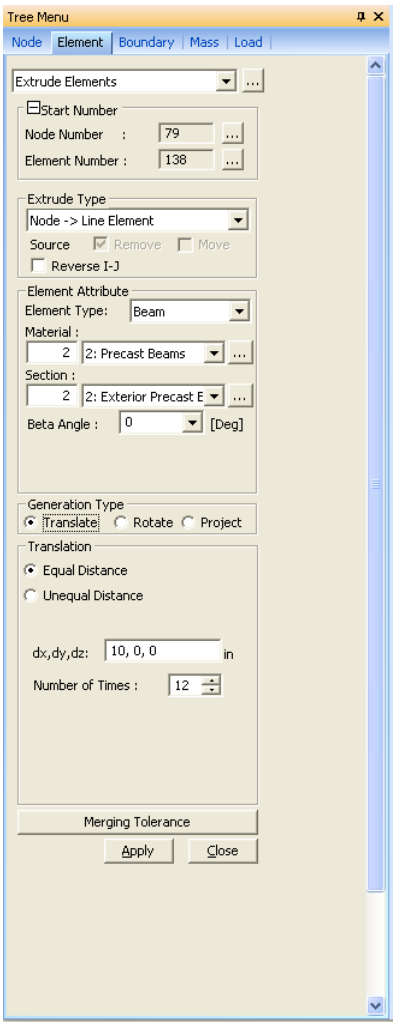


Figure 10 Extrude Elements Window

Select Window >Nodes 2to5
Extrude Type>**Node→ Line Element**
Element Attribute>Element Type>**Beam**
Material>**2: Precast Beams**
Section>**1: Interior Precast Beams**
Generation Type>**Translate**
Translation>dx,dy,dz>**30, 0, 0**
Number of Times>**4**
Click Apply

Cross Beams:

Select Window >Nodes 1 and 29
Extrude Type>**Node→ Line Element**
Element Attribute>Element Type>**Beam**
Material>**4: Cross Beams**
Section>**3: End Cross Beams**
Generation Type>**Translate**
Translation>dx,dy,dz>**0, 9, 0**
Number of Times>**5**
Click Apply

Select Window >Nodes 7to27by2
Extrude Type>**Node→ Line Element**
Element Attribute>Element Type>**Beam**
Material>**4: Cross Beams**
Section>**4: Interior Cross Beams**
Generation Type>**Translate**
Translation>dx,dy,dz>**0, 9, 0**
Number of Times>**5**
Click Apply
Click Close

Toggle on the **Element Number** to check the model geometry and the numbering of nodes and elements, and then toggle it off.

Change Element Dependent Material Property:

This function is related to the Notational Size of Members and should be applied once the elements have been generated.

In the **Tree Menu:** Geometry / Properties / **Change Element Dependent Material Property** ↵

Click **Select All**
Element Dependent Material>**Notational Size of Membe**r
Select **Auto Calculate**
Click Apply
Click Close

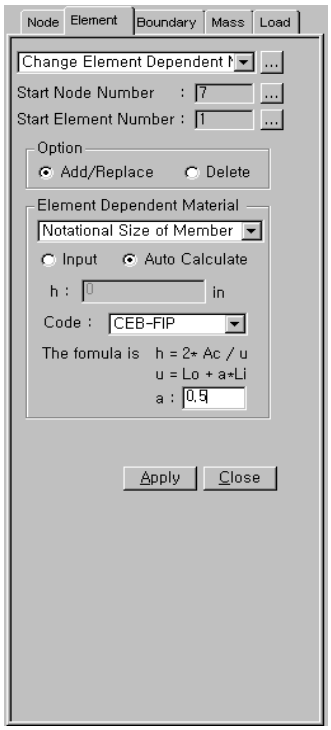


Figure 11 Change Element Dependent Material Property Window

Structure Support Conditions

In the **Tree Menu:** Geometry / Boundaries / **Supports** ↵
Select Window >Node 1
Options>**Add**
Support Type (Local Direction)>**D-ALL**
Click Apply

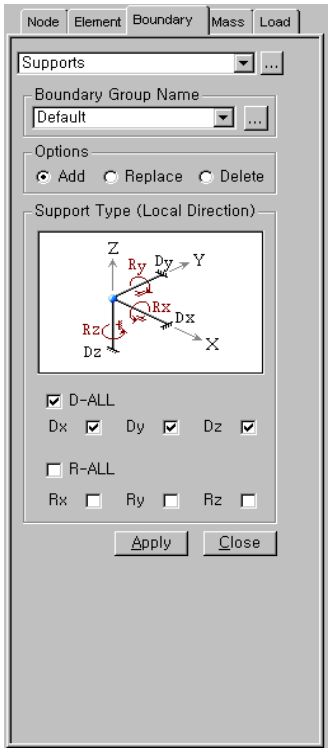


Figure 12 Supports Window

Select Window >Node 29
Support Type (Local Direction)> **Dy, Dz**
Click Apply

Select Window >Nodes 2to6
Support Type (Local Direction)> **Dx, Dz**
Click Apply

Select Window >Nodes 30, 75to78
Support Type (Local Direction)> **Dz**
Click Apply
Click Close

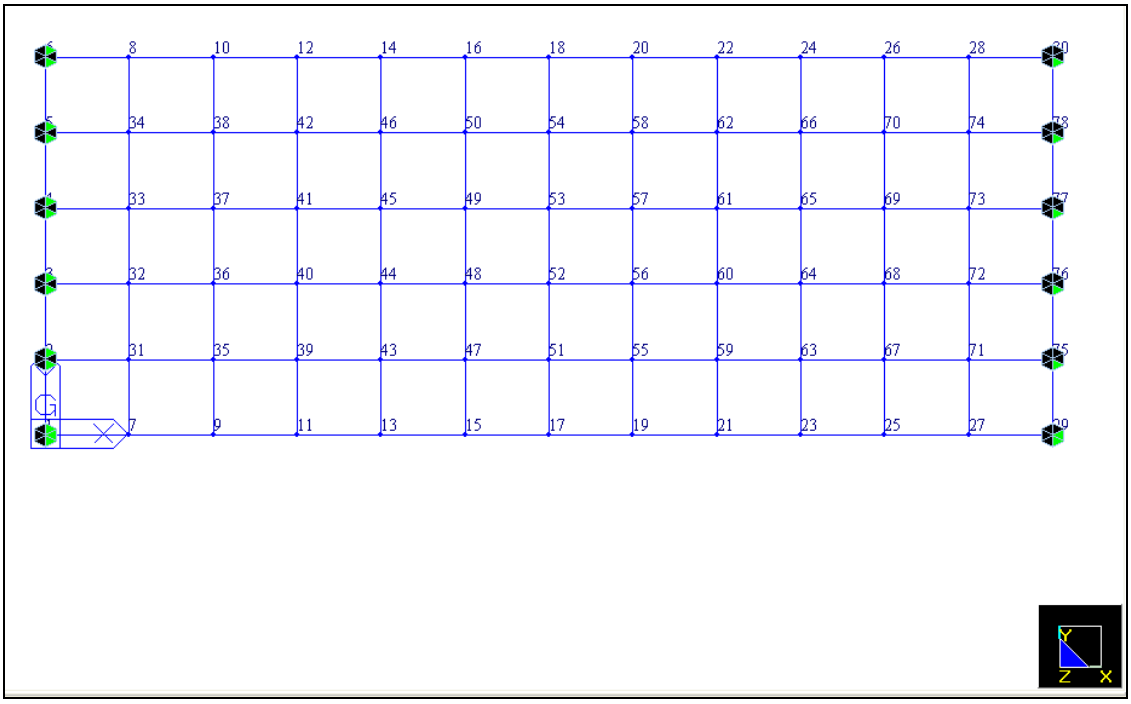


Figure 13 Model Boundary Conditions

Loading Data

The following items are defined in this section:
Load groups
Static loads
Prestress loads
Moving loads

Load Groups:

To perform Construction Stage analyses it is required to define groups of elements, boundary conditions and loads. Load groups are defined here to facilitate the assignment of loads to their respective groups.

In the **Tree Menu:** Click ***Group*** tab
Right-click **Load Group**
Select **New…**
Name>**PC & C/B**
Click Add
Name> **Deck**
Click Add
Name> **Barrier**
Click Add
Name> **Wearing surface**
Click Add
Name> **Prestress**
Click Add
Click Close

Static Loads:

In the **Tree Menu:** Click ***Menu*** tab
Static Loads> **Static Load Cases** ↵
Name>**Deck**
Type>**Dead Load of Component and Attachments (DC)**
Click Add
Name>**Wearing surface**
Type>**Dead Load of Wearing Surfaces and Utilities (DW)**
Click Add
Name>**Barrier**
Type> **Dead Load of Wearing Surfaces and Utilities (DW)**
Click Add
Name>**PC & C/B**
Type> **Dead Load of Component and Attachments (DC)**
Click Add

Name>**Prestress**
Type>**Prestress (PS)**
Click Add
Name>**Creep**
Type>**Creep (CR)**
Click Add
Name>**Shrinkage**
Type>**Shrinkage (SH)**
Click Add
Click Close

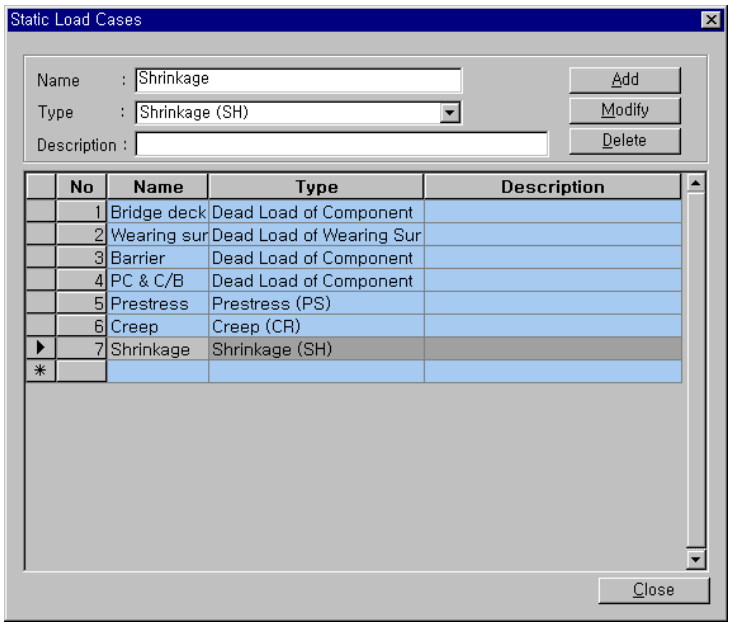


Figure 14 Static Load Cases Window

Click **Iso View**
Click **Select Identity-Elements**
Select Identity dialog box>Select Type>**Section**
Click "**1: Interior Precast Beams**"
Click Add
Click Close

In the **Tree Menu:** Static Loads>**Element Beam Loads** ↵
Load Case Name>**Deck**
Load Group Name>**Deck**
Direction>**Global Z**
Projection>**No**
Value>**Relative**
w>**-0.922** (0.900 deck + 0.022 haunch)
Click Apply

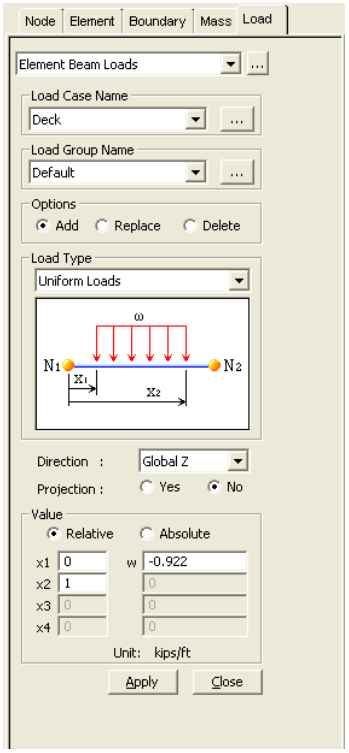


Figure 15 Element Beam Loads Window

The loading is displayed in the **Model View** window.

