Advanced Application 6

Construction Stage Analysis for ILM

Civil

CONTENTS

Overview

The basic concept of Incremental Launching Method (ILM) is that approximately 15 to 30m long units of bridge segments are prefabricated under plant conditions behind an abutment and launched by means of sliding (on Teflon bearings), one segment at a time. Each current prefabricated segment is post-tensioned with the previously erected box girder segments and pushed in the direction of the bridge by launching equipment, which consists of a combination of hydraulic jacks acting vertically and horizontally and sliding bearings. The structural system continuously changes during the construction stage. The geometry, support condition and loading condition of the temporary structure of each stage vary from one stage to the next, without having any resemblance to the finally completed structure. In addition, bridges constructed by ILM are affected by time dependent material properties such as concrete strength (elasticity of modulus), creep and shrinkage. Without reflecting all these time dependent variables, analysis results will deviate considerably from the true behaviors.

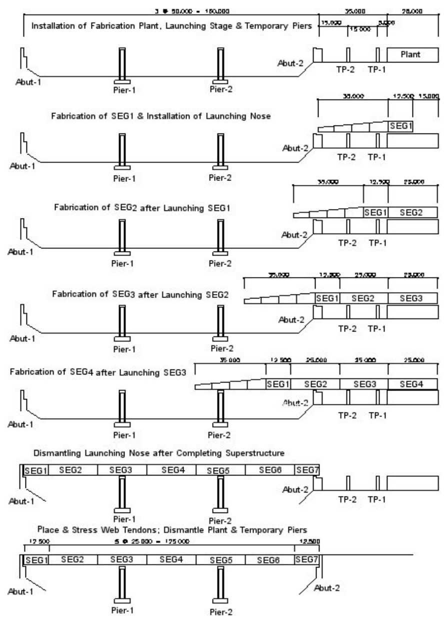


Figure 1 Construction stage of super structure

This example examines the procedure of carrying out a construction stage analysis for a PSC (pre-stressed or post-tensioned concrete) box bridge constructed by ILM reflecting the change of structural system.

MIDAS/Civil provides the following special functions for the convenience of analysis and design by ILM:

I. ILM Bridge Model Wizard : automatic generation of bridge model including tendon placement.

II. ILM Bridge Stage Wizard : automatic definition of conditions such as element activation and deactivation, change of boundary condition, loading input, etc. for each construction stage.

The construction stage analysis by ILM must reflect upon the changes in boundary conditions and loadings through advancing construction stages as shown in Fig. 1. The following outlines the procedure for performing a construction stage analysis of an ILM bridge by using ***ILM Bridge Wizard*:**

1. Define material properties and sections
2. Model the bridge using ILM Bridge Model
3. Define the construction stages using ILM Bridge Stage
4. Perform structural analysis
5. Verify results

The example presents the bridge type and span configuration as follows:

Bridge type: PSC box bridge continuous over 3 spans (constructed by ILM)

Bridge Length: L = 3@50.0 = 150.0 m

Bridge width: B = 12.315 m

Skew: 90˚(Straight bridge)

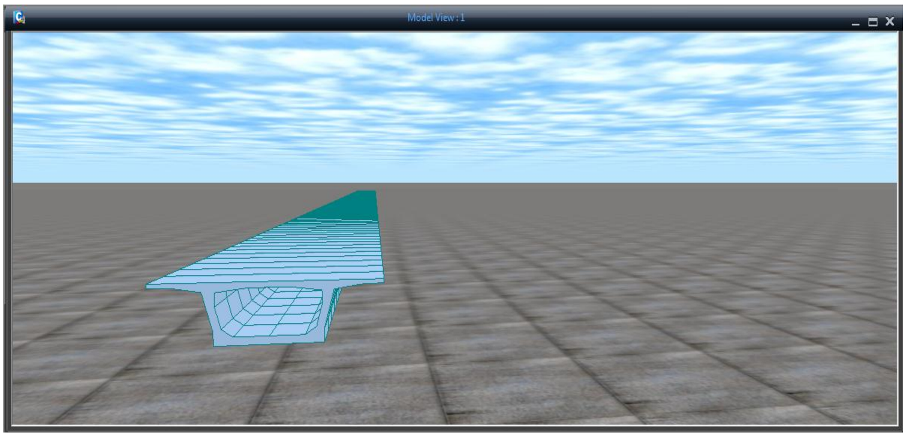


Figure 2 Analysis model

Cross Section

[unit : mm]

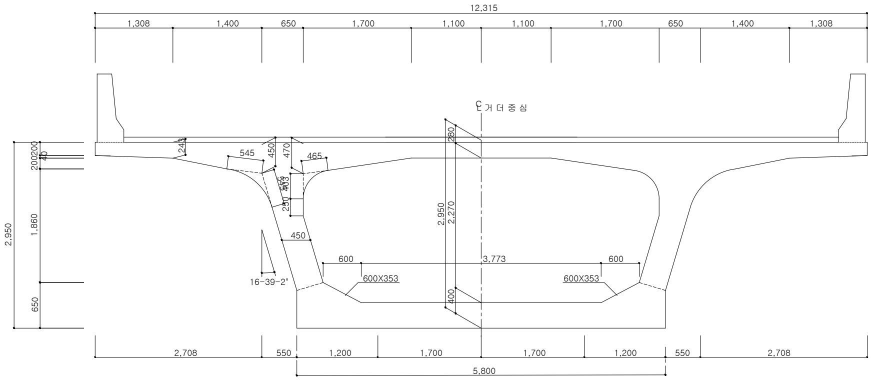


Figure 3 Typical section

In the case of a PSC bridge constructed by ILM, the tendon placement generally takes place in two main stages. The primary tendons placed in the top and bottom slabs resist the self-weight. The secondary tendons are placed and tensioned in the webs after the entire bridge becomes completed (upon completion of launching all the bridge segments).

The primary tendons are post-tensioned in 2 cycles (tensioning over 2 segments) or 3 cycles (tensioning over 3 segments). This example adopts a 2-cycle tensioning method. The first cycle and second cycle tendons can be separately identified.

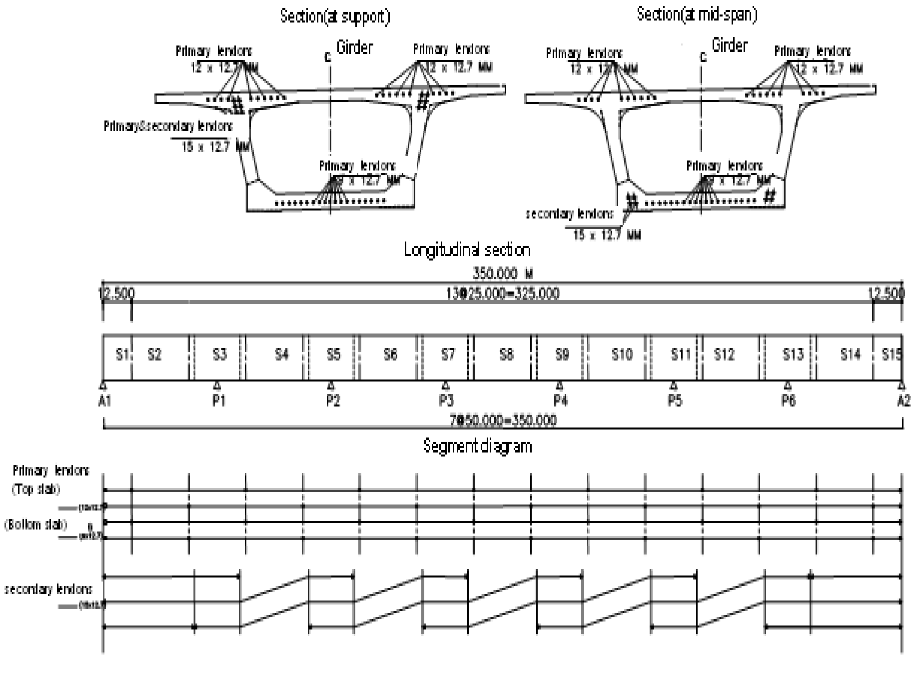


Figure 4 Segment diagram and tendon layout

Maximum forces result in the bridge superstructure constructed by ILM immediately before a pier supports it during the construction stage. This pertains to a stage where its cantilever becomes the longest. In order to reduce the high temporary negative moment, a lightweight structural steel girder, which is referred to as a launching nose, is attached to the front segment. The length of the nose is usually 70% of the normal span, and its stiffness is about 10% of the PSC box girder. The actual configuration must be determined on the basis of the span, stiffness and self-weight of the bridge.

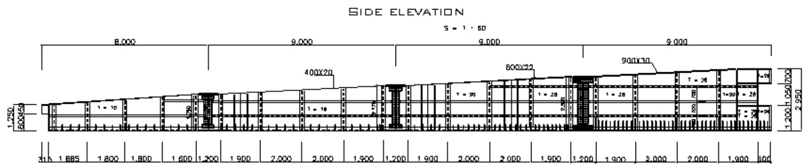


Figure 5 Nose Side Elevation

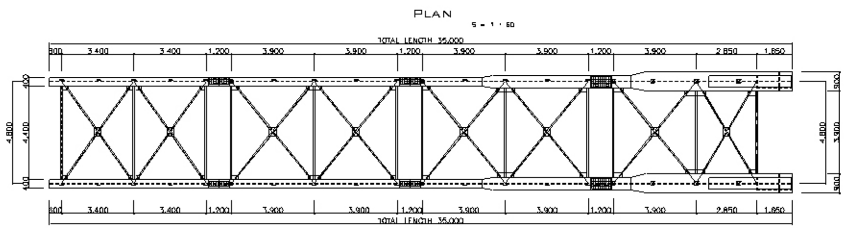


Figure 6 Nose Plan

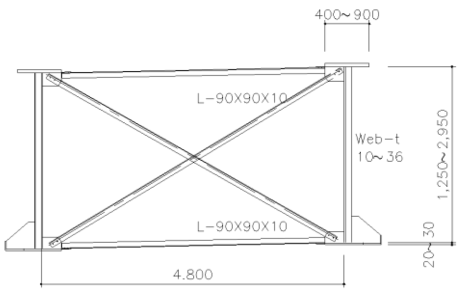


Figure 7 Nose Section

Materials and Allowable Stresses

- Concrete (Using compressive strength gain curve for concrete maturity)
  Design compressive strength : ASTM Grade C5000

- Prestressing strands: Φ12.7 mm (0.5″)

  Yield strength: $f_{py} = 1600\ N/mm^2$
  Ultimate strength: $f_{pu} = 1900\ N/mm^2$
  Modulus of elasticity: $E_p = 2.0\times10^5\ N/mm^2$
  Prestress transfer: $f_{pj} = 0.7 f_{pu} = 1330\ N/mm^2$
  Anchorage slip: $\Delta s = 6\ \mathrm{mm}$
  Friction loss coefficients: $\mu = 0.30\ /\mathrm{rad}$
  $k = 0.0066\ /\mathrm{m}$

Loads

- **Dead load (self-weight applied during erection)**
  The program automatically computes the self-weight.
  The self-weights of diaphragm, deviation blocks and anchor blocks can be input as beam loads. (nose connection part: 763 kN, pier part: 516.1 kN)

- **Prestress**
  - Primary tendon
  Top tendon : $\varphi 12.7\ \mathrm{mm} \times 12 (\varphi 0.5 - 12)$
  $A_p = 0.9871 \times 12 = 11.85\ \mathrm{cm}^2$
  Duct size : 63 mm

  Bottom tendon : $\varphi 12.7\ \mathrm{mm} \times 9 (\varphi 0.5 - 9)$
  $A_p = 0.9871 \times 9 = 8.88\ \mathrm{cm}^2$
  Duct size : 51 mm

  - Secondary tendon : $\varphi 12.7\ \mathrm{mm} \times 15 (\varphi 0.5 - 15)$
  $A_p = 0.9871 \times 15 = 14.81\ \mathrm{cm}^2$
  Duct size : 75 mm
  - Post-tensioned force: 70% of tension strength at transfer
  $f_{pj} = 0.7 f_{pu} = 1330\ N/mm^2$

Loss immediately after anchoring (calculated internally by program)

Friction loss: $P_{(X)} = P_0 \cdot e^{-(\mu\alpha + kL)}$ ($\mu = 0.30$, $k = 0.0066$)

Loss due to anchorage slip: $\Delta I_c = 6$ mm

Loss due to elastic shortening: magnitude of loss, $\Delta P_E = \Delta f_P \cdot A_{SP}$

Final loss (calculated internally by program)

Relaxation

Losses due to creep and shrinkage

- **Creep and Shrinkage**

- Conditions

Cement: Normal Portland cement

Concrete age at the time of launching: $t' = 7 days$

Concrete age when exposed to air: $t_s = 3 days$

Relative humidity: RH = 70 %

Ambient or curing temperature: $t = 20°C$

- Creep coefficient (calculated internally by program as per CEP-FIP code)

Deformation due to concrete creep and shrinkage (calculated internally by program as per CEP-FIP code)

Setting Work Environment and Defining Section/Material Properties

To model the **ILM** bridge, open a new file (***New Project***) and save (***Save***) as 'ILM-Bridge'.

➢ ***New Project***
➢ ***Save* (ILM-Bridge)**

Setting Work Environment

Define the unit system for modeling with kN (force) and m (Length).

Tools / ***Unit System***
Length>**m** ; Force>**kN** ↵

Unit system may be changed using the status bar at the lower part of the screen (Fig. 8)

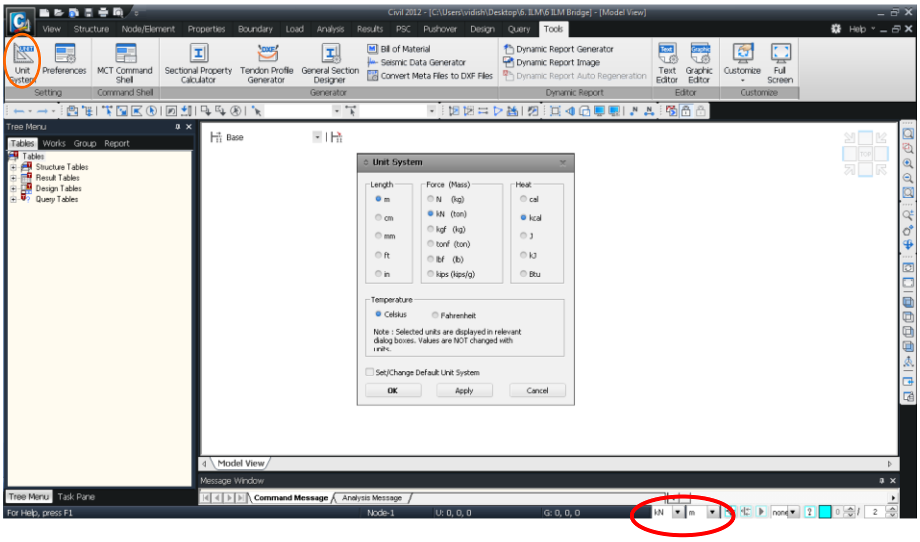


Figure 8 Initial screen and unit system dialog box

Definition of Material Properties

Define the material properties for the nose and girder using the database contained in MIDAS/Civil. Define the tendon as a user defined type by entering only its modulus of elasticity.

Use the Apply button when entering a number of material properties at the same time.

Property Tab > ***Material Properties***

Type>**Steel** ; Standard>**ASTM(S)**

DB>**A36** ↵

Type>**Concrete** ; Standard>**ASTM(RC)**

DB>Grade **C5000** ↵

Type>**User Defined** ; Name **(Tendon)**; Standard>**None**

Analysis Data>Modulus of Elasticity **(2e8)** ↵

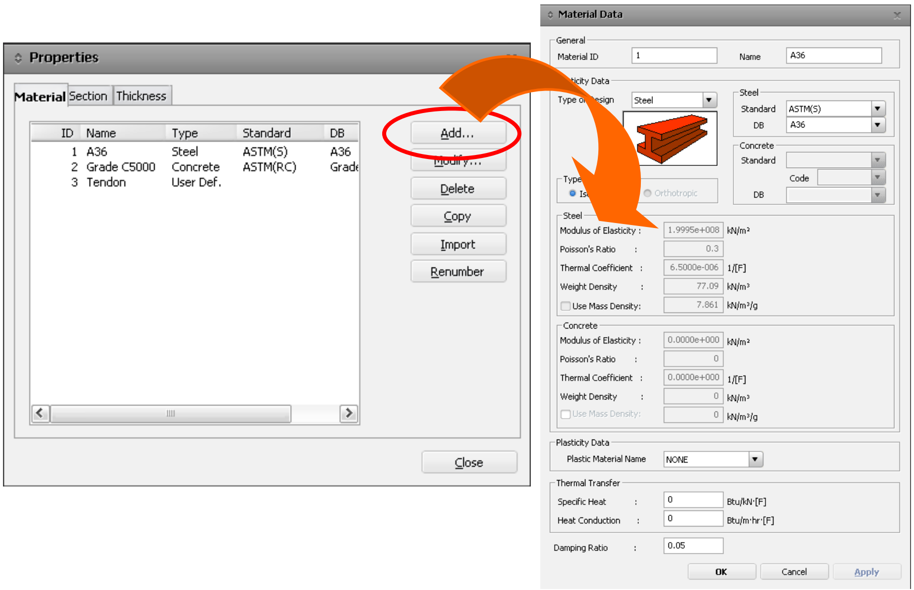


Figure 9 Input for Material Property data

The characteristics of time dependent material properties are separately defined to reflect the changing modulus of elasticity due to the change of concrete strength and creep and shrinkage based on maturity. The time dependent material properties are defined as per CEB-FIP code.

Notational size of member is automatically assigned by selecting the element after modeling the girder. For details, refer to "Using Civil >Model>Properties>Change element dependent property" of On-line manual

- 28 day strength : 35000 KN/m^2
- Relative humidity: 70%
- Notational size : automatically calculated based on box girder section area and perimeter length
- Type of concrete : normal weight concrete
- Time of form removal : 3 days after casting (time at which shrinkage begins)

ACI, CEB-FIP or user-defined properties can be used for Creep and shrinkage properties. If user defined, appropriate properties are directly defined in Time Dependent Material (Creep/Shrinkage) Function.

Properties Tab > Time Dependent Material > ***Creep & Shrinkage* >Add**

Name **(Mat-1)** ; Code>**CEB-FIP(1990)**

Compressive strength of concrete at the age of 28 days **(35000)**

Relative humidity of ambient environment (40 ~ 99) **(70)**

Notational size of member **(1)**

Type of cement>**Normal or rapid hardening cement (N, R)**

Age of concrete at the beginning of shrinkage **(3)** ↵

Click Show Result... to check the entered creep and shrinkage properties in graphs.

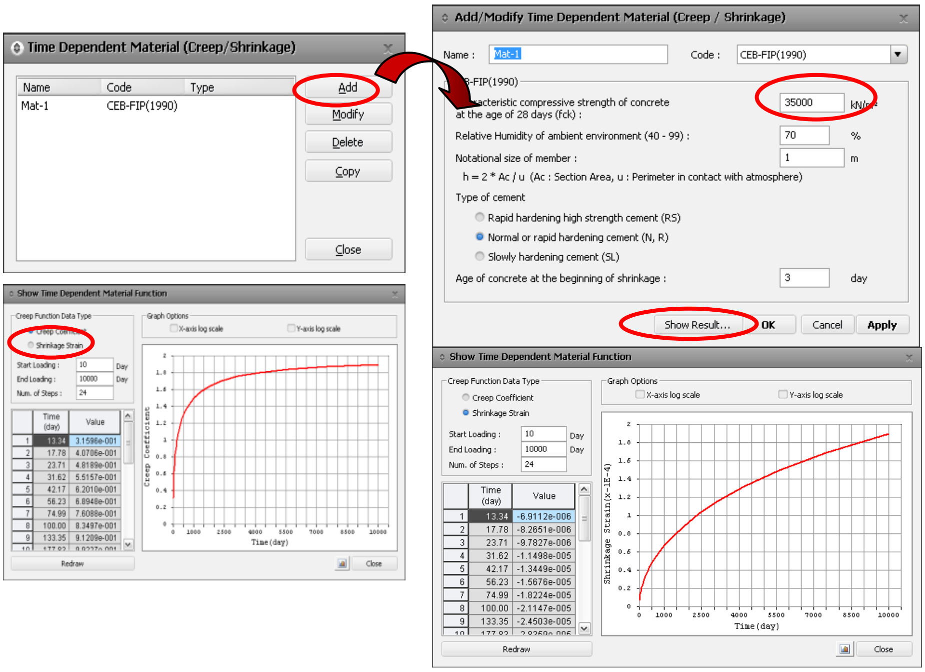


Fig. 10. Definition of time dependent properties of concrete (Creep and shrinkage)

Concrete strength increases with time. We will use the CEB-FIP code for the concrete strength gain function in this example. Refer to the material properties used for defining the creep and shrinkage properties.

In defining the concrete strength development function, the proposed equations of ACI, CEB-FIP and Ohzagi can be adopted.

Properties Tab > ***Comp. Strength***
Name **(Mat-1)** ; Type>**Code**
Development of Strength>Code>**CEB-FIP**
Mean Concrete Compressive Strength at 28 Days (S28)**(38000)**
Cement Type(a) **(N, R : 0.25)** Redraw Graph ↵

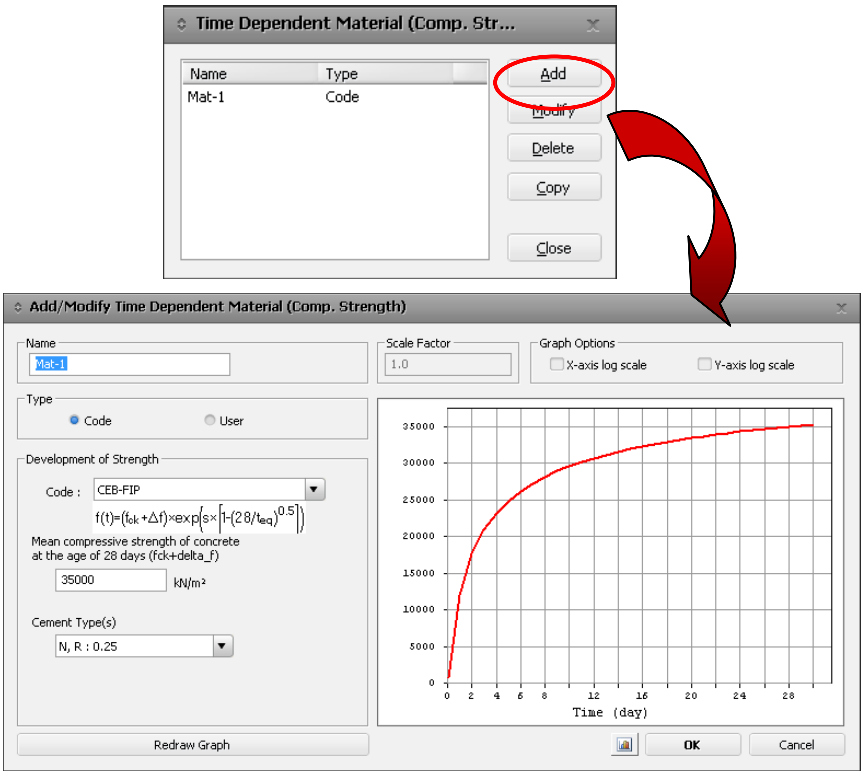


Figure 11 : Definition of strength development function varying with time

MIDAS/Civil requires that the time dependent and general material properties be independently defined and subsequently linked. This example links the material properties (C5000) of the box girder to the time dependent material properties.