Click **Select Identity-Elements**
Select Identity dialog box>Select Type>**Section**
Click "**2: Exterior Precast Beams**"
Click Replace
Click Close

Load Case Name>**Deck**
Load Group Name>**Deck**
w>**-0.772** (0.750 deck + 0.022 haunch)
Click Apply

Click **Select Identity-Elements**
Select Identity dialog box>Select Type>**Section**
Click "**1: Interior Precast Beams**"
Click Add
Click "**2: Exterior Precast Beams**"
Click Replace
Click Add
Click Close

Load Case Name>**Wearing surface**
Load Group Name>**Wearing surface**
w>**-0.2**
Click Apply

Click **Select Previous**
Load Case Name> **Barrier**
Load Group Name> **Barrier**
w>**-0.1**
Click Apply
Click Close

In the **Tree Menu:** Static Loads> **Self Weight** ↵
Load Case Name> **PC & C/B**
Load Group Name> **PC & C/B**
Self Weight Factor>Z>**-1**
Click Add
Click Close

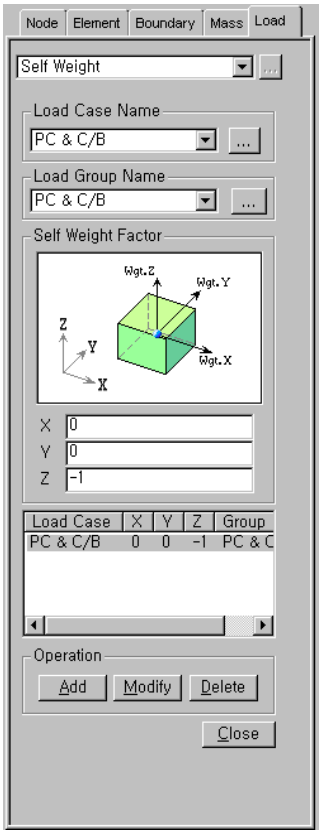


Figure 16 Self Weight Window

Prestress Data and Loads:

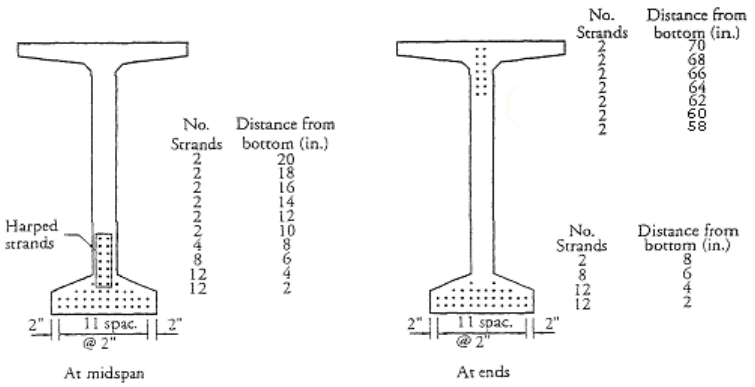


Figure 17 Strand Pattern

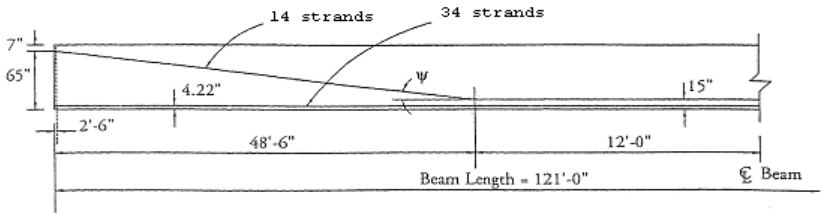


Figure 18 Longitudinal Strand Profile

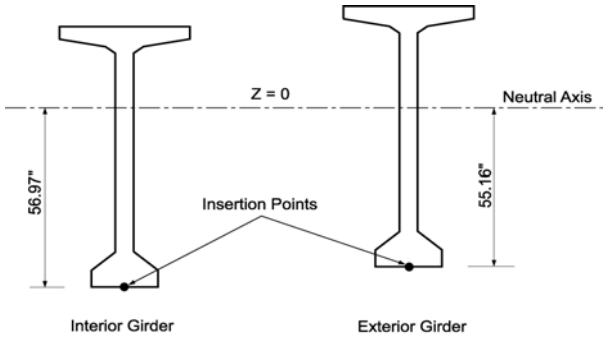


Figure 19 Profile Insertion Points

In the **Tree Menu:** Click ***Group*** tab
Right-click **Tendon Group**
Select **New…**
Name>**Tendon**
Suffix>**1to12**
Click Add
Click Close

Tools / ***Unit System***
Length>**in**; Force (Mass)>**kips** ↵

In the **Tree Menu:** Click ***Menu*** tab
Static Loads>Prestress Loads>**Tendon Property** ↵
Tendon Property dialog box>Click Add
Tendon Name>**TH**
Tendon Type>**Internal (Pre-Tension)**
Material>**3: Tendon**
Click … to the right of **Total Tendon Area**
Tendon Area dialog box> Strand Diameter**>12.7mm (0.5")**
Number of Strands>**14**
Click OK
Select **Relaxation Coefficient**
Relaxation Coefficient>Magura>**45**
Ultimate Strength>**270**
Yield Strength>**243**
Click OK

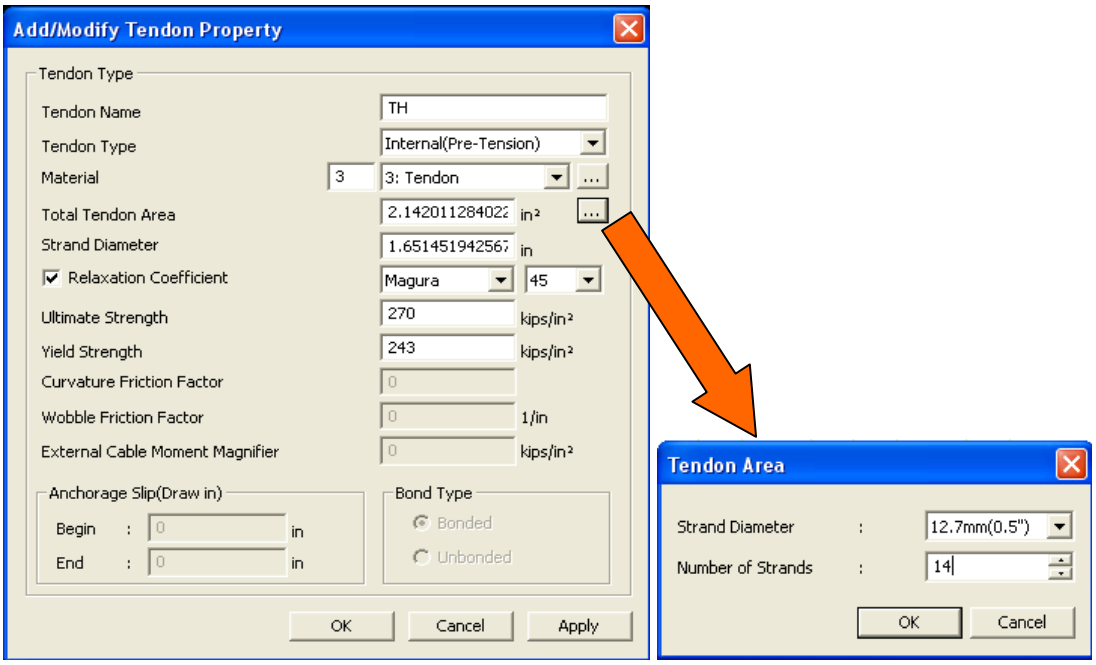


Figure 20 Add/Modify Tendon Property Window

Tendon Property dialog box>Click Add
Tendon Name>**TS**
Tendon Type>**Internal (Pre-Tension)**
Material>**3: Tendon**
Click ... to the right of **Total Tendon Area**
Tendon Area dialog box> Strand Diameter**>12.7mm (0.5")**
Number of Strands>**34**
Click OK
Select **Relaxation Coefficient**
Relaxation Coefficient>Magura>**45**
Ultimate Strength>**270**
Yield Strength>**243**
Click Close

Toggle off **Node Number**
Toggle on **Element Number**
Click **Top View**
Click **Select Identity-Elements**
Select Identity dialog box>Select Type>**Section**
Click "**1: Interior Precast Beams**"
Click "**2: Exterior Precast Beams**"
Click Add
Click Close
Click **Activate**

In the **Tree Menu:** Static Loads>Prestress Loads> **Tendon Profile** ↵
Tendon Profile dialog box>Click Add
Tendon Name>**TH1**
Group>**Tendon1**
Tendon Property>**TH**
Click in **Assigned Elements**
Select Window > Elements 1to23by2
Input Type>**3-D**
Curve Type>**Spline**
Profile>Reference Axis>**Straight**
Enter the following data in the **Profile** window:

	x(in)	y(in)	z(in)	fix	Ry[deg]	Rz[deg]
1	0.0000	0.0000	64.0000	☐	0.00	0.00
2	582.000	0.0000	14.0000	☑	0.00	0.00
3	870.000	0.0000	14.0000	☑	0.00	0.00
4	1452.00	0.0000	64.0000	☐	0.00	0.00
5				☐		

Figure 21 TH Tendon Profile Data

Profile Insertion Point>**-6, 0, -52.79**
x-Axis Direction>**X**
Click OK

Note:
An insertion point is used as a point of reference for the tendon profile in the Global Coordinate System (GCS). Only one profile is needed for a precast beam in spite of the number of elements (four in this example) that we are using to model it.

As it is shown in Figure 19, the insertion points of both exterior and interior girders are located at the bottom of the lower flanges. However, the vertical (Z-axis) coordinate of these points are different. This is because the distances from the neutral axis to the bottom fiber are not the same due to the differences in their respective effective widths of concrete slab.

Tendon Profile dialog box>Select **TH1**
Distance>**0, 108, 0**
Click Copy / Move
Select **TH1-Copy**
Click Modify
Tendon Name>Change to **TH2**
Group> Change to **Tendon2**
Click in **Assigned Elements**
Click **Unselect All**
Select Window > Elements 25to69by4
Profile Insertion Point>Change to **-6, 108, -54.57**
Click OK

Tendon Profile dialog box>Select **TH2**
Click Copy / Move
Select **TH2-Copy**
Click Modify
Tendon Name> Change to **TH3**
Group> Change to **Tendon3**
Click in **Assigned Elements**
Click **Unselect All**
Select Window > Elements 26to70by4
Click OK

Use the same method described above to generate profiles for tendons TH4 & TH5. Change ***Group*** to **Tendon4** and **Tendon5** for TH4 and TH5, respectively.

Tendon Profile dialog box>Select **TH5**
Click Copy / Move
Select **TH5-Copy**
Click Modify
Tendon Name> Change to **TH6**
Group> Change to **Tendon6**
Click in **Assigned Elements**
Click **Unselect All**
Select Window > Elements 2to24by2
Profile Insertion Point>Change to **-6, 540, -52.79**
Click OK

Figure 22 Add/Modify Tendon Profile Window

Tendon Profile dialog box>Click Add
Tendon Name>**TS1**
Group>**Tendon7**
Tendon Property>**TS**
Click in **Assigned Elements**
Select Window > Elements 1to23by2
Input Type>**3-D**
Curve Type>**Spline**
Profile>Reference Axis>**Straight**
Enter the following data in the **Profile** window:

	x (in)	y (in)	z (in)	fix	Ry [deg]	Rz [deg]
1	0.0000	0.0000	4.0000	☐	0.00	0.00
2	1452.00	0.0000	4.0000	☐	0.00	0.00
3				☐		

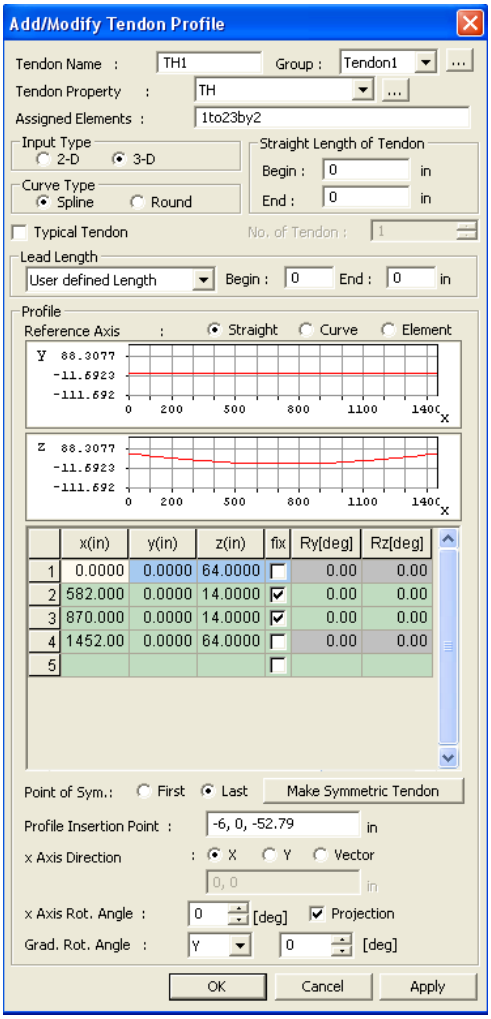


Figure 23 TS Tendon Profile Data

Profile Insertion Point>**-6, 0, -52.79**
x-Axis Direction>**X**
Click OK

Tendon Profile dialog box>Select **TS1**
Distance>**0, 108, 0**
Click Copy / Move
Select **TS1-Copy**
Click Modify
Tendon Name>Change to **TS2**
Group> Change to **Tendon8**
Click in **Assigned Elements**
Click **Unselect All**
Select Window > Elements 25to69by4
Profile Insertion Point>Change to **-6, 108, -54.57**
Click OK

Tendon Profile dialog box>Select **TS2**
Click Copy / Move
Select **TS2-Copy**
Click Modify
Tendon Name> Change to **TS3**
Group> Change to **Tendon9**
Click in **Assigned Elements**
Click **Unselect All**
Select Window > Elements 26to70by4
Click OK

Use the same method described above to generate profiles for tendons TS4 & TS5. Change ***Group*** to **Tendon10** and **Tendon11** for TS4 and TS5, respectively.