Properties Tab >Time Dependent Material ***Material Link***
Time Dependent Material Type>Creep/Shrinkage>**Mat-1**
Comp. Strength>**Mat-1**
Select Material for Assign>Materials>**2:Grade C5000** > Selected Materials ; Operation> Add / Modify

---

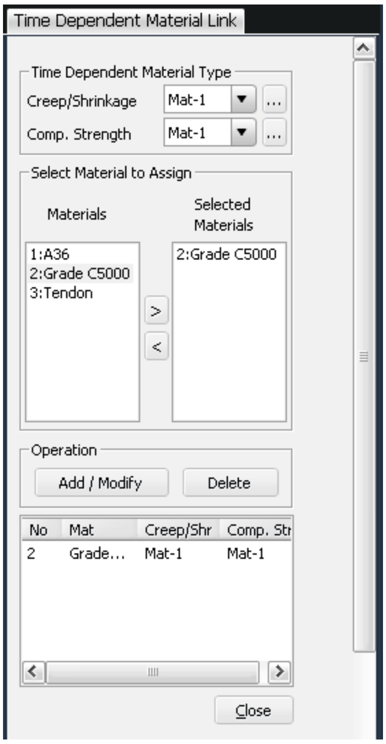


Figure 12 Link between general and time dependent material properties

Definition of Section

The cross section of the launching nose is a tapered section varying from one end to the other, which is made up of a pair of girders inter-connected by cross bracings. Since the PSC box is modeled as a single member, the double-section nose girder will be idealized as a single member as well. For equivalent sections, the flange width and web thickness are doubled at each end. “mm” unit is used for dimensions.

Nose section

Location	Actual girder size	Equivalent girder size
End of nose	I 1250×400×10/20	I 1250×800×20/20
Connection part to the main	I 2950×900×26/30	I 2950×1800×52/30

Offset refers to a location on the cross-section, at which the stiffness center of the beam element is located. If Center-Bottom is selected for the Offset, separate offset distances need not be entered to specify boundary conditions.

Tools Tab / ***Unit System***

Length>**mm**

Properties Tab > ***Section Properties***>Add

Tapered tab

Section ID **(1)** ; Name **(Nose)** ; Offset>**Center-Bottom**

Section Type>**I-Section** ; **User**

Section-i >H **(1250)** ; B **(800)** ; tw **(20)** ; tf1 **(20)**

Section-j >H **(2950)** ; B **(1800)** ; tw **(52)** ; tf1 **(30)**

y-Axis Variation>**Cubic** ; z-Axis Variation>**Cubic** ↵

y(z)-Axis Variation defines the method of varying element stiffness in y(z)-Axis. For details, refer to “Using CIVIL> Properties>Section” of On-line manual.

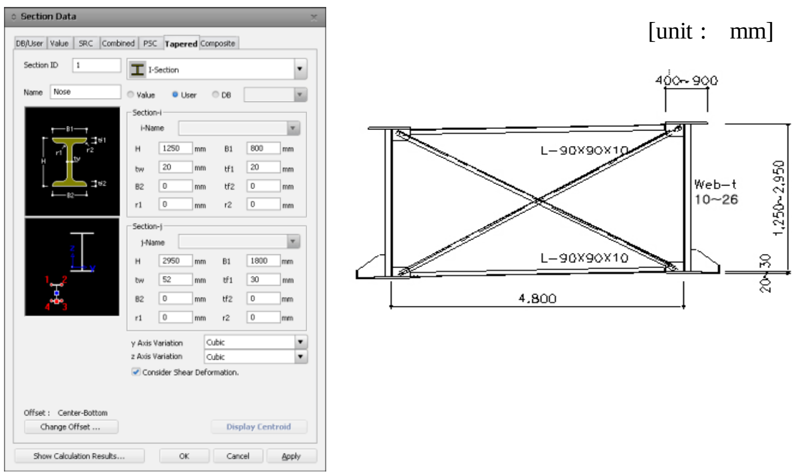


Figure 13 Nose section & input dialog

The shape and section dimensions of the girder are as shown in Fig. 14.

[Unit : mm]

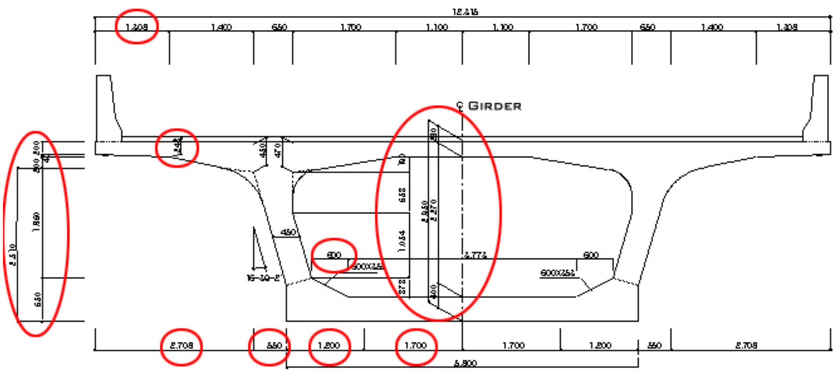


Figure 14 Typical section of bridge

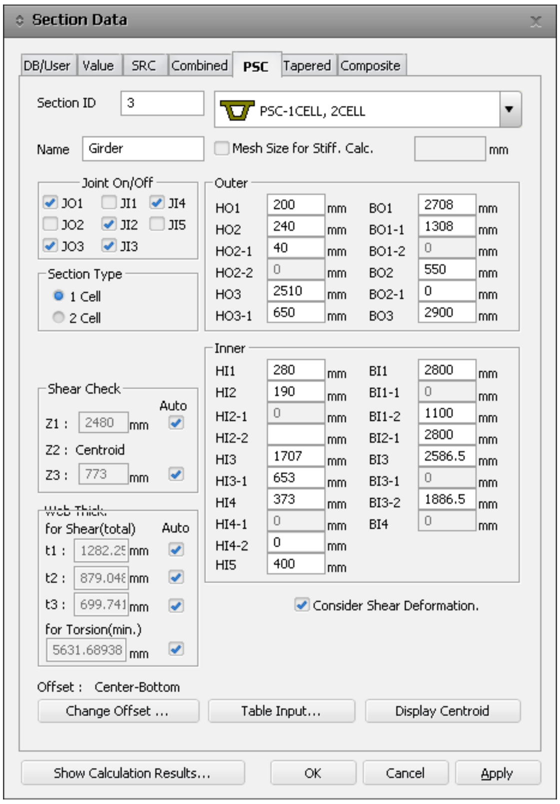


Figure 15 Section data input

PSC tab

Section ID **(2)** ; Name **(Girder), PSC-1Cell, 2Cell**

Section Type>**1 Cell**

Joint On/Off>**JO1** (on), **JO3** (on), **JI2**(on), **JI3**(on), **JI4**(on)

Offset>**Center-Bottom**

Outer

HO1 **(200)** ; HO2 **(240)** ; HO2-1 **(40)**

HO3 **(2510)** ; HO3-1 **(650)**

BO1 **(2708)** ; BO1-1 **(1308)** ; BO2 **(550)**

BO2-1 **(0)** ; BO3 **(2900)**

Inner

HI1 **(280)** ; HI2 **(190)** ; HI2-2 **(0)** ; HI3 **(1707)**

HI3-1 **(653)** ; HI4 **(373)** ; HI4-2 **(0)** ; HI5 **(400)**

BI1 **(2800)** ; BI1-1 **(1100)** ; BI2-1 **(2800)**

BI3 **(2486.5)** ; BI3-1 **(1886.5)** ↵

Shear Check >Z1 & Z3>Auto (Checked)

Web Thick> t1, t2, t3 & For Torsion (min.)>(Checked)

Consider Shear Deformation > (Checked)

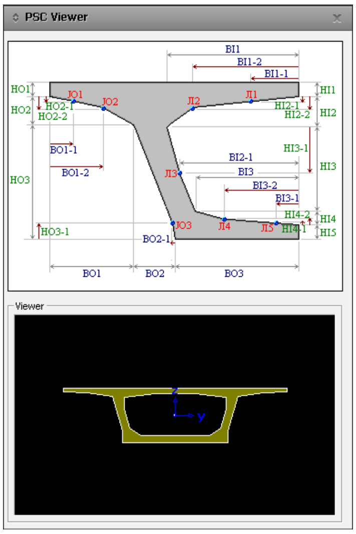


Figure 16 Section shape reflecting the dimension entries

Bridge Modeling using ILM Bridge Model Wizard

An ILM Bridge can be modeled using the ILM Bridge Model Wizard or general modeling functions of MIDAS/Civil. We will first review the method of generating a bridge model using the ILM Bridge Model Wizard, which includes the nose and tendon placement.

If the general method is applied, the 2-D (X-Z) plane needs to be defined in the Structure Type for this straight bridge example. Since we model the bridge by the ILM Bridge Model Wizard, the Wizard automatically determines if it is 2-D or 3-D.

ILM Model

Launching Nose & Bridge Information

The element length for the launching nose is 2.5 m, and the nose is 35m long, which generates a 14@2.5m nose. The number of workdays for each segment is set to 12 days. The initial maturity of concrete is set to 7 days, which means the concrete is poured after 5 days of re-bar placing and cured for 7 days before launching.

Tools Tab > Unit System > Length > m
Structure Tab > Wizard / ***ILM Bridge/ ILM Bridge Model***
ILM Model tab
Bridge Information
Element Length **(2.5)** ; Stage Duration **(12)**
Segment Age **(7)**
Launching Nose
Material>**1:A36** ; Section>**1:Nose** ; Length **(35)**

The age (7 days) at which the concrete segment becomes activated.

When a curved bridge is modeled, the radius of the bridge is defined.

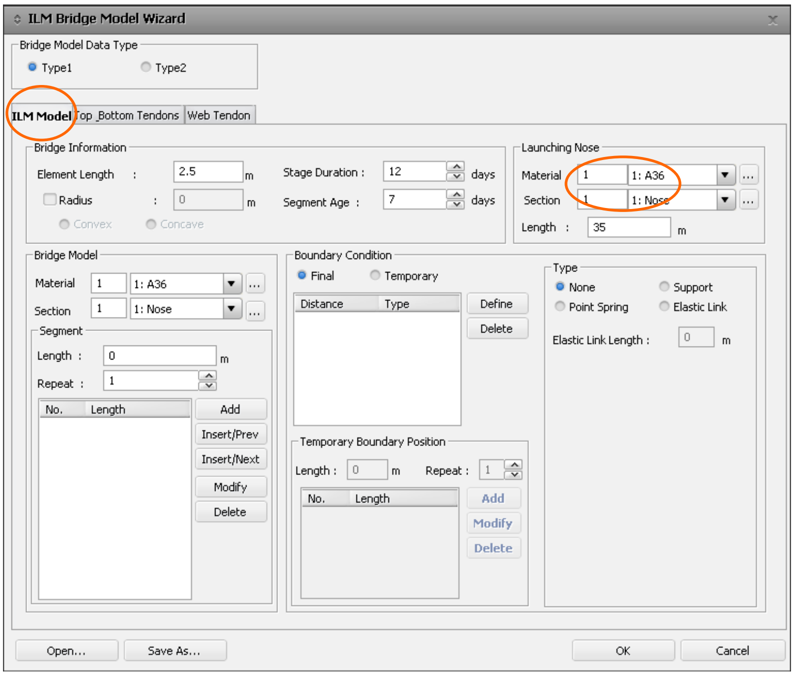


Figure 17 ILM Bridge Model Wizard Dialog Box-ILM Model

Bridge Model

We now enter the information on the material and section properties of the bridge segments. There are 15 segments (L = 12.5 + 5@25.0 + 12.5 = 150.0 m) in total. Each segment length must be a multiple of the launching distance, otherwise it will prompt an error message when the Add button is clicked.

Bridge Model

Material>**2:Grade C5000** ; Section>**2:Girder**

Segment>Length **(12.5)** ; Repeat **(1)** Add

Segment>Length **(25)** ; Repeat **(5)** Add

Segment>Length **(12.5)** ; Repeat **(1)** Add

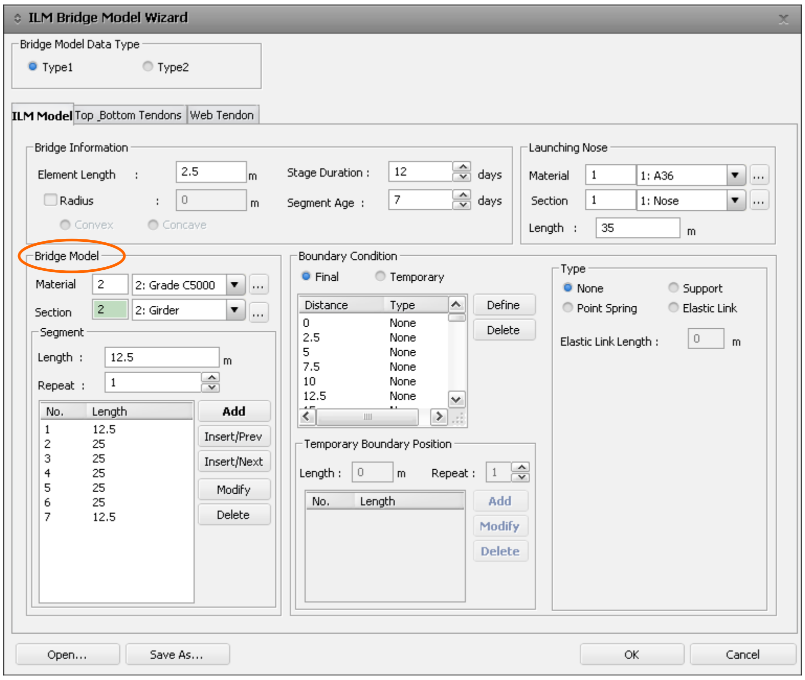


Figure 18 ILM Bridge Model Wizard - Bridge Model Input Dialog Box

Boundary Conditions

Boundary condition for the Final (completed) structure

The support condition of piers and abutments (3@50 m = 150 m) is entered and used for the construction stage analysis. In this example, a hinge condition is assigned at the launching abutment where braking saddle is located throughout all the stages. The boundary condition for the final structure is entered at the last stage.

A number of piers (support conditions) can be selected while the [Ctrl] key is pressed.

Boundary Condition

Final

Distance>**0, 50, 100, 150**

Type>**Support** ; **Dy, Dz, Rx**(on) Define

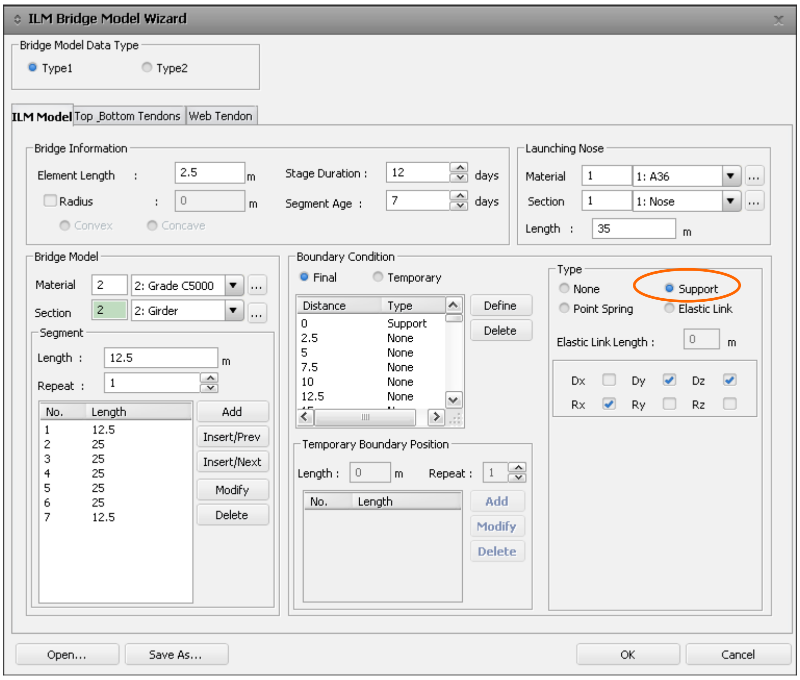


Fig. 19 ILM Bridge Model Wizard – Piers of Final System

Boundary Condition Input for Fabrication Plant

The Temporary Boundary Position defines the boundary condition reflecting the box girder fabrication plant and temporary piers (refer to ① of Fig. 20). These are not capable of resisting uplift reactions, and as such the “compression-only” type of boundary conditions are assigned. To avoid a Singular Error (instability) during the stages, a boundary constraint (Dx) is assigned to the end of the bridge.

Boundary Condition

Temporary

Temporary Boundary Position>Length **(150)** ; Repeat **(1)** Add

Temporary Boundary Position>Length **(15)** ; Repeat **(2)** Add

Temporary Boundary Position>Length **(5)** ; Repeat **(1)** Add

Temporary Boundary Position>Length **(10)** ; Repeat **(2)** Add

Temporary Boundary Position>Length **(5)** ; Repeat **(1)** Add

Distance>**150**

Type>**Support** ; **Dx** (on) Define

Distance>**165, 180, 185, 195, 205, 210**

Type>**Elastic Link** ; Elastic Link Length **(1)**

Link Type>**Comp.-only** ; SDx **(1e10)** ; Beta Angle **(0)** Define

The length in Temporary Boundary position represents the relative distance from the starting point of the bridge. Note that the launching direction starts from the finishing point of the bridge (right side)

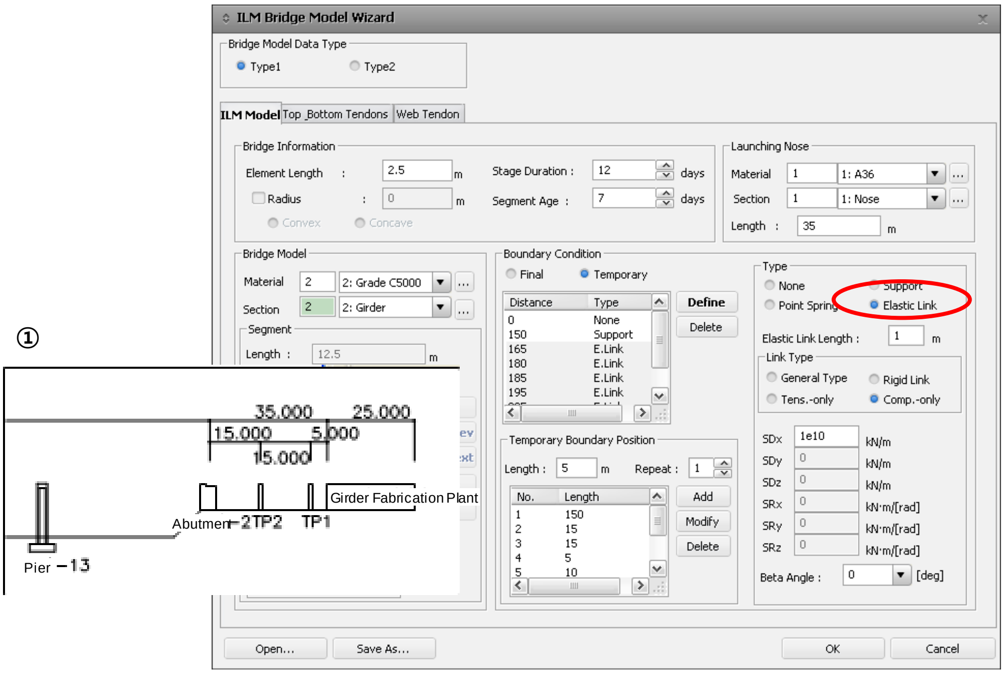


Fig. 20 ILM Bridge Model Wizard – Boundary Condition

Top & Bottom Tendons

We enter the slab tendons (primary tendons), which are tensioned at the time of launching.

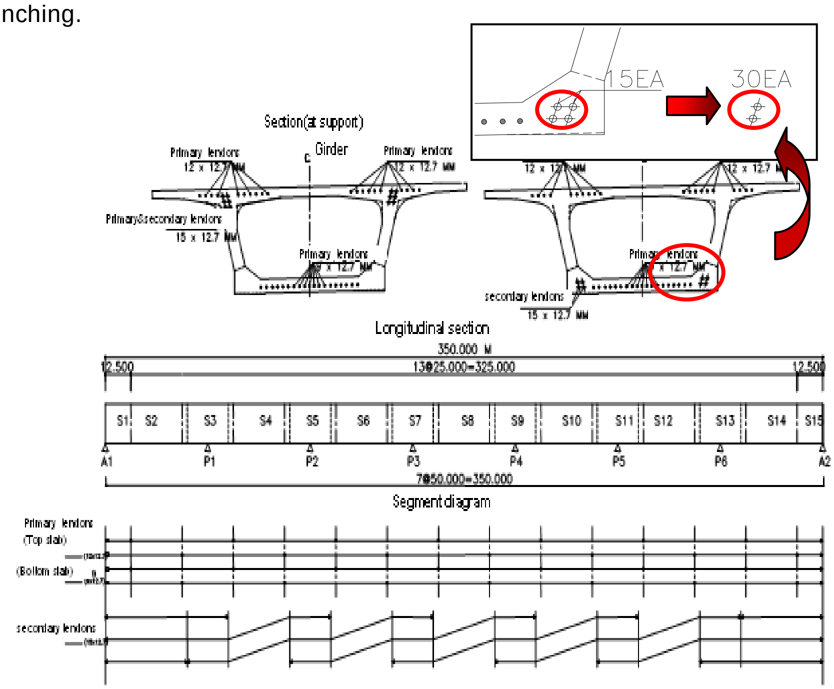


Figure 21 The conceptual tendon placement

Tendon property definition

First, we define the characteristics of the primary (slab) and secondary (web) tendons. The tendons are defined as to whether or not they are internal or external, followed by defining tendon section area, duct diameter, relaxation coefficient, curvature friction factor, wobble friction factor, ultimate strength, yield strength, jacking method and the slip quantity at the anchor locations. Table 1 summarizes the number of strands, duct diameters and tendon names pertaining to each location.

Table 1 Tendon information at each location

Location		Number of strands	Duct diameter	Tendon name
Primary tendon	Top slab	12EA	0.063 m	TT
	Bottom slab	9EA	0.051 m	BT
Secondary tendon		30EA	0.106 m	WT

Select 'One Cell' among the types standardized in MIDAS/CIVIL, and define the primary and secondary tendons at the same time. The secondary (web) tendons are made up of 15 strands. Assume that 2 tendons are combined into one, thereby defining the number of strands as 30 and the duct diameter as $\sqrt{2}\phi = \sqrt{2} \times 0.075\text{m} = 0.1061\text{m}$ (refer to the detail portion of Fig. 21). Specify the jacking force of the tendons assuming 70% of the ultimate strength. Refer to the example definition part for various coefficients.

Top & Bottom Tendon tab (Fig. 22 ①)

Type> One Cell

Tendon Property ... ; Add

Tendon Name **(TT)** ; Tendon Type>**Internal** ; Material>**3:Tendon**

Total Tendon Area ...