Tendon Profile dialog box>Select **TS5**
Click Copy / Move
Select **TS5-Copy**
Click Modify
Tendon Name> Change to **TS6**
Group> Change to **Tendon12**
Click in **Assigned Elements**
Click **Unselect All**
Select Window > Elements 2to24by2
Profile Insertion Point>Change to **-6, 540, -52.79**
Click OK
Click Close

Visually verify that the tendon profiles have been entered correctly.
Click **Iso View**
In the **Tree Menu:** Click ***Works*** tab
Prestressing Tendon>**Tendon Profile**
Right-click mouse and select **Display**
Click **Initial View**

In the **Tree Menu:** Click ***Menu*** tab
Static Loads>Prestress Loads>**Tendon Prestress Loads** ↵
Load Case Name>**Prestress**
Load Group Name>**Prestress**
Select Tendon for Loading>Tendon> Select all tendons **(TH1~TH6, TS1~TS6)**
Click >
Stress Value>**Stress**
1st Jacking>**Begin**
Begin>**183.9**
End>**0**

Click Add
Click Close

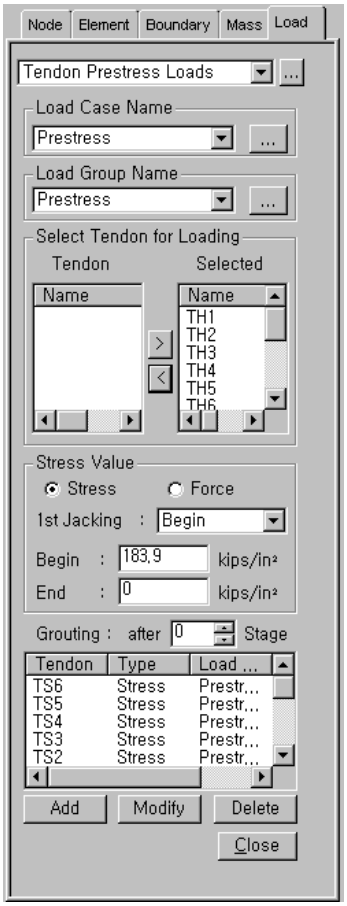


Figure 24 Tendon Prestress Loads Window

Moving Loads:

Click **Initial View**
Click **Top View**
Click **Activate All**
Click **Element Number**

In the **Tree Menu:** Click ***Group*** tab
Right-click **Structure Group**
Select **New…**
Name>**Cross Beam**
Suffix>**1to5**
Click Add
Click Close

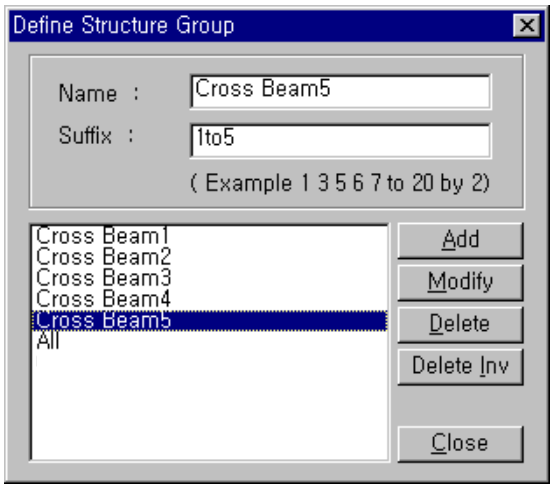


Figure 25 Define Structure Group Window

Select Intersect > Elements 73to85
"Drag & Drop" **Cross Beam 1** from the **Tree Menu** to **Model View**.

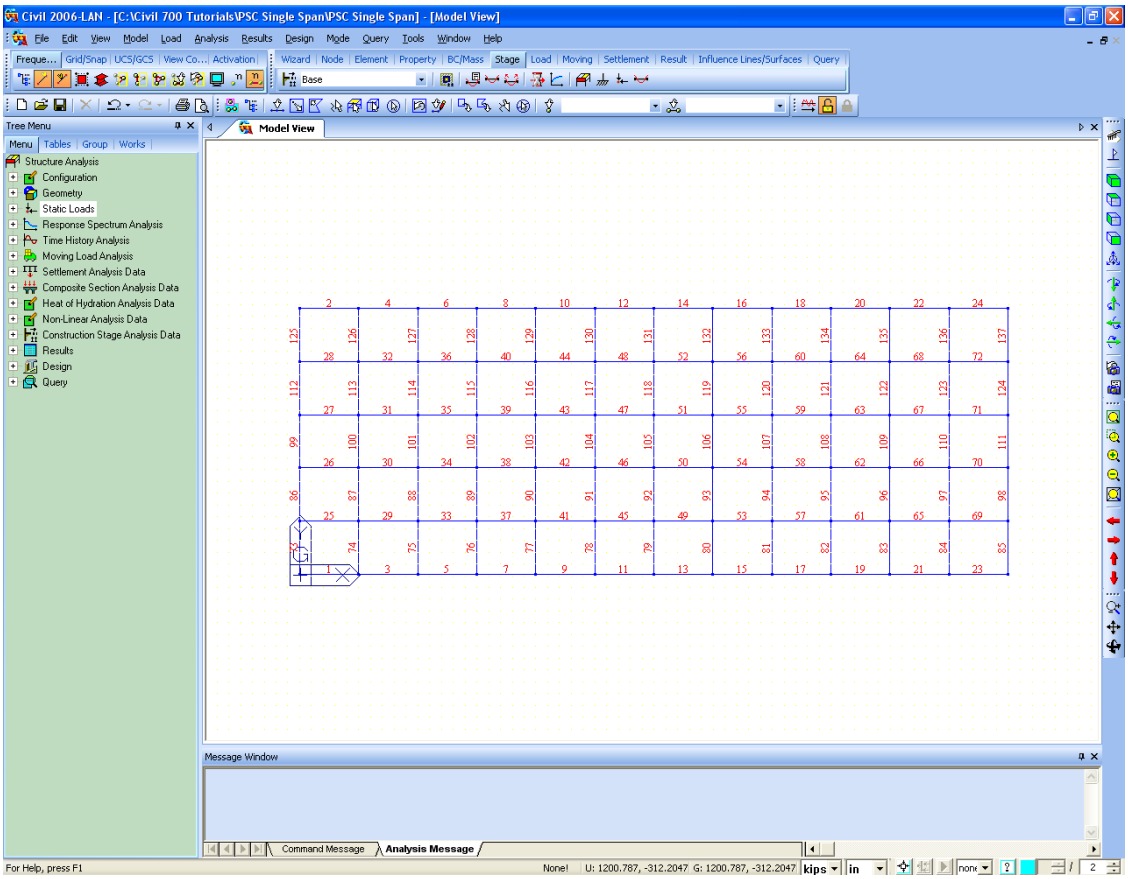


Figure 26 Assignment of Cross Beam Groups

Select Intersect > Elements 86to98

"Drag & Drop" **Cross Beam 2** from the **Tree Menu** to **Model View**.

Repeat the steps to define ***Groups*** **Cross Beam 3, Cross Beam 4 & Cross Beam 5.**

The assignment of elements to groups can be verified by double-clicking each of the ***Groups*** in the **Tree Menu** and displaying their elements in the **Model View**.

Note:
To increase the accuracy of vehicular live load analysis, the number of Cross Beams may be increased. This can be done by providing large number of equally spaced fictitious "Cross Beams" in the transverse direction, having weight density = 0. The depth and width of these "Cross Beams" will be equal to the deck slab thickness and center-to-center distance between the "Cross Beams", respectively.

Tools / ***Unit System***
Length>**ft**; Force (Mass)>**kips** ↵
Toggle on **Node Number**
Toggle off **Element Number**

In the **Tree Menu:** Click ***Menu*** tab
Moving Loads Analysis> **Moving Load Code** ↵
Select Moving Load Code dialog box>Moving Load Code>**AASHTO LRFD**

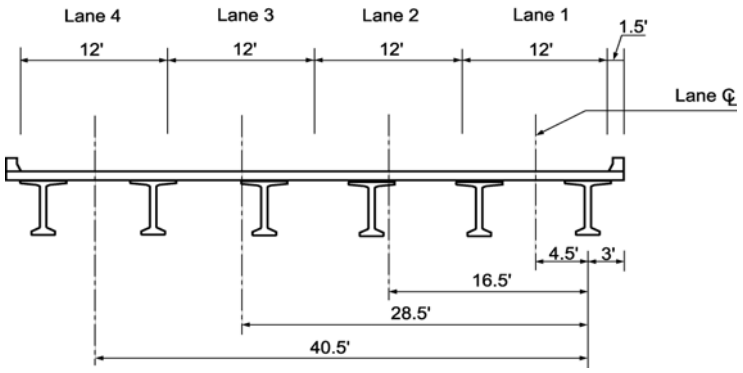


Figure 27 Traffic Lanes and their Eccentricities

Moving Loads Analysis> **Traffic Line Lanes** ↵
Traffic Line Lanes dialog box>Click Add
Lane Name>**Lane 1**
Eccentricity>**-4.5**
Vehicular Load Distribution>**Cross Beam**
Cross Beam Group>**Cross Beam 1**
Moving Direction>**Both**
Selection by>**2 points**
Click in the first box below
Select Window > Nodes 1 and 29

Click Add
Click OK

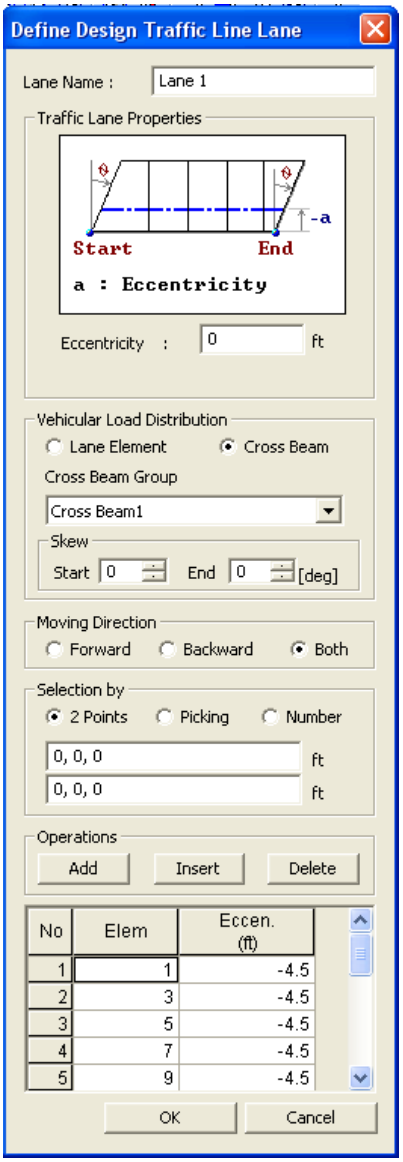


Figure 28 Definition of Design Traffic Line Lanes

Traffic Line Lanes dialog box>Click Add
Lane Name>**Lane 2**
Eccentricity>**-16.5**
Vehicular Load Distribution>**Cross Beam**
Cross Beam Group>**Cross Beam 2**
Moving Direction>**Both**
Selection by>**2 points**
Click in the first box below

Select Window > Nodes 1 and 29
Click Add
Click OK

Traffic Line Lanes dialog box>Click Add
Lane Name>**Lane 3**
Eccentricity>**-28.5**
Vehicular Load Distribution>**Cross Beam**
Cross Beam Group>**Cross Beam 4**
Moving Direction>**Both**
Selection by>**2 points**
Click in the first box below
Select Window > Nodes 1 and 29
Click Add
Click OK

Traffic Line Lanes dialog box>Click Add
Lane Name>**Lane 4**
Eccentricity>**-40.5**
Vehicular Load Distribution>**Cross Beam**
Cross Beam Group>**Cross Beam 5**
Moving Direction>**Both**
Selection by>**2 points**
Click in the first box below
Select Window > Nodes 1 and 29
Click Add
Click OK
Click Close

In the **Tree Menu**: Moving Load Analysis > **Vehicles** ↵
Vehicles dialog box>Click Add Standard
Standard Name>**AASHTO LRFD Load**
Vehicular Load Name>**HL-93TDM**
Vehicular Load Type>**HL-93TDM**
Dynamic Allowance: **33** (%)
Click OK

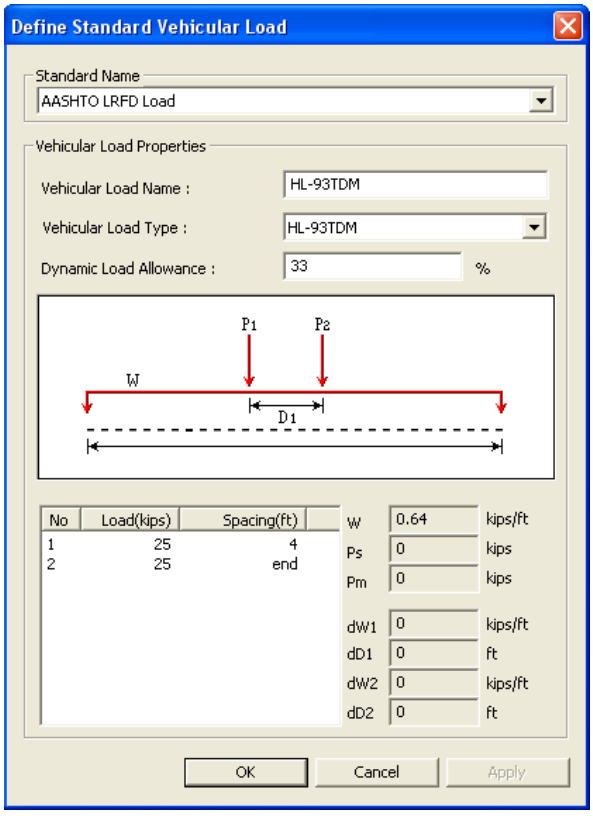


Figure 29 Definition of Standard Vehicular Loads

Vehicles dialog box>Click Add Standard
Standard Name>**AASHTO LRFD Load**
Vehicular Load Name>**HL-93TRK**
Vehicular Load Type>**HL-93TRK**
Dynamic Allowance: **33** (%)
Click OK
Click Close

In the **Tree Menu**: Moving Load Analysis > **Moving Load Cases**
Moving Load Cases dialog box>Click Add
Load Case Name>**MLC**
Enter the following data in the **Multiple Presence Factor** window:

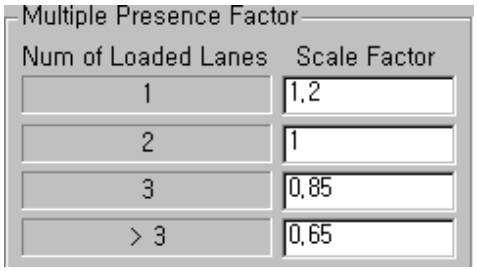


Figure 30 Multiple Presence Factors

Sub-Load Cases>Loading Effect>**Independent**
Click Add
Sub-Load Case dialog box>Vehicle Class>**VL:HL-93TDM**
Scale Factor>**1**
Min. Number of Loaded Lanes>**1**
Max. Number of Loaded Lanes>**4**
Assignment Lanes>List of Lanes>Select all lanes **(Lane 1, Lane 2, Lane 3, Lane 4)**
Click ->
Click OK

Define Moving Load Case dialog box>Click Add
Sub-Load Case dialog box>Vehicle Class>**VL:HL-93TRK**
Scale Factor>**1**
Min. Number of Loaded Lanes>**1**
Max. Number of Loaded Lanes>**4**
Assignment Lanes>List of Lanes>Select all lanes **(Lane 1, Lane 2, Lane 3, Lane 4)**
Click ->
Click OK
Click Close

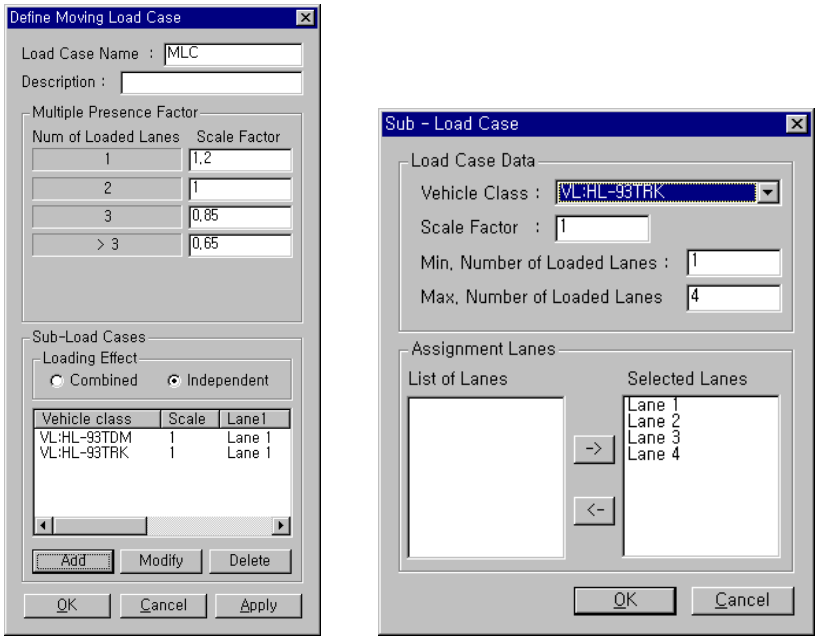


Figure 31 Definition of Moving Load Cases

Construction Stage Analysis Data

Three stages are defined to model the bridge during construction. Details of the construction stages are shown below:

Stage	***Day***	***Description***
Stage 1 (30 days)	1	Placing of precast beams and cross beams. Prestressing of strands.
	21	Pouring deck slab.
Stage 2 (30 days)	1	Composite beam & slab behavior takes place.
	1	Installation of barrier.
	6	Placing of wearing surface.
Stage 3 (10000 days)	-	-

Note: *Age of all precast members (precast beams & cross beams) is 7 days at the time of placing (1st day of Stage 1).*

Table 1.2 Construction Stages

Groups:

In the **Tree Menu:** Click ***Group*** tab
Right-click **Structure Group**
Select **New…**
Name>**All**
Click Add
Click Close
Click **Select All**
"Drag & Drop" **All** from the **Tree Menu** to **Model View**

Right-click **Boundary Group**
Select **New…**
Name> **Supports**
Click Add
Click Close
Click **Select All**
"Drag & Drop" **Supports** from the **Tree Menu** to **Model View**
Select Boundary Type dialog box>**Support**
Click OK

Define Construction Stages:

In the **Tree Menu:** Click ***Menu*** tab
Construction Stage Analysis Data > **Define Construction Stage** ↵

Construction Stage dialog box>Click Generate
Stage>Name>**Stage**
Stage>Suffix>**1to3**
Save Result>**Stage, Additional Steps** (on)
Click OK

Construction Stage dialog box>Click Add
Stage>**Stage 1**
Name>**Stage 1**
Duration>**30**
Additional Steps>Day>**21**
Click Add in the ***Additional Steps*** window

Click ***Element*** tab
Group List>**All**
Activation>Age>**7**
Click Add in the ***Activation*** window

Click ***Boundary*** tab
Group List>**Supports**
Support/Spring Position>**Deformed**
Click Add in the ***Activation*** window

Click ***Load*** tab
Group List>**PC & C/B**
Click Add in the ***Activation*** window
Group List>**Prestress**
Click Add in the ***Activation*** window
Group List>**Deck**
Active Day>**21**
Click Add in the ***Activation*** window
Click OK

Construction Stage dialog box>Click Add
Stage>**Stage 2**
Name>**Stage 2**
Duration>**30**
Additional Steps>Day>**6**
Click Add in the ***Additional Steps*** window

Click ***Load*** tab
Once activated, the Element, Boundary and Load groups remain active unless they are specifically deactivated.
Group List>**Barrier**
Click Add in the ***Activation*** window
Group List> **Wearing Surface**
Active Day>**6**
Click Add in the ***Activation*** window
Click OK

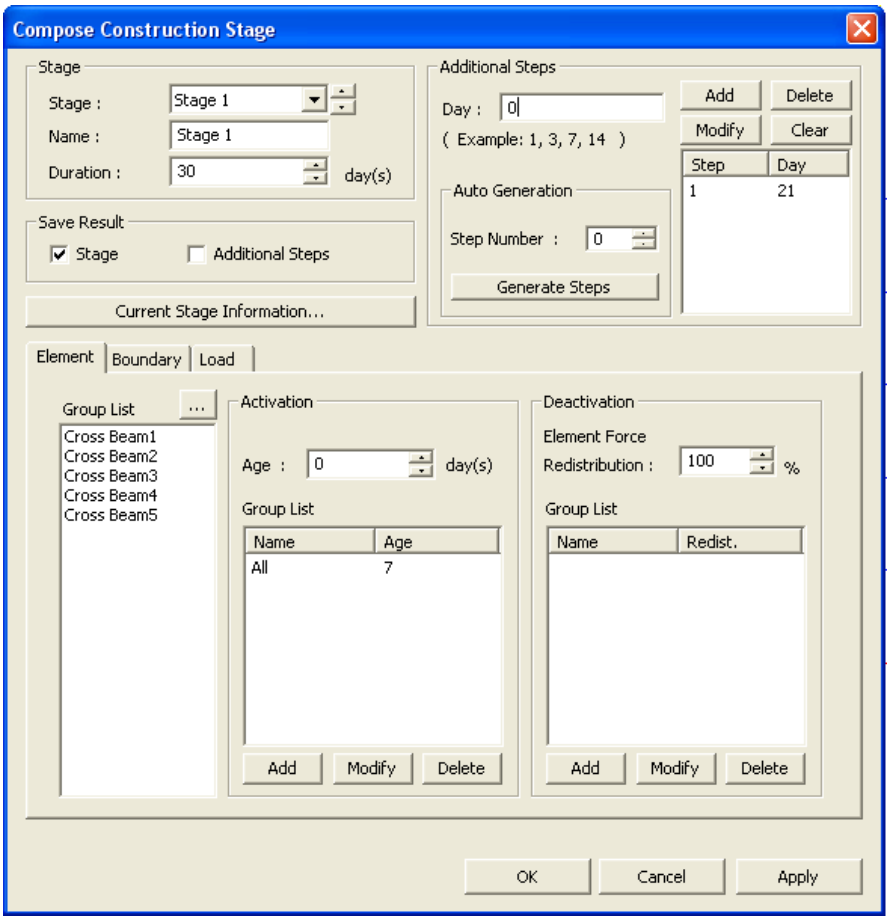


Figure 32 Definition of Construction Stage 1

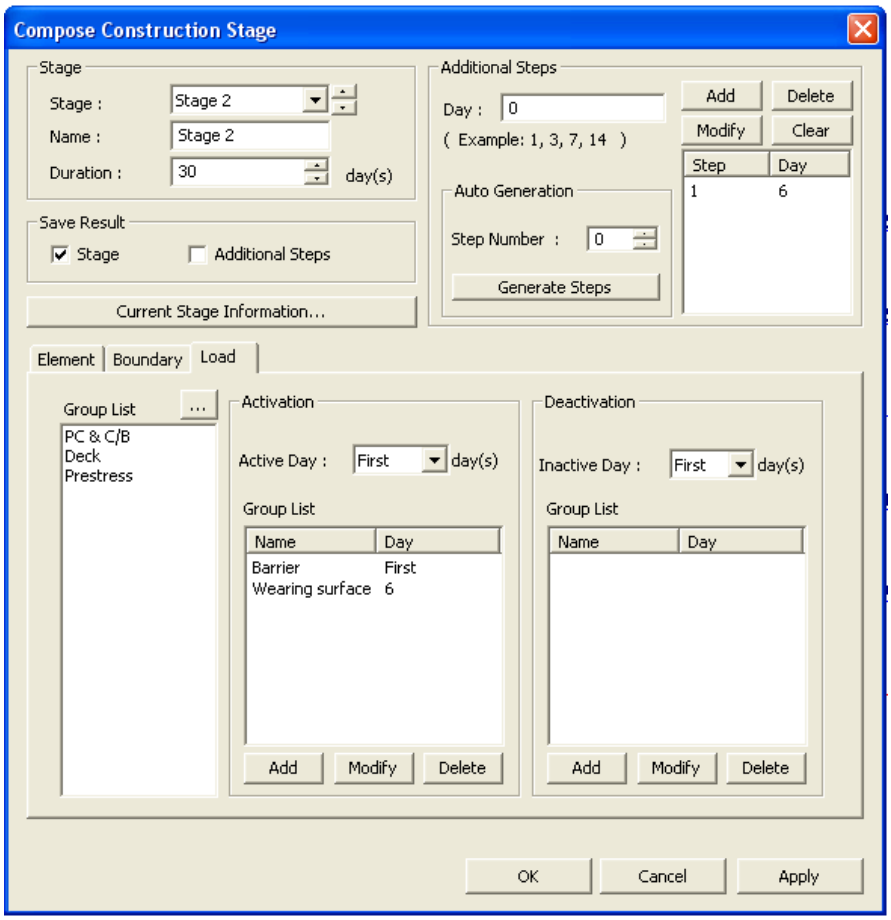


Figure 33 Definition of Construction Stage 2

Construction Stage dialog box>Click Add
Stage>**Stage 3**
Name>**Stage 3**
Duration>**10000**
Click OK
Click Close

In the **Tree Menu**: Construction Stage Analysis Data > **Composite Section for Construction Stage** ↵

Composite Section for Construction Stage dialog box>Click Add
Active Stage>**Stage 1**
Section>**1: Interior Precast Beams**
Composite Type>**Normal**

Construction Sequence>Part>**1**:
Material Type>**Material**
Material>**2:Precast**
Composite Stage>**Active Stage**
Age>**7**

Construction Sequence>Part>**2**:
Material Type>**Material**
Material>**1:Deck**
Composite Stage>**Stage 2**
Age>**10**
Click OK

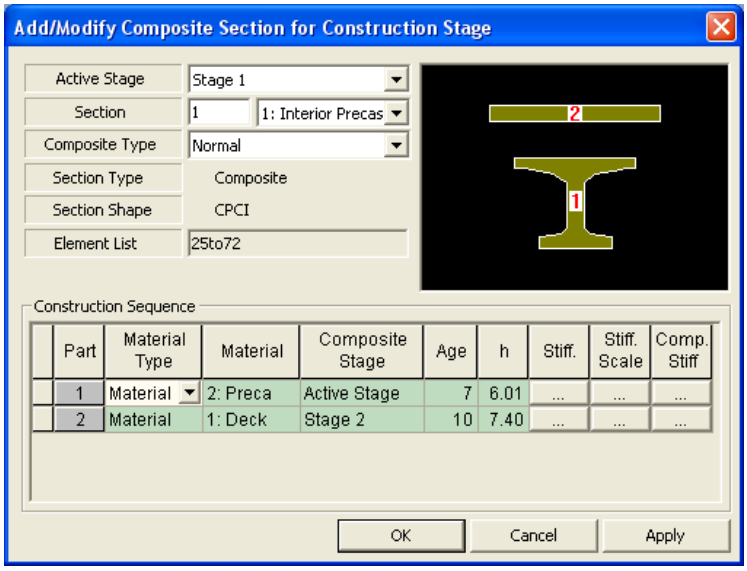


Figure 34 Composite Section 1 (Interior Precast Beams) during Construction Stages

Composite Section for Construction Stage dialog box>Click Add
Active Stage>**Stage 1**
Section>**2: Exterior Precast Beams**
Composite Type>**Normal**

Construction Sequence>Part>**1**:
Material Type>**Material**
Material>**2:Precast**
Composite Stage>**Active Stage**
Age>**7**

Construction Sequence>Part>**2**:
Material Type>**Material**
Material>**1:Deck**
Composite Stage>**Stage 2**
Age>**10**
Click OK
Click Close

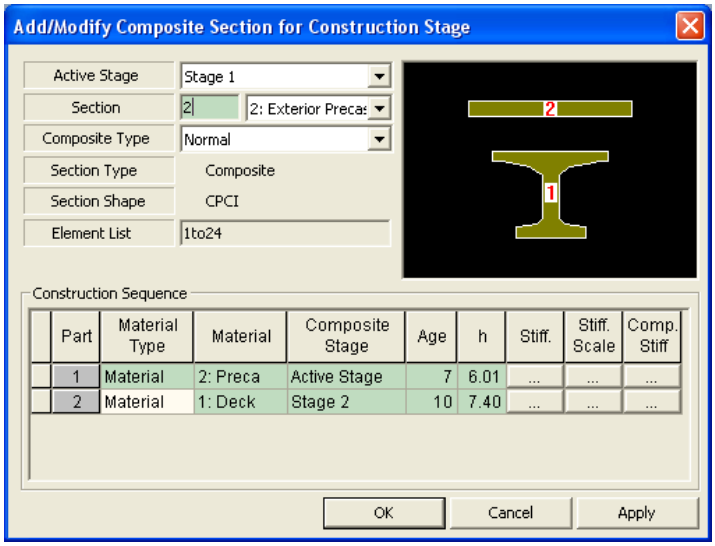


Figure 35 Composite Section 2 (Exterior Precast Beams) during Construction Stages

In the **Main Menu: Analysis** / ***Construction Stage Analysis Control***
Final Stage>**Last Stage**
Analysis Option>**Include Time Dependent Effect** (on)
Time Dependent Effect>**Creep & Shrinkage** (on)
Type>**Creep & Shrinkage**
Auto Time Step Generation for Large Time Gap (on)
Tendon Tension Loss Effect (Creep & Shrinkage) (on)
Variation of Comp. Strength (on)
Tendon Tension Loss (Elastic Shortening) (on)
Frame Output>**Calculate Output of Each part of Composite Section** (on)