Strand Diameter>**12.7mm (0.5 ')** ; Number of Strands **(12)**

Duct Diameter **(0.063)** ; Relaxation Coefficient>**45**

Curvature Friction Factor **(0.3)** ; Wobble Friction Factor **(0.0066)**

Ultimate Strength **(1900000)** ; Yield Strength **(1600000)**

Load Type>**Post-Tension**

Anchorage Slip>Begin **(0.006)** ; End **(0.006)** ↵

Tendon types determine Relaxation Coefficients. Use 45 for low relaxation tendons. Check off the box to the right if relaxation is to be ignored.

Refer to Table 1 for the remaining tendon data input.

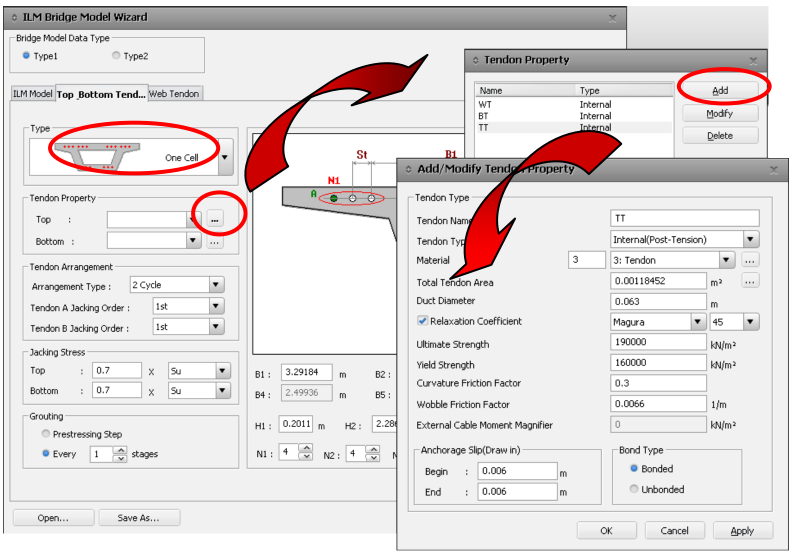


Figure 22 Dialog Box for defining primary and secondary tendon properties

Primary Tendon Input

Specify the primary tendons as per Fig. 23.

[unit : mm]

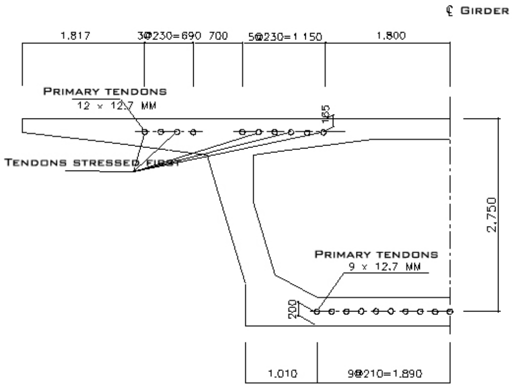


Figure 23 Primary tendon layout

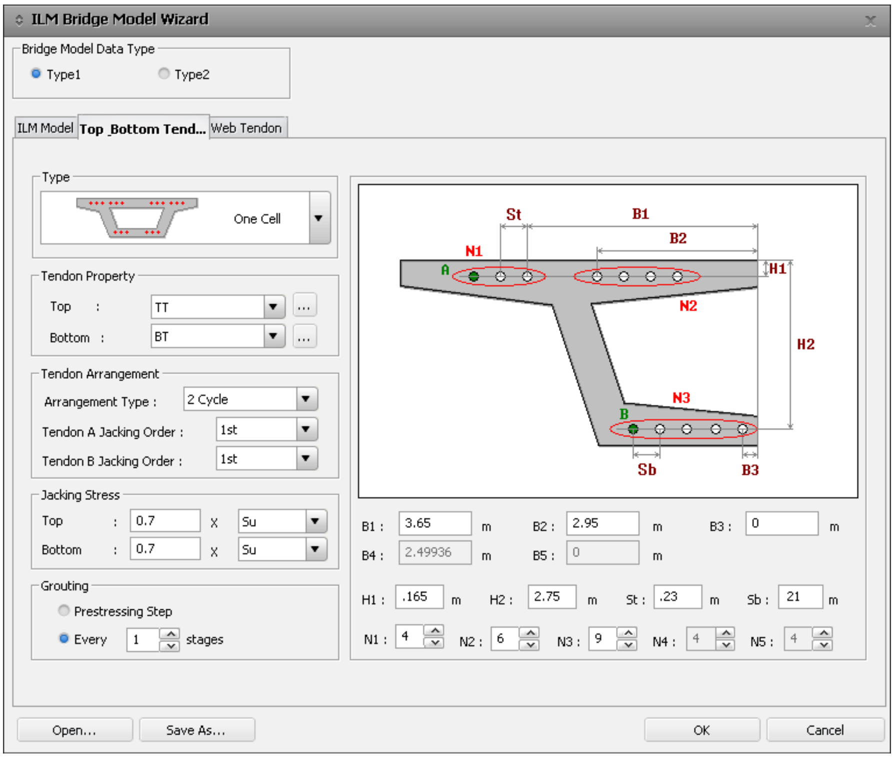


Figure 24 Dialog box for primary tendon input

Select ‘2 cycle’ for placing the primary tendons. The tendons are tensioned in 2 cycles, the group of tendons stressed first and the remaining tendons stressed next. The tendons in the first group start from the most-exterior tendon as shown in Fig. 24. Likewise, the bottom slab tendons are defined as the top slab. A ‘2 cycle’ method refers to tensioning the total tendons in two stages over 2 segments. A 3-cycle method involves dividing the total tendons into three groups and tensioning 3 segments together at each stage. Also, define the tendon jacking orders for Tendon A (Top slab) and Tendon B (bottom slab) in ‘Tendon A Jacking Order’ and ‘Tendon B Jacking Order’ respectively.

Enter “0” for “B3” if an odd number of tendons is placed in the bottom slab, and enter a one half spacing for “B3” if an even number of tendons is placed.

Top & Bottom Tendon tab

Type> 

Tendon Property>Top>**TT** ; Bottom>**BT**

Tendon Arrangement>Arrangement Type>**2 Cycle**

Tendon A Jacking Order>**1st** ; Tendon B Jacking Order>**1st**

Jacking Stress>Top **(0.7)**, **Su** ; Bottom **(0.7)**, **Su**

Grouting>**Every** **(1)**

B1 **(3.65)**, B2**(2.95)**, B3 **(0)**, H1 **(0.165)**, H2**(2.75)**

St **(0.23)**, Sb **(0.21)**, N1 **(4)**, N2 **(6)**, N3 **(9)**

If Prestressing Step is selected in Grouting, the grouting takes place at the stage of tensioning the tendons. If Every is selected, the tendons are tensioned at the number of stages entered, and the grouting takes place at the beginning of the subsequent stage.

Web Tendon

Specify the secondary tendons as per Fig. 25. Two groups of tendons are placed to overlap and alternate over two segments at each stage throughout the entire bridge length. Additional tendons will be placed in the first three and the last three segments of the bridge. Those tendons that are placed in a regular cycle are entered under Web Tendon tab of ILM Model Wizard, and the additional tendons at the beginning and end are entered by the ***Tendon Profile*** function.

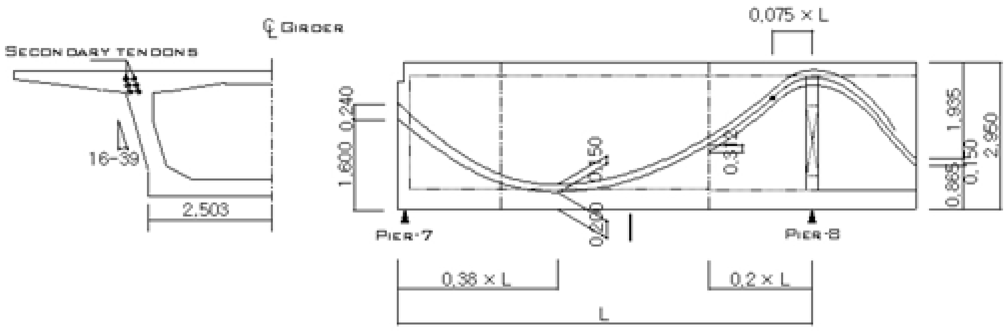
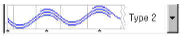


Figure 25 Secondary tendon arrangement diagram (longitudinal vertical cross section) and tendon coordinates

Select a type of type-2 among the standardized types predefined in the program. The secondarytendons can then be defined by simplyfilling in the entry fields.

Web Tendon tab

Type> 

Tendon Property>1st Tendon>**WT**

Jacking Stress **(0.7)**, **Su**

Position>Ey **(2.503)** ; Theta **(16.39)**

Grouting>**Every (1)**

H **(1.6)**, G1 **(0.24)**, G2 **(0.15)**, G3 **(0.312)**, C **(0.2)**

S1 **(0.38)**, S2 **(0.2)**, S3 **(0.075)**, a **(4)**, b **(4)** ↵

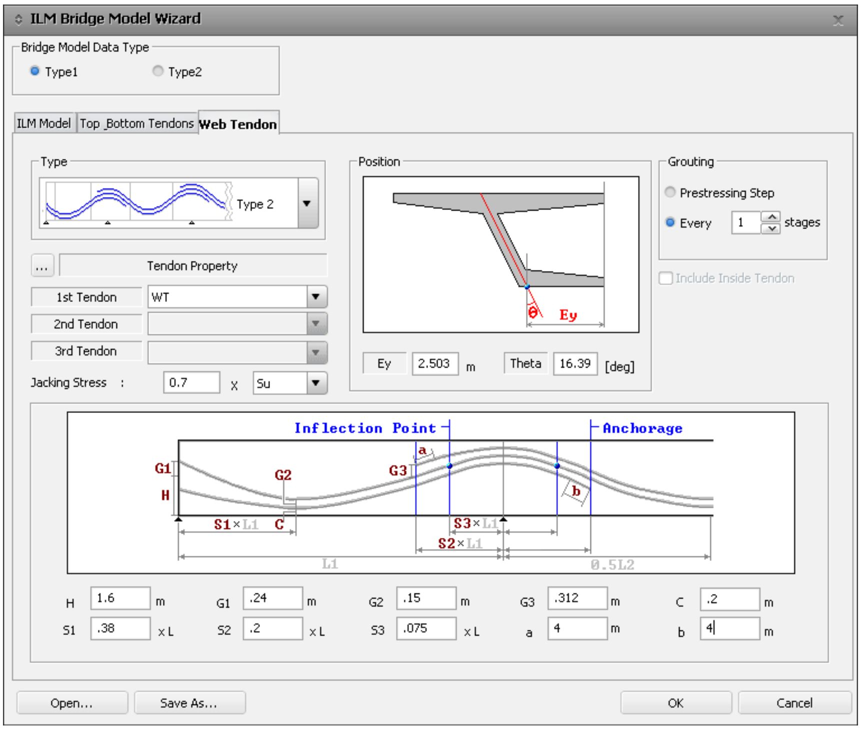


Figure 26 Secondary tendon input dialog box

Once the model is completed using Wizard, use the '***Change Element Dependent Material Property'*** function to enter the value of Notational Size of Member that will be used at the time of computing shrinkage.

Property Tab >Time Dependant Material > ***Change Property***

Select Identity-Elements

Select Type>**Section**

2:Girder Add

Option>**Add/Replace**

Element Dependent Material

Notational Size of Member>**Auto Calculate** ↲

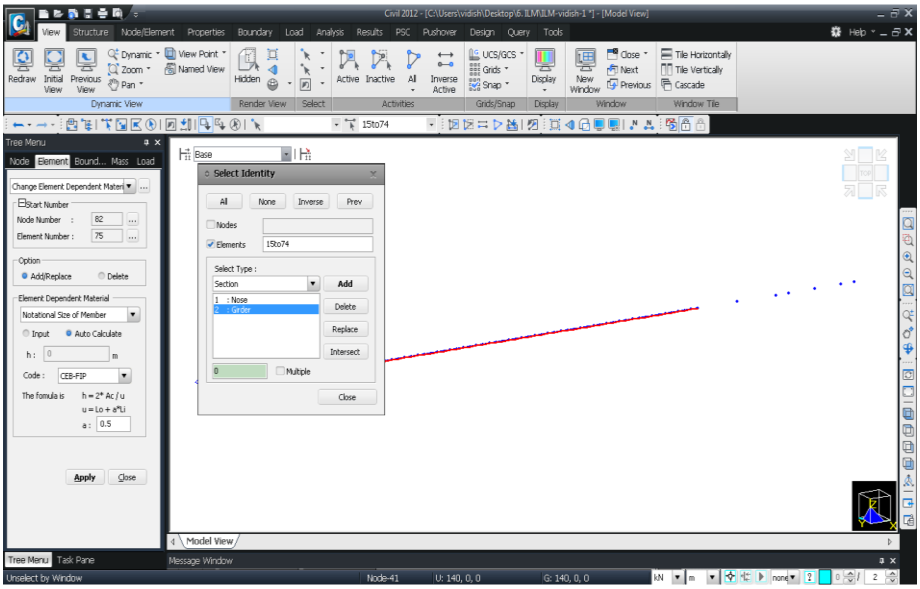


Figure 27 Revision of the value of Notational Size of Member for Girder

Once the modeling is completed, verify the section shape using ***View* / *Hidden*** and the tendon placement in a 3-D view.

Hidden (on)

Display

Boundary>**All** ; **Support** (on), **Elastic Link** (on) ↵

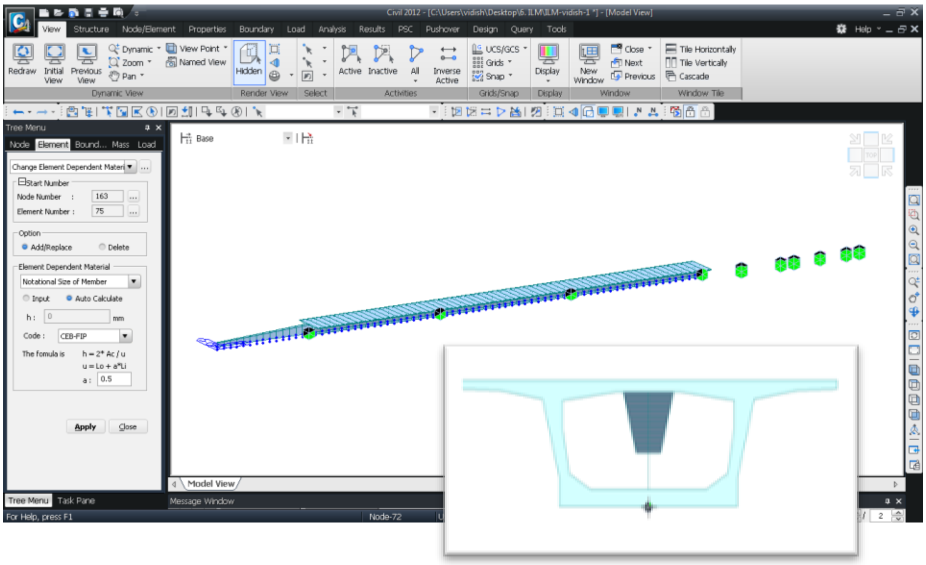


Figure 28 ILM bridge and tendon placement diagrams after completing input

After the bridge model is generated by ***ILM Bridge Model Wizard,*** the Structure Group (Fig. 28) for the nose and each segment, the Boundary Group (Fig. 28) for the Final System and Temporary Group (Fabrication plant), and the Load Group for jacking forces and self-weights are automatically generated.

Define the additional tendons at the starting and ending zones of the bridge using the ***Tendon Profile*** function. Activate the corresponding segments first and input the tendon layout as per Fig. 29.

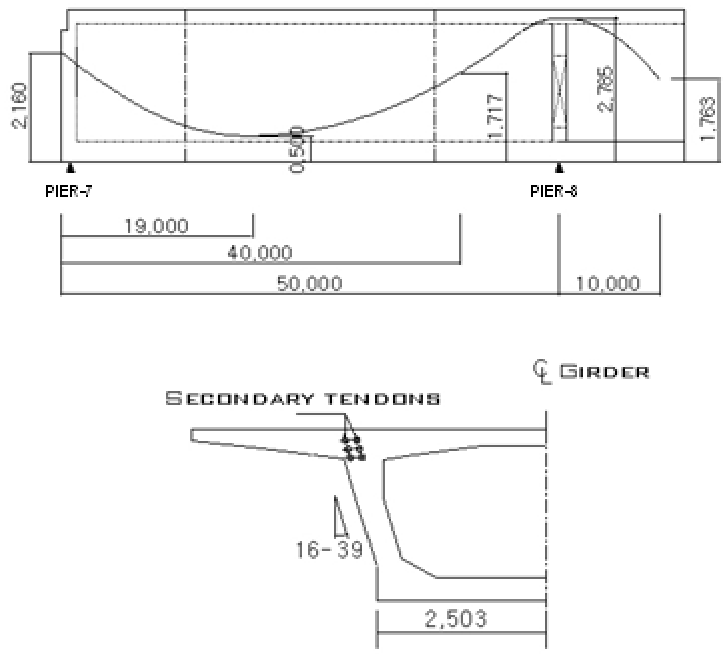


Figure 29 Secondary tendon layout

Table 2 Coordinates of secondary tendons

[unit : m]

Description	**1**	**2**	**3**	**4**	**5**
Distance	0	19	40	50	60
Ez	2.160	0.500	1.717	2.785	1.763

Point Grid (off), ***Point Grid Snap*** (off), ***Line Grid Snap*** (off)

Hidden (off)

Display

Misc>**Tendon Profile Point** (off) ↵

Boundary>**All** ; **Elastic Link** (off) ↵

Active Identity

Group>**Segment1, Segment2, Segment3** Active

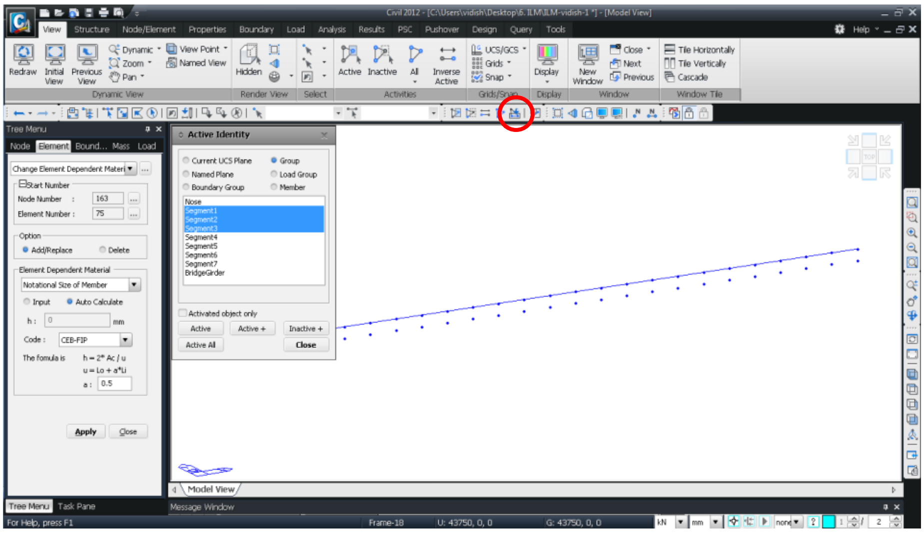


Figure 30 Activation of segments 1, 2 & 3 only

The *Tendon Profile* function automatically places tendons in an optimum curvature that minimizes the rate of curvature changes by incorporating the user defined tendon coordinates. The more coordinates that the user specifies the closer to the actual profile of the tendons will be placed.

The profile of the web tendons is defined with the y-coordinates being “0” as if the tendons were placed on the vertical x-z plane. Only the variation in the vertical (z) direction along the (x) direction of the bridge is identified, and the coordinates are projected later onto the web in the y-direction. The rotational angle of the web is entered to identify the projection plane. The highest and lowest points of the profile are “fixed” to specify fixed curvature vectors. In this case, the vectors are fixed to the horizontal-tangential direction in the direction of the bridge.

Element Number (on)

Load Tab/**Temp./Prestress** / Prestress Loads / ***Tendon Profile*** Add

Tendon Name **(WT-End1)** **;** Tendon Property>**WT**

Select Window (Elements : **15 ~ 38**) or Assigned Elements **(15to38)**

Straight Length of Tendon>Begin **(4)** : End **(4)** : input Type>**3D** : Curve Type>**Spline**:

Reference Axis>**Straight** ; Profile Insertion Point **(35, 2.503, 0)**

x Axis Direction>**X** ; x Axis Rot. Angle **(-16.39)**, **Projection** (on)

Grad Rot. Angle>**Y**, **(0)**

1>x **(0)** ; y **(0)** ; z **(2.16)**

2>x **(19)** ; y **(0)** ; z **(0.5)** ; **fix** (on)

3>x **(40)** ; y **(0)** ; z **(1.717)**

4>x **(50)** ; y **(0)** ; z **(2.785)** ; **fix** (on)

5>x **(60)** ; y **(0)** ; z **(1.763)** ↵

Grad. Rot. Angle represents the angle of the bridge expressed in terms of rotation about the axis specified. Y & 0 represent that no vertical slope exists.

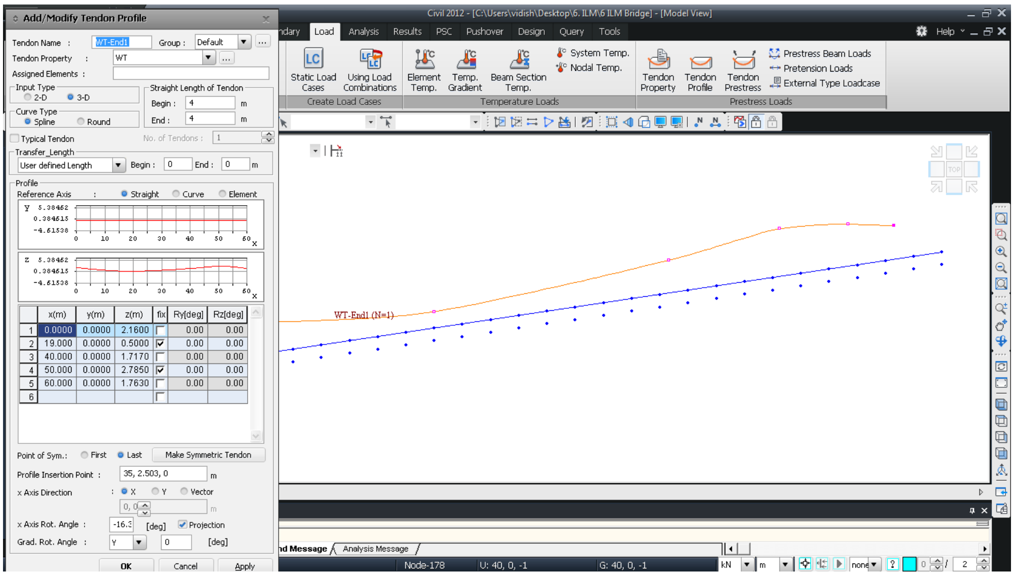


Figure 31 Definition of additional tendons in left web at left end of the bridge

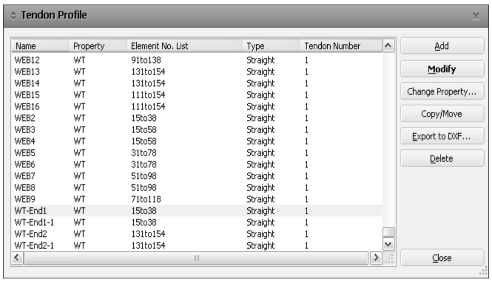
Copying the tendon WT-End1 and changing the Profile Insertion Point and web rotation angle can create the tendons in the right web at the left end of the bridge.