Load Cases to be Distinguished from Dead Load for CS Output:
Load Case>**Wearing Surface**
Click Add
Load Case>**Barrier**
Click Add
Click OK

Beam Section Property Changes>**Change with Tendon**
Save Output of Current Stage (Beam/Truss) (on)

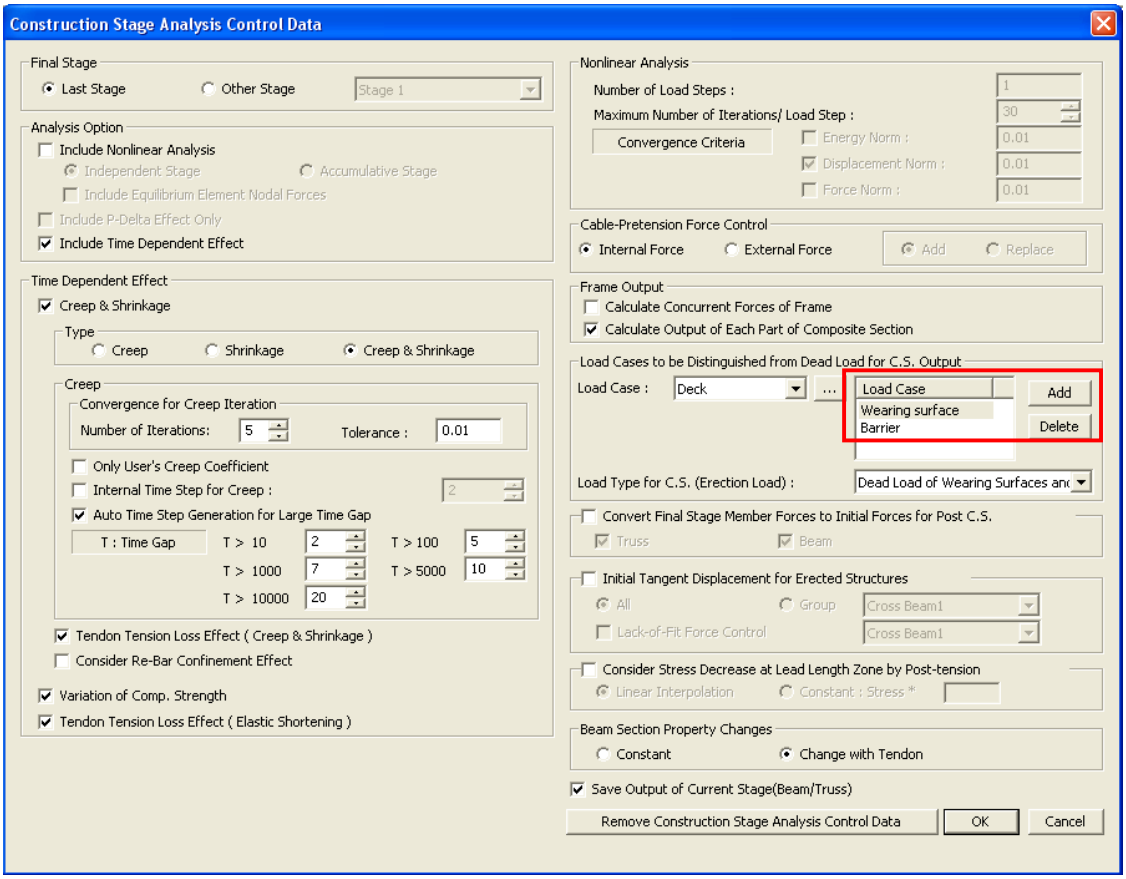


Figure 36 Construction Stage Analysis Control Data

Perform Structural Analysis

In the **Main Menu: Analysis /** ***Perform Analysis***

Verification and Interpretation of Results

Load Combinations:

Select the Post Construction Stage (PostCS).
In the **Tree Menu:** Click **Menu** tab.
Results>**Combinations** ↵

Load Combinations dialog box>***General*** tab>Click Auto Generation...
Option>**Add**
Add Envelope (on)
Code Selection>**Concrete**
Design Code>**AASHTO-LRFD02**
Manipulation of Construction Stage Load Case>**CS Only**

Note:
Do not select "ST+CS" option (Static Load + Construction Stage) since the output will be misleading. "Bridge Deck" and "PC & C/B" have been defined as both "Dead Load of Components and Attachments (DC)" as well as "Construction Stage (CS) Loads". Similarly, "Wearing Surface" and "Barrier" have been defined as both "Dead Load of Wearing Surface and Utilities (DW)" as well as "Erection Loads (EL)". (Refer to "Loading Data" and Figure 36 for details). Thus, these dead loads will appear twice in the output if the "ST+CS" option is selected.

Load Modifier>**1**
Load Factors for Permanent Loads (Yp):
Component and Attachments>Load Factor>**Both**
Wearing Surfaces and Utilities>Load Factor>**Both**

Condition for Temperature, Creep, Shrinkage Factor>**Deformation Check**
Click OK

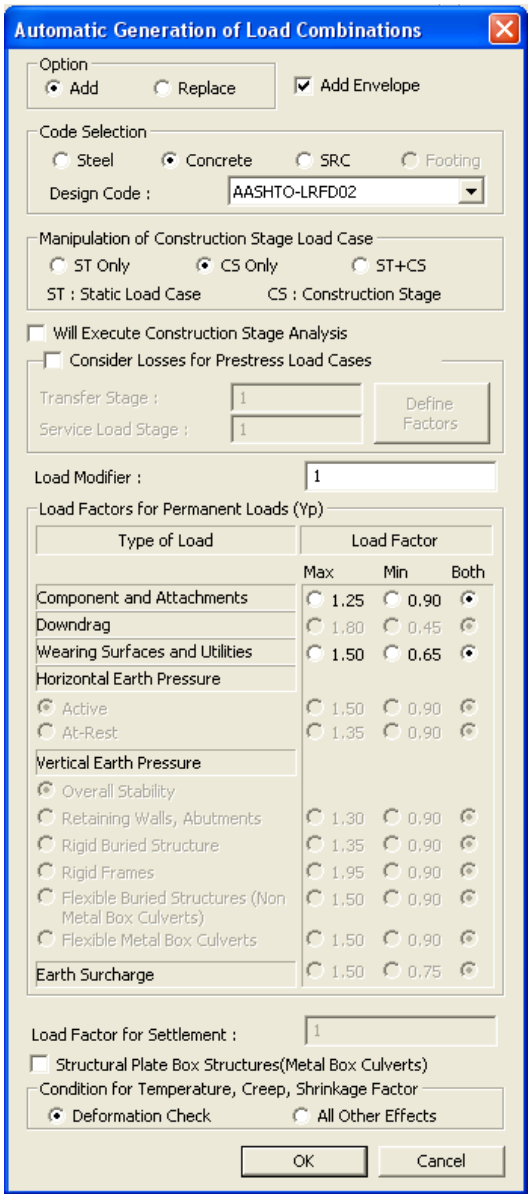


Figure 37 Generation of Load Combinations

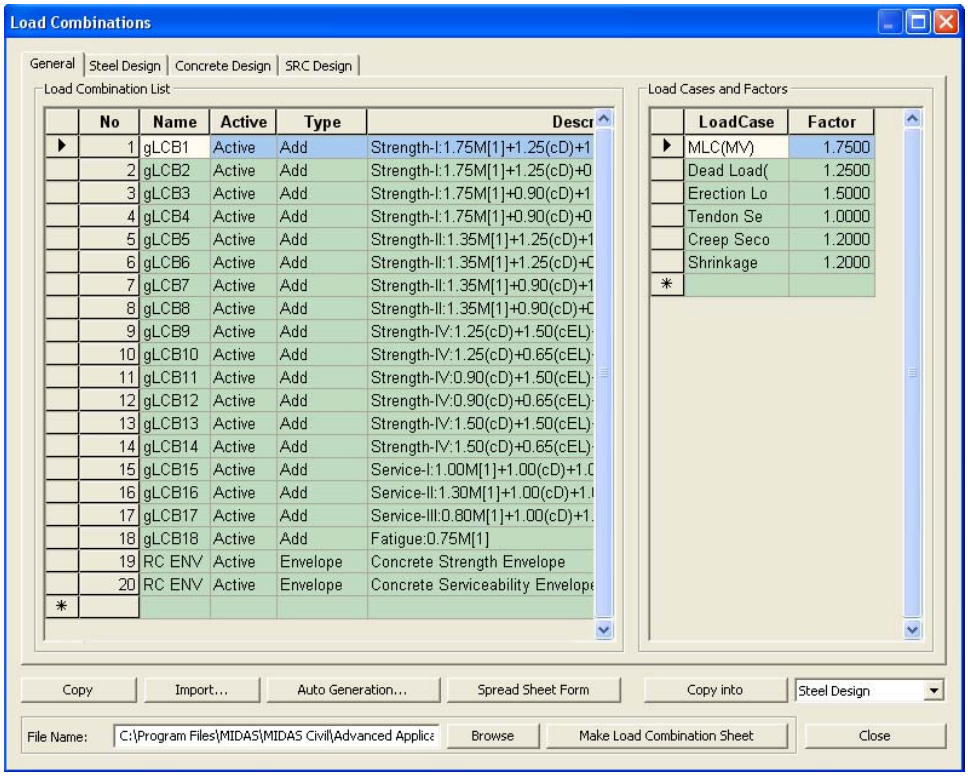


Figure 38 Auto Generated Load Combinations

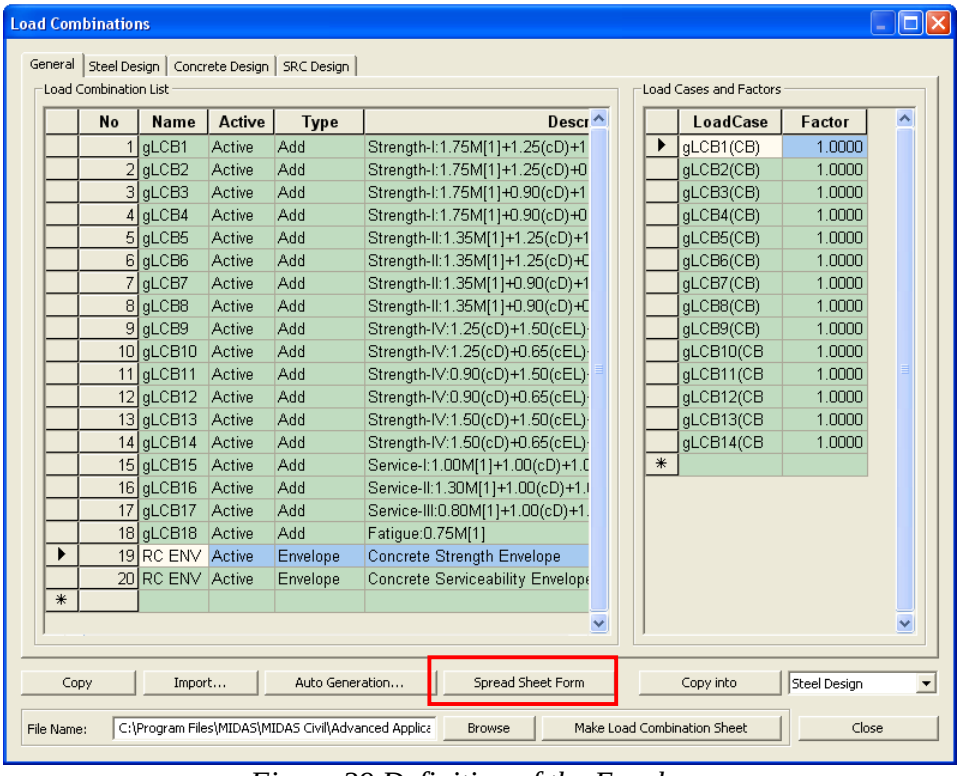


Figure 39 Definition of the Envelope

Click Spread Sheet Form

Change the load factors for the load combinations gLCB15, gLCB16 and gLCB17 (Service I, II and III, respectively) as shown in Table 1.3.

This step is required to generate appropriate deformations as per AASHTO-LRFD02.

The factors for loads not listed in Table 1.3 are zero and their cells can be left blank.

Click Close

Load Comb.	MLC	Dead Load	Erection Load	Tendon Primary	Tendon Secondary	Creep Primary	Creep Secondary	Shrinkage Primary	Shrinkage Secondary
gLCB1	1.75	1.25	1.50		1.00		1.20		1.20
gLCB2	1.75	1.25	0.65		1.00		1.20		1.20
gLCB3	1.75	0.90	1.50		1.00		1.20		1.20
gLCB4	1.75	0.90	0.65		1.00		1.20		1.20
gLCB5	1.35	1.25	1.50		1.00		1.20		1.20
gLCB6	1.35	1.25	0.65		1.00		1.20		1.20
gLCB7	1.35	0.90	1.50		1.00		1.20		1.20
gLCB8	1.35	0.90	0.65		1.00		1.20		1.20
gLCB9		1.25	1.50		1.00		1.20		1.20
gLCB10		1.25	0.65		1.00		1.20		1.20
gLCB11		0.90	1.50		1.00		1.20		1.20
gLCB12		0.90	0.65		1.00		1.20		1.20
gLCB13		1.50	1.50		1.00		1.20		1.20
gLCB14		1.50	0.65		1.00		1.20		1.20
gLCB15	1.00	1.00	1.00	1.00		1.20		1.20	
gLCB16	1.30	1.00	1.00	1.00		1.20		1.20	
gLCB17	0.80	1.00	1.00	1.00		1.20		1.20	
gLCB18	0.75								

Table 1.3 Load Combinations for Construction Stages

Tendon Time-dependent Loss Graph:

In this tutorial, the prefix TH stands for harped tendon and TS stands for straight tendon. **Animate** button can be used to view the Loss Graphs for all the stages for the selected tendon, sequentially.

In the **Tree Menu:** Results> **Tendon Time-dependent Loss Graph** ↵
Graph for tendon "**TH1**" in Stage 1 is automatically displayed.
Tendon>**TH2**
Stage>**Stage 1**
Step>**Last Step**
Click Animate

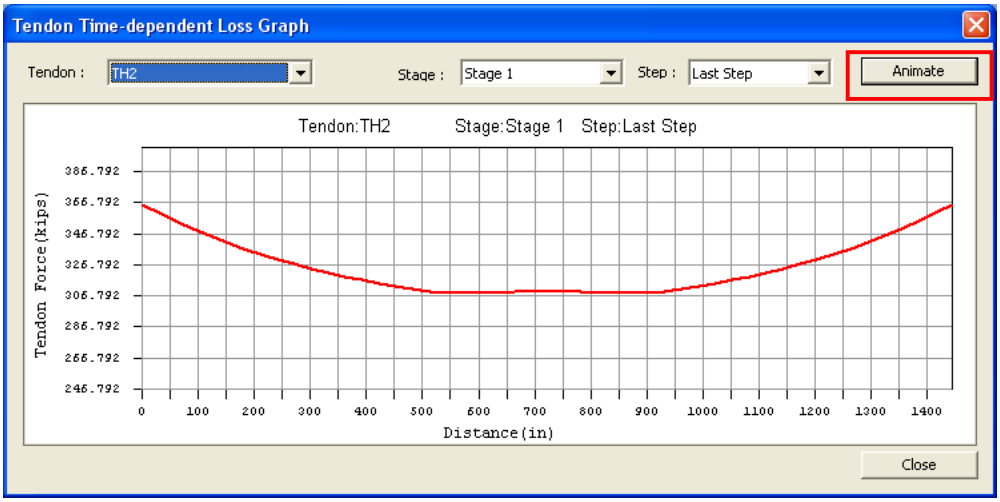


Figure 40 Tendon TH2 Loss Graph

Tendon> **TS2**

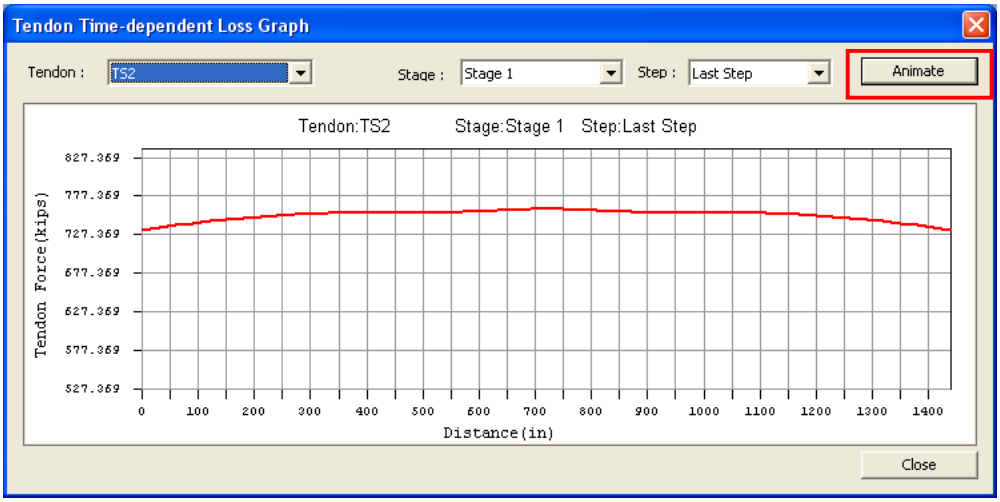


Figure 41 TS2 Tendon Loss Graph

Click Close

Pretension Losses in Tendons:

In the **Tree Menu:** Click **Tables** tab.

Result Tables > Tendon > **Tendon Loss** ↵

Click ***Tendon Group*** > **Tendon 1, Tendon 2, Tendon 3, Tendon 4, Tendon 5, Tendon 6, Tendon 7, Tendon 8, Tendon 9, Tendon 10, Tendon 11,** or **Tendon 12.**

Click ***Stage*** > **Stage 1, Stage 2,** or **Stage 3.**

Click Apply

Since losses are calculated using CEB-FIP code, they are different from those given in the PCI Bridge Design Manual, where losses are calculated using AASHTO code.

Model View | **Result-[Tendon Loss (Tendon Group)]**

Elem	Part	Stress (Immediate Loss) (kips/in²)	Elastic Deform. Loss (kips/in²)	Stress(Elastic Loss)/ Stress (Immediate Loss)	Creep/Shrinkage Loss (kips/in²)	Relaxation Loss (kips/in²)	Stress(All Loss)/ Stress(Immediate Loss)	Effective Num.
The Loss of tendon group [Tendon1] at the stage of [Stage 1]								
Tendon Group		Tendon1	Stage	Stage 1	Apply			
1	I	178.1563	-0.2990	0.9983	-4.9623	-2.0167	0.9591	1.0000
1	J	172.1362	-1.6996	0.9901	-6.8818	-1.6164	0.9408	1.0000
3	I	172.1362	-1.7474	0.9898	-6.8666	-1.6153	0.9406	1.0000
3	J	165.0729	-0.6094	0.9963	-7.9731	-1.2644	0.9403	1.0000
5	I	165.0729	-0.6439	0.9961	-7.9676	-1.2635	0.9402	1.0000
5	J	157.5059	2.2255	1.0141	-8.5261	-0.9593	0.9539	1.0000
7	I	157.5059	2.1997	1.0140	-8.5209	-0.9586	0.9538	1.0000
7	J	150.4389	5.6927	1.0378	-8.7772	-0.7136	0.9748	1.0000
9	I	150.4389	5.6649	1.0377	-8.7708	-0.7129	0.9746	1.0000
9	J	147.1889	7.6187	1.0518	-8.7210	-0.6149	0.9883	1.0000
11	I	147.1889	7.6078	1.0517	-8.7183	-0.6146	0.9883	1.0000
11	J	147.1889	7.7907	1.0529	-8.5981	-0.6200	0.9903	1.0000
13	I	147.1889	7.7973	1.0530	-8.5984	-0.6201	0.9903	1.0000
13	J	147.1889	7.6153	1.0517	-8.7189	-0.6147	0.9883	1.0000
15	I	147.1889	7.6400	1.0519	-8.7222	-0.6152	0.9885	1.0000
15	J	150.4389	5.6908	1.0378	-8.7721	-0.7133	0.9748	1.0000
17	I	150.4389	5.7374	1.0381	-8.7796	-0.7143	0.9750	1.0000
17	J	157.5059	2.2604	1.0144	-8.5230	-0.9595	0.9541	1.0000
19	I	157.5059	2.3183	1.0147	-8.5308	-0.9606	0.9545	1.0000
19	J	165.0729	-0.5236	0.9968	-7.9724	-1.2654	0.9409	1.0000
21	I	165.0729	-0.4337	0.9974	-7.9855	-1.2670	0.9413	1.0000
21	J	172.1362	-1.5302	0.9911	-6.8815	-1.6187	0.9417	1.0000
23	I	172.1362	-1.3942	0.9919	-6.9187	-1.6211	0.9423	1.0000
23	J	178.1563	0.0725	1.0004	-5.0080	-2.0229	0.9609	1.0000

Figure 42 Pretension Losses (Stress) in Tendons

Model View | **Result-[Tendon Loss (Tendon Group)]**

Elem	Part	Force (Immediate Loss) (kips)	Elastic Deform. Loss (kips)	Force(Elastic Loss)/ Force(Immediate Loss)	Creep/Shrinkage Loss (kips)	Relaxation Loss (kips)	Force(All Loss)/ Force(Immediate Loss)	Effective Num.
The Loss of tendon group [Tendon1] at the stage of [Stage 1]								
Tendon Group		Tendon1	Stage	Stage 1	Apply			
1	I	381.6129	-0.6404	0.9983	-10.6293	-4.3199	0.9591	1.0000
1	J	368.7177	-3.6406	0.9901	-14.7410	-3.4624	0.9408	1.0000
3	I	368.7177	-3.7429	0.9898	-14.7084	-3.4599	0.9406	1.0000
3	J	353.5879	-1.3054	0.9963	-17.0786	-2.7084	0.9403	1.0000
5	I	353.5879	-1.3792	0.9961	-17.0666	-2.7065	0.9402	1.0000
5	J	337.3793	4.7671	1.0141	-18.2630	-2.0548	0.9539	1.0000
7	I	337.3793	4.7118	1.0140	-18.2519	-2.0534	0.9538	1.0000
7	J	322.2418	12.1939	1.0378	-18.8009	-1.5286	0.9748	1.0000
9	I	322.2418	12.1343	1.0377	-18.7872	-1.5271	0.9746	1.0000
9	J	315.2804	16.3194	1.0518	-18.6804	-1.3172	0.9883	1.0000
11	I	315.2804	16.2961	1.0517	-18.6747	-1.3165	0.9883	1.0000
11	J	315.2804	16.6877	1.0529	-18.4172	-1.3281	0.9903	1.0000
13	I	315.2804	16.7019	1.0530	-18.4178	-1.3283	0.9903	1.0000
13	J	315.2804	16.3120	1.0517	-18.6760	-1.3167	0.9883	1.0000
15	I	315.2804	16.3649	1.0519	-18.6831	-1.3178	0.9885	1.0000
15	J	322.2418	12.1897	1.0378	-18.7900	-1.5279	0.9748	1.0000
17	I	322.2418	12.2895	1.0381	-18.8060	-1.5299	0.9750	1.0000
17	J	337.3793	4.8417	1.0144	-18.2564	-2.0553	0.9541	1.0000
19	I	337.3793	4.9659	1.0147	-18.2730	-2.0577	0.9545	1.0000
19	J	353.5879	-1.1216	0.9968	-17.0771	-2.7104	0.9409	1.0000
21	I	353.5879	-0.9291	0.9974	-17.1051	-2.7140	0.9413	1.0000
21	J	368.7177	-3.2777	0.9911	-14.7403	-3.4673	0.9417	1.0000
23	I	368.7177	-2.9864	0.9919	-14.8200	-3.4725	0.9423	1.0000
23	J	381.6129	0.1552	1.0004	-10.7271	-4.3331	0.9609	1.0000

Figure 43 Pretension Losses (Force) in Tendons

Tendon Elongation:

Tools / ***Unit System***
Length>**in**; Force (Mass)>**kips** ↵

In the **Tree Menu:** Click ***Tables*** tab.
Result Tables>Tendon> **Tendon Elongation** ↵

Model View | **Tendon Elongation**

	Tendon Name	Stage	Step	Tendon Elongation		Element Elongation		Summation	
				Begin (in)	End (in)	Begin (in)	End (in)	Begin (in)	End (in)
▶	TH1	Stage 1	001(first)	4.6604	4.6604	0.0815	0.0815	4.7419	4.7419
	TH2	Stage 1	001(first)	4.6604	4.6604	0.0815	0.0815	4.7419	4.7419
	TH3	Stage 1	001(first)	4.6604	4.6604	0.0815	0.0815	4.7419	4.7419
	TH4	Stage 1	001(first)	4.6604	4.6604	0.0815	0.0815	4.7419	4.7419
	TH5	Stage 1	001(first)	4.6604	4.6604	0.0815	0.0815	4.7419	4.7419
	TH6	Stage 1	001(first)	4.6604	4.6604	0.0815	0.0815	4.7419	4.7419
	TS1	Stage 1	001(first)	4.6459	4.6459	0.1979	0.1979	4.8438	4.8438
	TS2	Stage 1	001(first)	4.6459	4.6459	0.1979	0.1979	4.8438	4.8438
	TS3	Stage 1	001(first)	4.6459	4.6459	0.1979	0.1979	4.8438	4.8438
	TS4	Stage 1	001(first)	4.6459	4.6459	0.1979	0.1979	4.8438	4.8438
	TS5	Stage 1	001(first)	4.6459	4.6459	0.1979	0.1979	4.8438	4.8438
	TS6	Stage 1	001(first)	4.6459	4.6459	0.1979	0.1979	4.8438	4.8438

Figure 44 Tendon Elongation

Influence Line:

Click ***Model View*** tab.
Click **Initial View**
Click **Iso View**
Tools / ***Unit System***
Length>**ft**; Force (Mass)>**kips** ↵

In the **Tree Menu:** Click ***Menu*** tab.
Results > Influence Lines > **Beam Forces/Moments** ↵
Line/Surface Lanes>**LANE all**
Key Element>**13**
Scale Factor>**1**
Parts>**j**
Components>**My**
Type of Display>**Legend** (on)
Click Apply

The influence line diagram for moment (**My**) at the end of Element 13 is displayed. This position corresponds to the mid-span of one of the interior girders.