Load>Temp./Prestress/Prestress Loads / Tendon Profile > Name> **WT-End1** ; Postfix **(-1)** Copy

WT-End1-1 Modify

Profile Insertion Point **(35, -2.503, 0)**

x Axis Direction>**X** ; x Axis Rot. Angle **(16.39)**, **Projection** (on) ↵



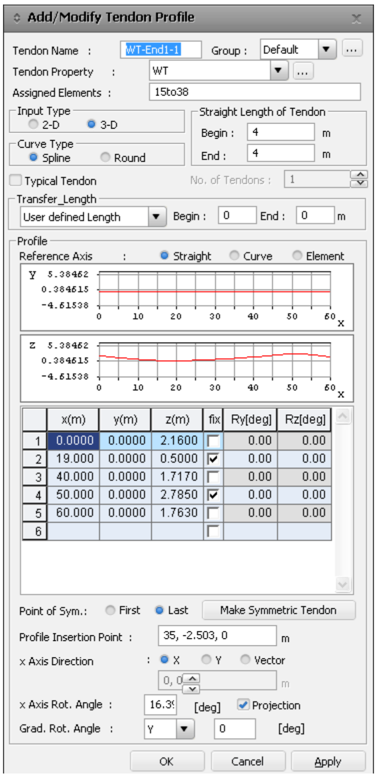


Fig. 32 ***Definition of additional tendons in right web at left end of the bridge***

Active Identity

Group>**Segment5, Segment6, Segment7** Active

Activate only the segments 5, 6 & 7 for defining the web tendons at the right end of the bridge.

Similarly, copying the tendon WT-End1 and changing the Profile Insertion Point, x Axis direction and web rotation angle can create the additional tendons in the left web at the right end of the bridge.

Tendon Profile>Name>**WT-End1** : Copy

WT-End1-Copy Modify

Unselect All, ***Select Window*** (Elements : **51 ~ 74**)

Tendon Name **(WT-End2)**

Profile Insertion Point **(185, 2.503, 0)**

x Axis Direction>**Vector (-15, 0)**

x Axis Rot. Angle **(16.39)**, **Projection** (on) ↵

Tendon Profile>Name>**WT-End2** ; Postfix **(-1)** Copy

WT-End2 Modify

Profile Insertion Point **(185, -2.503, 0)**

x Axis Rot. Angle **(-16.39)**, **Projection** (on) ↵

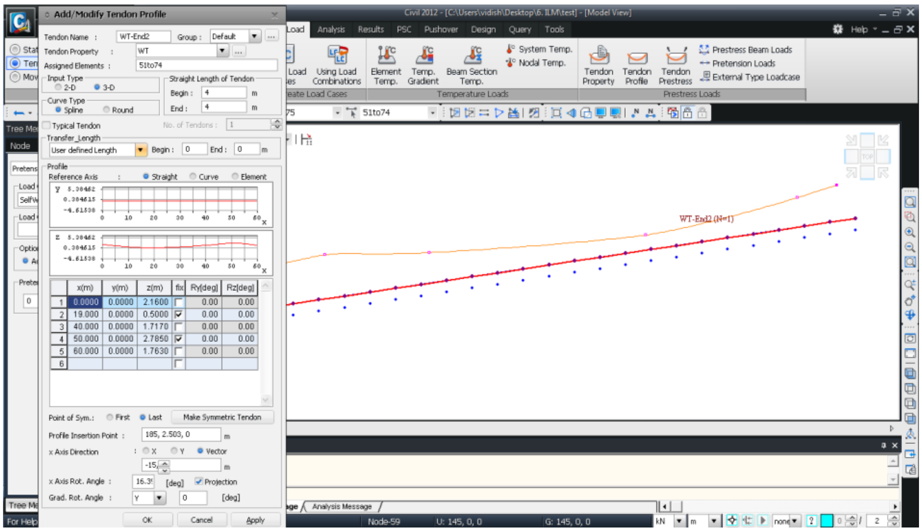


Figure 33 Definition of additional tendons in left web at right end of the bridge

When ILM Bridge Model Wizard is executed, "SelfWeight" and "Prestress" load conditions are automatically generated and entered.

Assign the jacking forces for the additional tendons placed at both ends of the bridge. Tension the tendons at the left end of the bridge first, and specify the tension forces at both ends of the tendons.

Load / Temp./Prestress / Prestress Loads / ***Tendon Prestress***

Load Case Name>**Prestress**

Load Group Name>**Web-Prestress**

Selected Tendon for Loading>Tendon>

WT-End1, WT-End1-1 > Selected

Stress Value>**Stress** ; 1st Jacking>**Begin** (left-end of bridge)

Begin **(1330000)** ; End **(1330000)**

Grouting>After **(1)** Add

Selected Tendon for Loading>Tendon>

WT-End2, WT-End2-1 > Selected

WT-End1, WT-End1-1 < Tendon

Stress Value>**Stress** ; 1st Jacking>**End** (right-end of bridge)

Begin **(1330000)** ; End **(1330000)** Add

Grouting>After **(1)** Add

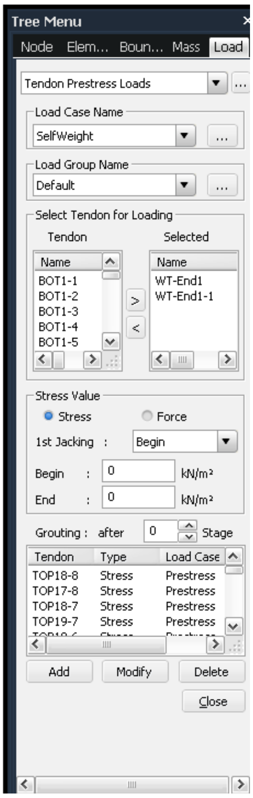


Fig. 34 Jacking force input of the additional tendon at the end of a

DEFINITION OF ILM CONSTRUCTION STAGE

MIDAS/Civil has two types of work modes(Base Stage Mode and Construction Stage Mode) when Construction Stage is defined for the construction stage analysis.

Base Stage mode denotes the stage where all structural structural modeling, and the input of loading conditions and the input for boundary conditions are possible, and the analysis of this stage is not actually performed. Construction Stage indicates the model condition that the structural analysis is performed. In Construction Stage mode, the structural model can not be revised, and no data information can be revised and deleted except the boundary condition and the loads.

Construction Stage consists of 'Activation and Deactivation' commands of Structure Group, Boundary Group and Load Group. In Construction Stage Mode, the boundary condition and the load condition can be revised and deleted, which are included in the activated Boundary Group and Load Group.

The girder self-weight and pre-stress load of construction stage in the ILM bridge are automatically input by ILM Bridge Model Wizard. The additional load and boundary condition can be revised or be input in 'Base Stage' after activation or deactivation of the concerned Construction Stage.

ILM Bridge Stage Wizard

In the construction stage analysis, the boundary conditions are changed at every construction stage because the segments are cured at the manufacturing plant and are launched in a particular order. MIDAS / Civil provides 'ILM Bridge Stage Wizard' function which can define the construction stage easily otherwise it is complicated to define the construction stages considering all the boundary conditions and the changes of the elements. If the Boundary Group of the manufacturing plant, the Boundary Group of the final system, and the launching distance of each construction stage are input, the ILM Bridge Stage Wizard function automatically defines the construction stages considering the change in boundary condition.

The data is inputted in 'ILM Bridge Stage Wizard' is as follows.

1. Select the 'Boundary Group', The boundary condition of a pier and an abutment which will be applied in the bridge model of final system.(Boundary Group is known as 'Final' is automatically generated in ILM Bridge Wizard function)
2. Input the total launching distance and direction in 'Launching Direction'.
   - **I.** Set the 'Reference Node' as the end of the nose.
   - **II.** Input as Start Node where the nose is resting at the initial construction stage.
   - **III.** If End Node is set at an arbitrary node in the left side of the Start Node, the segments are launched towards the end node direction starting from the start node.
3. If the launched location at the time of modeling and the location of piers and abutments after completion do not coincided, input General Tolerance which will consider the two locations as equivalent.
4. Input of launching information per each segment
   - **I.** Activate/deactivate all boundary conditions automatically at every construction stage including the boundary condition of the starting and ending construction stage using the final boundary condition of the completed bridge and the previously entered field boundary condition known as 'Field'.
   - **II.** The boundary conditions are automatically input at every construction stage if the boundary conditions of the field and the temporary piers at the initial construction stage are activated.
   - **III.** As the boundary conditions of the field and the temporary piers are

not needed at the final construction stage, which the jacking of the second tendon is completed, deactivate the Field boundary condition at the final construction stage.

Select the boundary condition of the final bridge, and input the reference launching node and the launching direction. Input the reference node at the end of nose, Start Node at the start point of launching which is the right end of the bridge, and End Node which is an arbitrary node at the left side of Start Node.

Element Number (off), ***Active All***

Display

Misc>**Tendon Profile** (off) ↵

Structure/ Wizard / ***ILM Bridge / ILM Bridge Stage***

Boundaries of Final Structure System>Boundary Group>**Final**

Launching Direction>Reference Node **(1)**

Start Node **(75)** ; End Node **(73)**

Generation Tolerance>**T (0.01)**

Input allowabletolerance to account for the difference between location of pier and segment boundary due to modeling error. Detailed explanation can be found ILM Bridge Stage Wizard in on-line manual.

Input radius in the case of curved bridge to be launched as curved bridge.

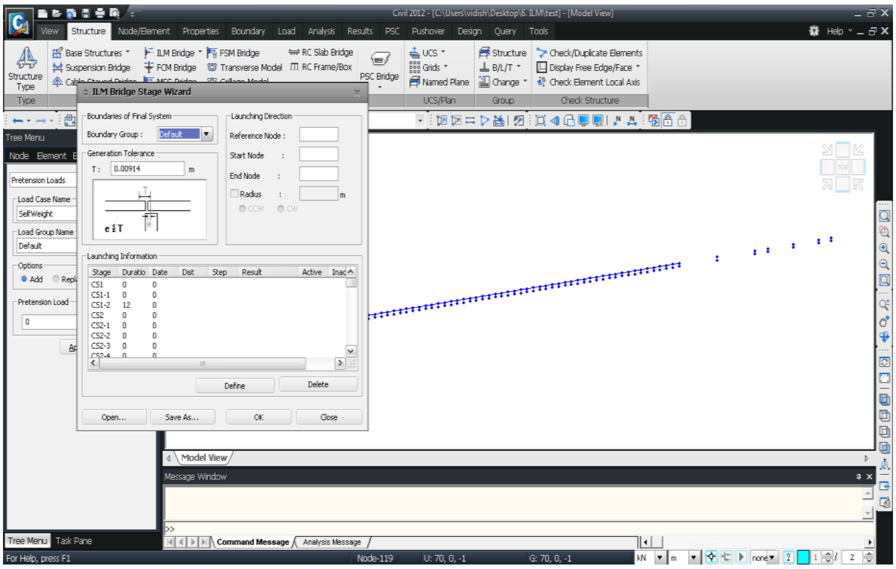


Fig. 35 Definition of construction stage using ILM Bridge

Input information about launching that will be used in construction stage definition. Input the launching length as 5m because the length for beginning/ending parts and inner parts are not identical. For the 1st segment, construction stages can be defined automatically as 2.5m lengths by input 2 for step because the total length is 12.5m. If the Field boundary conditions are activated for the 1st segment during the incremental launching information is defined, the defined boundary conditions are maintained until they are deactivated.