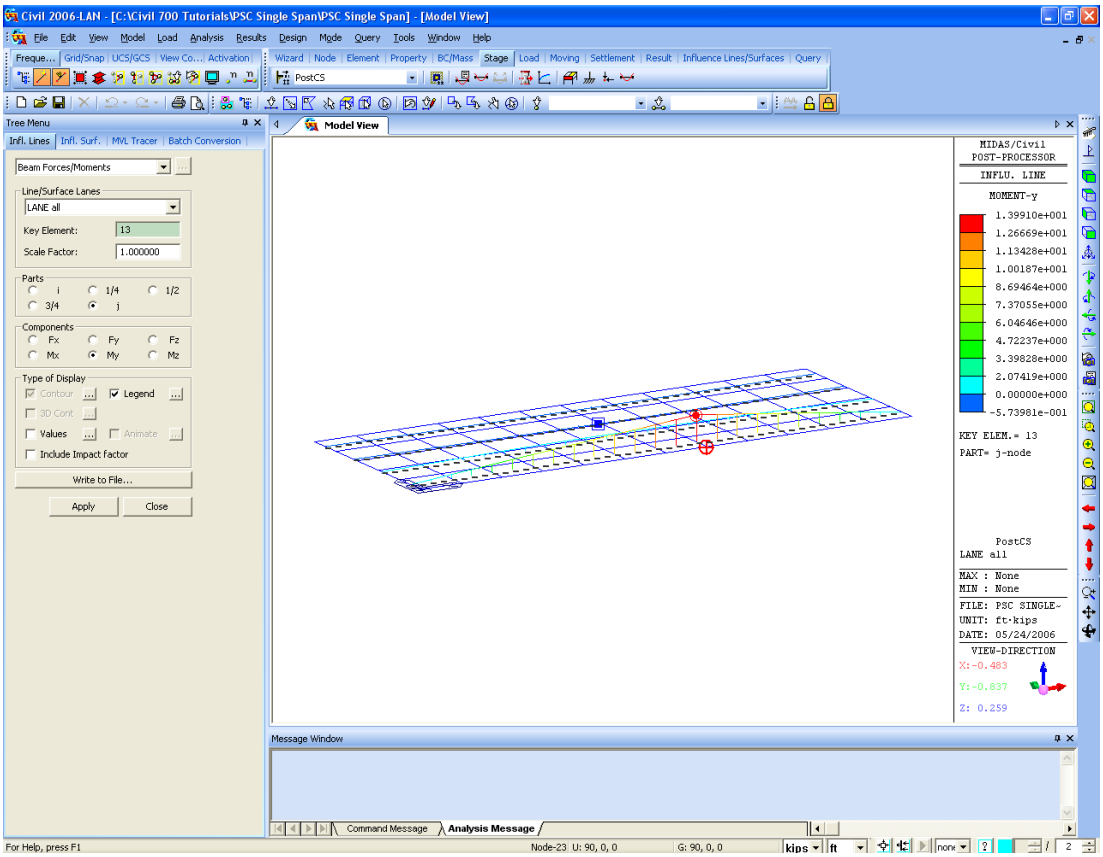


Figure 45 Influence Line Diagram

Moving Load Tracer:

Click ***MVL Tracer*** tab.
Select **Beam Forces/Moments**
Moving Load Cases>**MVmax: MLC**
Key Element>**13**
Scale Factor>**1**
Parts>**j**
Components>**My**
Type of Display>**Contour** (on) ; **Legend** (on) ; **Applied Loads** (on)
Click Apply
Click Close

The position of moving loads that generate maximum moment (**My**) at the end of Element 13 is displayed. This position corresponds to the mid-span of one of the interior girders.

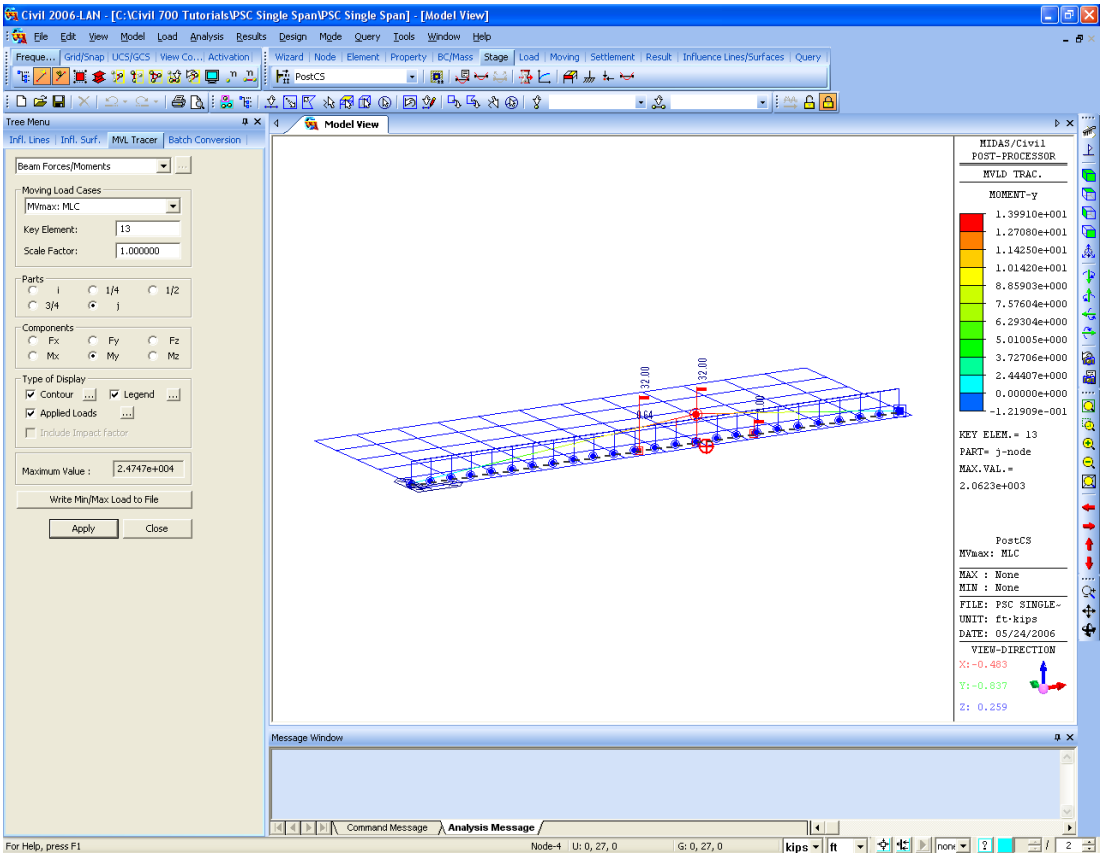


Figure 46 Moving Load Tracer

Stresses in Precast Beams during Construction Stages:

Select Stage 1 in the **Toolbar** (Stage 1)
Tools / ***Unit System***
Length>**in**; Force (Mass)>**kips** ↵

In the **Tree Menu:** Click ***Table*** tab.
Result Tables>Composite Section for C.S.> **Beam Stress** ↵
Records Activation dialog box:
Loadcase/Combination>**Summation(CS)**
Stage/Step>**Stage 1:0003(last) (on) ; Stage 2:0003(last) (on)**
Part Number>**Part j**
Click OK

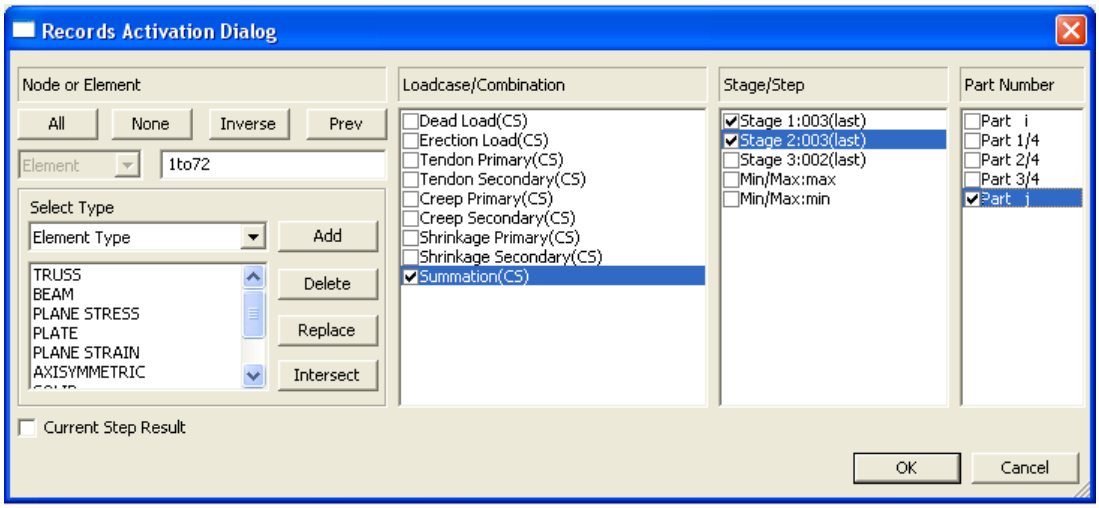
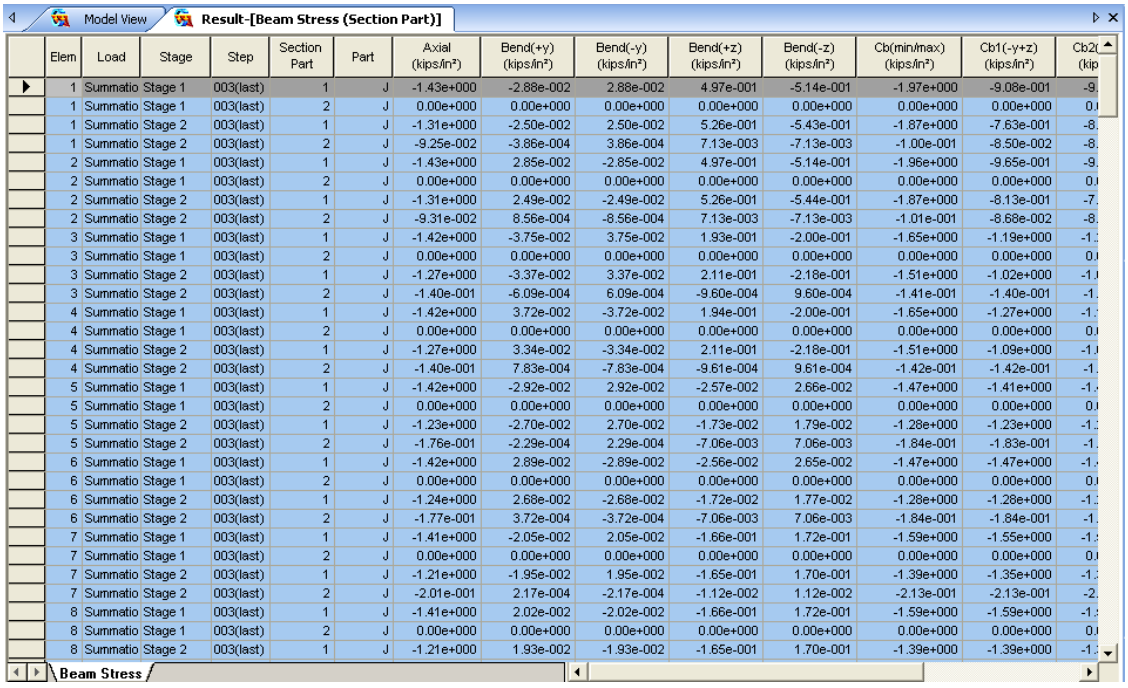


Figure 47 Records Activation Dialog Window

The table showing axial, bending and combined stresses for the precast beams (elements 1to72) at their "j" end in construction stages 1 and 2 is displayed.



Model View | Result-[Beam Stress (Section Part)]

	Elem	Load	Stage	Step	Section Part	Part	Axial (kips/in²)	Bend(+y) (kips/in²)	Bend(-y) (kips/in²)	Bend(+z) (kips/in²)	Bend(-z) (kips/in²)	Cb(min/max) (kips/in²)	Cb1(-y+z) (kips/in²)	Cb2((kip
▶	1	Summatio	Stage 1	003(last)	1	J	-1.43e+000	-2.88e-002	2.88e-002	4.97e-001	-5.14e-001	-1.97e+000	-9.08e-001	-9.
	1	Summatio	Stage 1	003(last)	2	J	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.
	1	Summatio	Stage 2	003(last)	1	J	-1.31e+000	-2.50e-002	2.50e-002	5.26e-001	-5.43e-001	-1.87e+000	-7.63e-001	-8.
	1	Summatio	Stage 2	003(last)	2	J	-9.25e-002	-3.86e-004	3.86e-004	7.13e-003	-7.13e-003	-1.00e-001	-8.50e-002	-8.
	2	Summatio	Stage 1	003(last)	1	J	-1.43e+000	2.85e-002	-2.85e-002	4.97e-001	-5.14e-001	-1.96e+000	-9.65e-001	-9.
	2	Summatio	Stage 1	003(last)	2	J	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.
	2	Summatio	Stage 2	003(last)	1	J	-1.31e+000	2.49e-002	-2.49e-002	5.26e-001	-5.44e-001	-1.87e+000	-8.13e-001	-7.
	2	Summatio	Stage 2	003(last)	2	J	-9.31e-002	8.56e-004	-8.56e-004	7.13e-003	-7.13e-003	-1.01e-001	-8.68e-002	-8.
	3	Summatio	Stage 1	003(last)	1	J	-1.42e+000	-3.75e-002	3.75e-002	1.93e-001	-2.00e-001	-1.65e+000	-1.19e+000	-1.
	3	Summatio	Stage 1	003(last)	2	J	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.
	3	Summatio	Stage 2	003(last)	1	J	-1.27e+000	-3.37e-002	3.37e-002	2.11e-001	-2.18e-001	-1.51e+000	-1.02e+000	-1.
	3	Summatio	Stage 2	003(last)	2	J	-1.40e-001	-6.09e-004	6.09e-004	-9.60e-004	9.60e-004	-1.41e-001	-1.40e-001	-1.
	4	Summatio	Stage 1	003(last)	1	J	-1.42e+000	3.72e-002	-3.72e-002	1.94e-001	-2.00e-001	-1.65e+000	-1.27e+000	-1.
	4	Summatio	Stage 1	003(last)	2	J	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.
	4	Summatio	Stage 2	003(last)	1	J	-1.27e+000	3.34e-002	-3.34e-002	2.11e-001	-2.18e-001	-1.51e+000	-1.09e+000	-1.
	4	Summatio	Stage 2	003(last)	2	J	-1.40e-001	7.83e-004	-7.83e-004	-9.61e-004	9.61e-004	-1.42e-001	-1.42e-001	-1.
	5	Summatio	Stage 1	003(last)	1	J	-1.42e+000	-2.92e-002	2.92e-002	-2.57e-002	2.66e-002	-1.47e+000	-1.41e+000	-1.
	5	Summatio	Stage 1	003(last)	2	J	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.
	5	Summatio	Stage 2	003(last)	1	J	-1.23e+000	-2.70e-002	2.70e-002	-1.73e-002	1.79e-002	-1.28e+000	-1.23e+000	-1.
	5	Summatio	Stage 2	003(last)	2	J	-1.76e-001	-2.29e-004	2.29e-004	-7.06e-003	7.06e-003	-1.84e-001	-1.83e-001	-1.
	6	Summatio	Stage 1	003(last)	1	J	-1.42e+000	2.89e-002	-2.89e-002	-2.56e-002	2.65e-002	-1.47e+000	-1.47e+000	-1.
	6	Summatio	Stage 1	003(last)	2	J	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.
	6	Summatio	Stage 2	003(last)	1	J	-1.24e+000	2.68e-002	-2.68e-002	-1.72e-002	1.77e-002	-1.28e+000	-1.28e+000	-1.
	6	Summatio	Stage 2	003(last)	2	J	-1.77e-001	3.72e-004	-3.72e-004	-7.06e-003	7.06e-003	-1.84e-001	-1.84e-001	-1.
	7	Summatio	Stage 1	003(last)	1	J	-1.41e+000	-2.05e-002	2.05e-002	-1.66e-001	1.72e-001	-1.59e+000	-1.55e+000	-1.
	7	Summatio	Stage 1	003(last)	2	J	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.
	7	Summatio	Stage 2	003(last)	1	J	-1.21e+000	-1.95e-002	1.95e-002	-1.65e-001	1.70e-001	-1.39e+000	-1.35e+000	-1.
	7	Summatio	Stage 2	003(last)	2	J	-2.01e-001	2.17e-004	-2.17e-004	-1.12e-002	1.12e-002	-2.13e-001	-2.13e-001	-2.
	8	Summatio	Stage 1	003(last)	1	J	-1.41e+000	2.02e-002	-2.02e-002	-1.66e-001	1.72e-001	-1.59e+000	-1.59e+000	-1.
	8	Summatio	Stage 1	003(last)	2	J	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.00e+000	0.
	8	Summatio	Stage 2	003(last)	1	J	-1.21e+000	1.93e-002	-1.93e-002	-1.65e-001	1.70e-001	-1.39e+000	-1.39e+000	-1.

Beam Stress

Figure 48 Stresses in Precast Beams during Construction Stages

Bending Moment Diagrams in Precast Beams:

Tools / ***Unit System***
Length>**ft**; Force (Mass)>**kips** ↵
Click **Top View**
Toggle on **Element Number**
Select Window > Elements 25to69by4
Click **Activate**
Click **Front View**

Click ***Model View*** tab.
In the **Tree Menu:** Click ***Menu*** tab
Results > Forces > **Beam Diagrams** ↵
Load Cases/Combinations>**CS: Summation**
Components>**My**
Display Options>**5 Points** (on) ; **Solid Fill** (on)
Type of Display>**Contour** (on) ; **Legend** (on)
Click Apply

The bending moment diagram (**My**) for the selected interior precast beams (elements 25to69by4) in the current construction stage (Stage 1), and under all the construction stage loads applied simultaneously, is displayed.

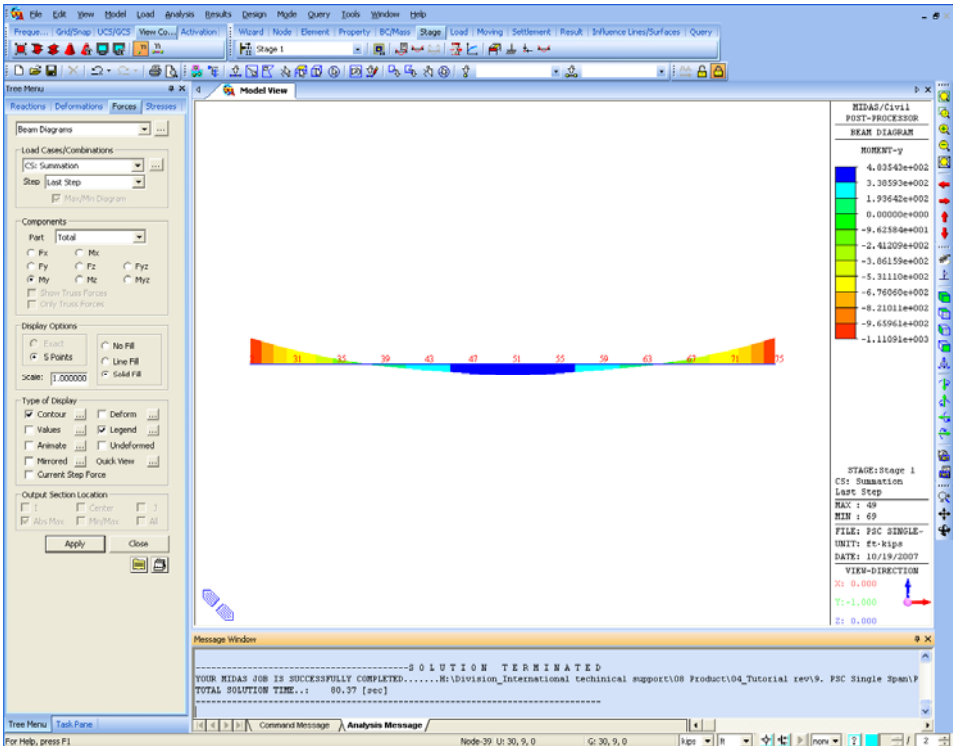


Figure 49 Stage 1 Bending Moment Diagram of Precast Beams

Toggle on **Active Fix** in the ***Status Bar***
Select **Post Construction Stage** (PostCS).
Load Cases/Combinations> **CBall: RC ENV_STR**
Components>**My**
Click Apply

The post-construction stage (Post CS) envelope of bending moment diagram (**My**) under strength condition for the selected interior precast beams (elements 25to69by4), is displayed.

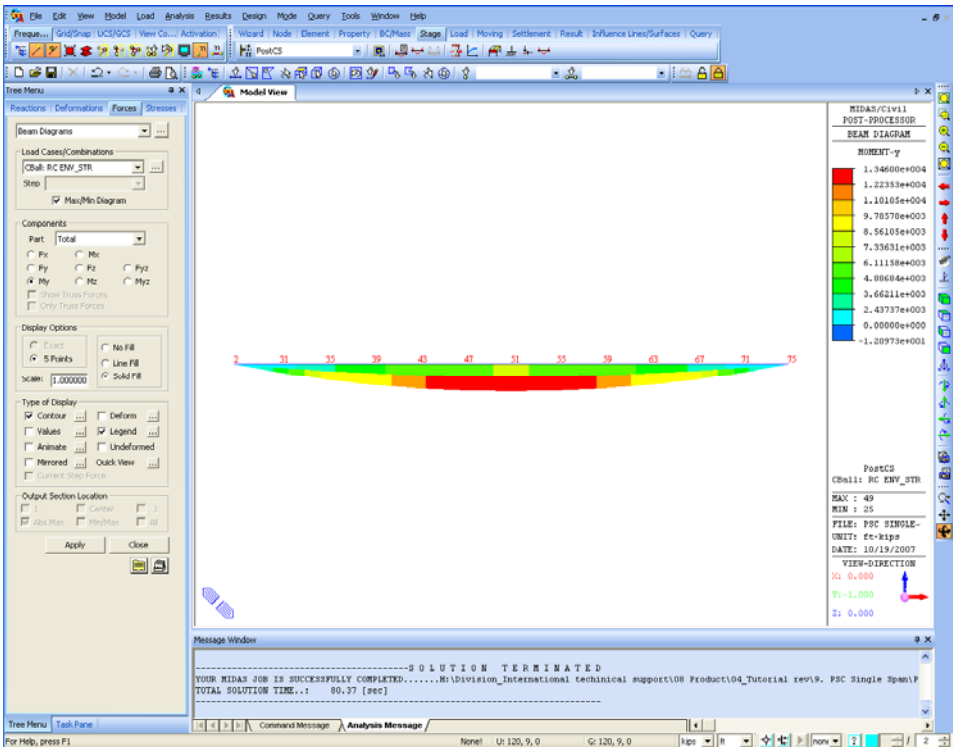


Figure 50 Post-Construction Stage Bending Moment Diagram Envelope of Precast Beam

Shear Force Diagrams in Precast Beams:

Load Cases/Combinations> **CBall: RC ENV_STR**
Components>**Fz**
Click Apply
Click Close

The post-construction stage (Post CS) envelope of shear force diagram (**Fz**) under strength condition for the selected interior precast beams (elements 25to69by4), is displayed.

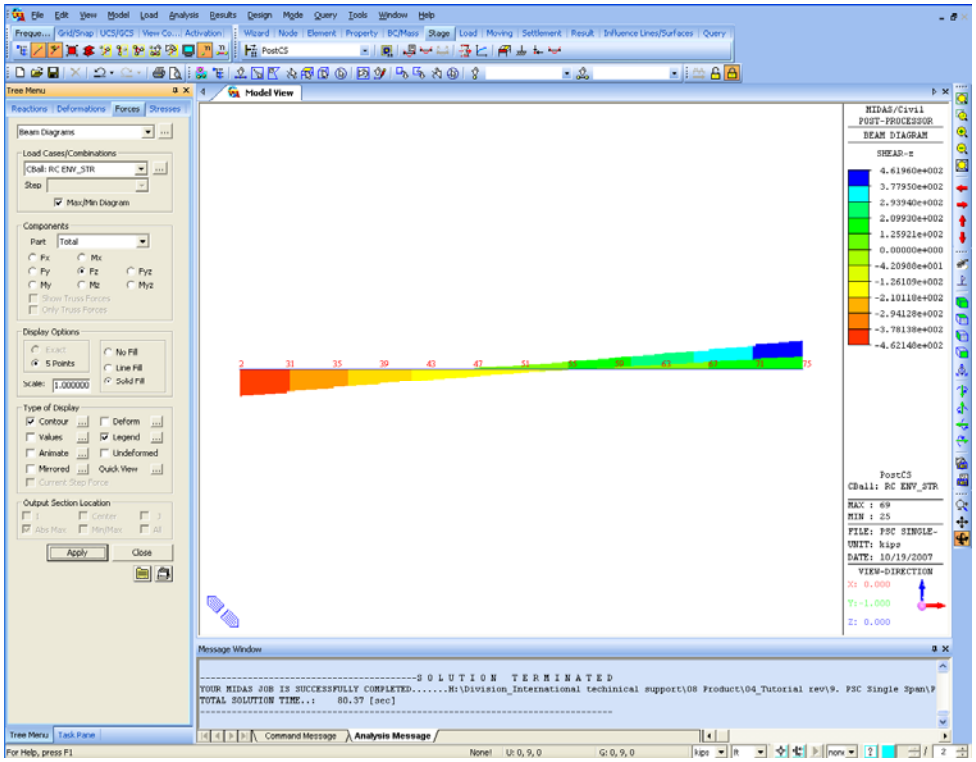


Figure 51 Post-Construction Stage Shear Force Diagram Envelope of Precast Beams

Reactions:

In the **Tree Menu:** Click **Tables** tab.
Result Tables > **Reaction** ↵
Records Activation dialog box>Loadcase Combination>**gLCB1(CB:max)**
Click OK

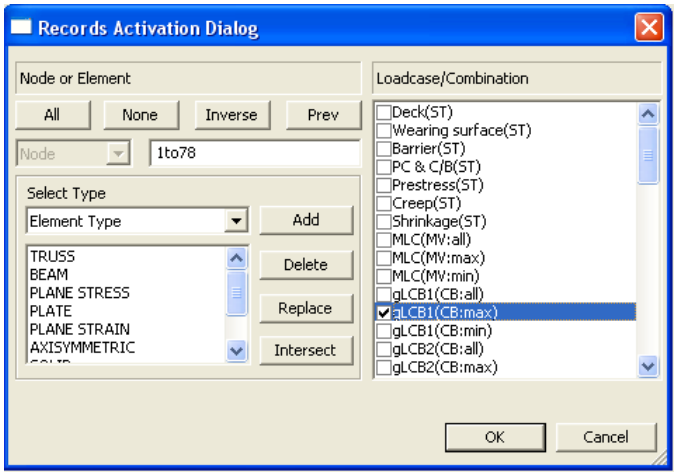


Figure 52 Records Activation Dialog Box

The table showing the maximum reactions corresponding to Load Combination LCB1 in the post construction stage (Post CS) is displayed.

Model View | Result-[Reaction]

Node	Load	FX (kips)	FY (kips)	FZ (kips)	MX (ft·kips)	MY (ft·kips)	MZ (ft·kips)
1	gLCB1(max)	-17.190713	18.705456	413.591312	0.000000	0.000000	0.000000
2	gLCB1(max)	75.214480	0.000000	493.544096	0.000000	0.000000	0.000000
3	gLCB1(max)	69.972350	0.000000	508.548977	0.000000	0.000000	0.000000
4	gLCB1(max)	71.436440	0.000000	508.547997	0.000000	0.000000	0.000000
5	gLCB1(max)	80.502018	0.000000	493.548241	0.000000	0.000000	0.000000
6	gLCB1(max)	-18.148774	0.000000	413.578517	0.000000	0.000000	0.000000
29	gLCB1(max)	0.000000	21.036127	413.554554	0.000000	0.000000	0.000000
30	gLCB1(max)	0.000000	0.000000	413.558670	0.000000	0.000000	0.000000
75	gLCB1(max)	0.000000	0.000000	493.409844	0.000000	0.000000	0.000000
76	gLCB1(max)	0.000000	0.000000	508.436341	0.000000	0.000000	0.000000
77	gLCB1(max)	0.000000	0.000000	508.436712	0.000000	0.000000	0.000000
78	gLCB1(max)	0.000000	0.000000	493.414854	0.000000	0.000000	0.000000
SUMMATION OF REACTION FORCES PRINTOUT							
	Load	FX (kips)	FY (kips)	FZ (kips)			
	gLCB1(max)	N/A	N/A	N/A			

Figure 53 Post-Construction Maximum Reactions due to Load Combination LCB1