Analysis results for each construction stage can be saved if Save Stage Result is selected

Structure Tab > ILM Bridge Stage>
Launching Information>Stage>**CS1** Define
Launching Information>Distance **(10)** ; Step **(2)**
Save Stage Result (on) ; **Save Step Result** (on)
Boundary Group for Temporary Support>Boundary Group>**Temporary**
Active Group Add ↵

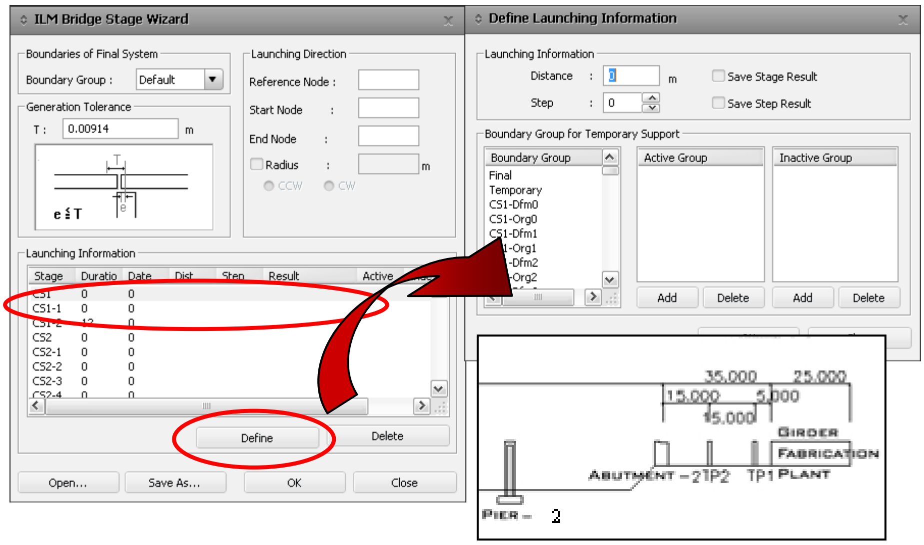


Fig. 36. Input Launching information for the 1st segment

Construction stages between CS2 to CS6 are defined at the same time because segment lengths are identical as 25m. Segment 7 is last segment and hence the end of segment should be launched at the location of abutment. Therefore the segment should be launched at 50m that is summation of segment length 12.5m and the distance between manufacturing yard and abutment (2@15m+5m). CS8 is the construction stage during which the 2nd tendon is acked. Boundary conditions in CS8 are the same as those of the other construction stages. In CS8 segment 7 is totally launched and only 2nd tendon is activated. Therefore the launching information is not given and only the results are saved. Deactivate the Field Boundary Group that represents temporary piers used during construction and manufacturing yard. CS9 represents the construction stage in which web tendons are grouted.

Launching Information>Stage>**CS2~CS6** Define
Launching Information>Distance **(25)** ; Step **(5)**
    Save Stage Result (on) ; **Save Step Result** (on) ↵
Launching Information>Stage>**CS7** Define
Launching Information>Distance **(50)** ; Step **(10)**
    Save Stage Result (on) ; **Save Step Result** (on) ↵
Launching Information>Stage>**CS8** Define
    Save Stage Result (on) ; **Save Step Result** (on)
    Boundary Group for Temporary Support>Boundary Group>**Temporary**
        Deactive Group Add ↵
Launching Information>Stage>**CS9** Define
    Save Stage Result (on) ; **Save Step Result** (on) ↵

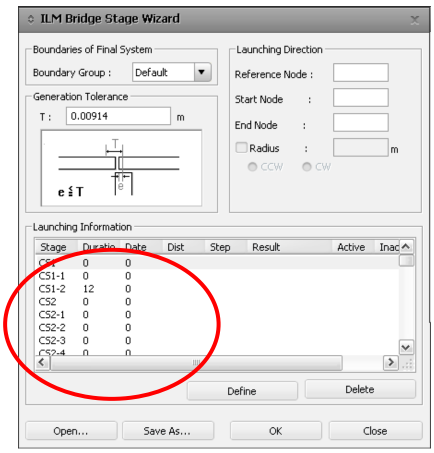


Figure 37 Input Launching Information for each Segment

After completion of construction stage definition by ILM Bridge Stage Wizard function, select construction stages that are to be modified and modify them. The hinge condition (Field Group) defined for the end of bridge is effective for all construction stages. After launching is completed, 2nd tendon is jacked and superimposed dead loads are applied in the construction stage CS8. In this construction stage the boundary conditions should be modified to simulate construction completed stage. It is convenient to modify boundary condition and load condition at each construction stage.

Hidden (off)

Stage>**CS8** (Fig 38 ①)

Boundary>***Supports>*** ***Define Supports***

Boundary Group Name>**CS7-Dfm9**

Select Window (Node : **75**)

Options>**Add** ; Support Type>**Dx** (on) ↵

In CS9, input the boundary condition at joint 35 similar to the procedures for CS8.

Modified results can be reviewed using Zoom Window.

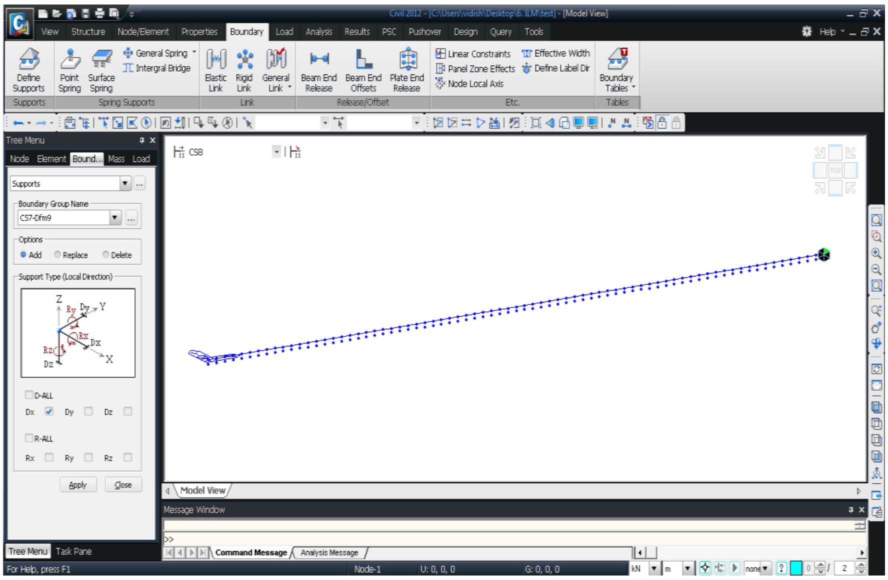


Figure 38 Modification of Boundary Conditions at CS8

After completion of analysis model input (element, boundary condition and load) and definition of construction stage, the tendon arrangement and boundary conditions can be easily reviewed using Stage Toolbar.

Hidden (on)
Display
Misc>**Tendon Profile** (on) ↵
Stage>**CS1** (Fig 39)

The construction stages can be easily changed using arrow key in key board.

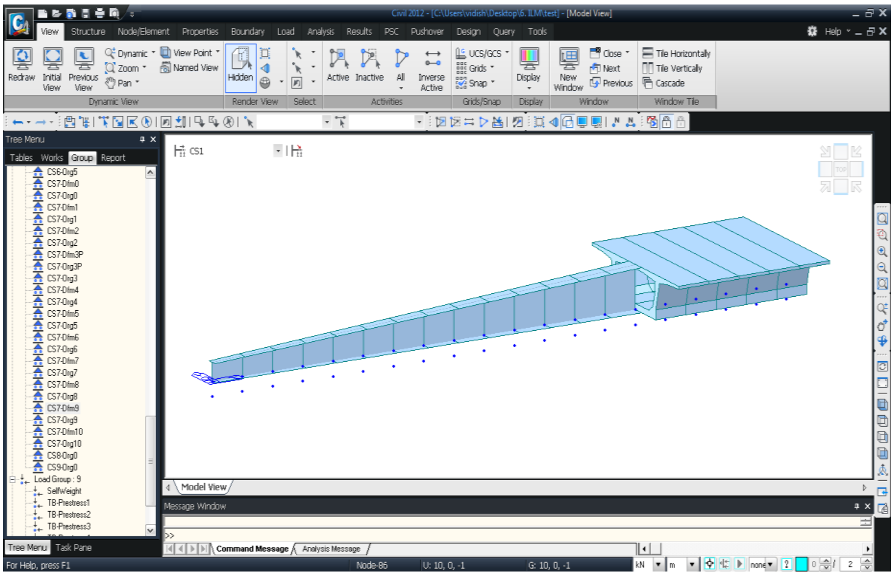


Figure 39 Element and Boundary Condition Review at Construction Stage CS1

Input Diaphragm and Superimposed Dead Load

After completion of construction stage definition, input the self-weight of diaphragm as beam load at the construction stage at which the segment that contains support diaphragm is activated. At stage CS8, input 2nd dead load as beam load.

- Superimposed dead load (2nd dead load)
- Input beam loads 31.5 kN/m for all bridge length at CS8.

Self-weight of diaphragm

Assume diaphragm makes little contribution to the stiffness of girder but only acts as load.
Therefore diaphragm is defined as the same section as other general sections and additional load is applied. In this tutorial, apply beam load at the construction stage at which support segment is activated.

I. INose connection part: 763.0 kN
II. Pier parts & Abutment part at bridge end: 516.1 kN

The self-weight of diaphragm is assigned as a uniform load along the thickness of diap-hragm which is 2m long.

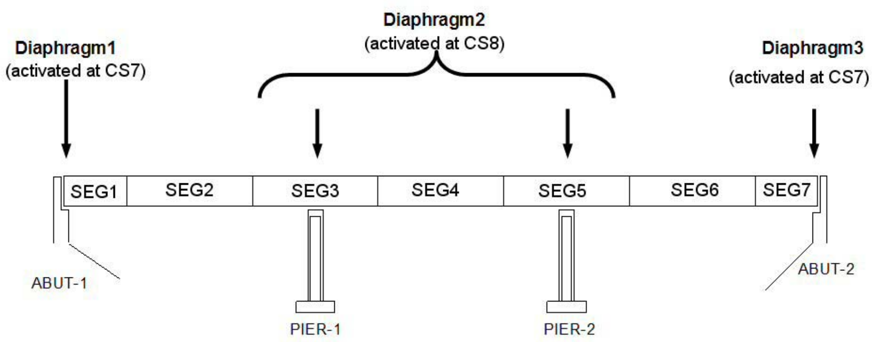


Figure 40 Input Diagram for Diaphragm Load

Table 3 Input Location for Diaphragm self-weight

Construction stage	Load Location(m) (from Beginning point of Bridge	Remark	**Load Group**
CS1	0	Nose connection part	Diaphragm1
CS8	49 ~	Pier part	Diaphragm2
CS8	99 ~	Pier part	Diaphragm2
CS7	148 ~	Abut part at bridge end	Diaphragm3

Define Load Group (2nd Dead) and Load Group (Diaphragm1, 2, 3) to input Super- imposed dead load and diaphragm load, respectively. Change the stage to Base Stage by Stage toolbar because addition of Groups is possible only at Base Stage. At Base Stage, analysis is never performed but boundary conditions, load conditions and elements for all construction stages are displayed. And so it is possible to modify, add and delete for these properties for all construction stages.

Display

Misc>**Tendon Profile** (off) ↵

Stage>**Base**

Group>**Load Group** **New…**

Name **(Diaphragm)** ; Suffix **(1to3)** Add

Name **(2nd Dead)** Add

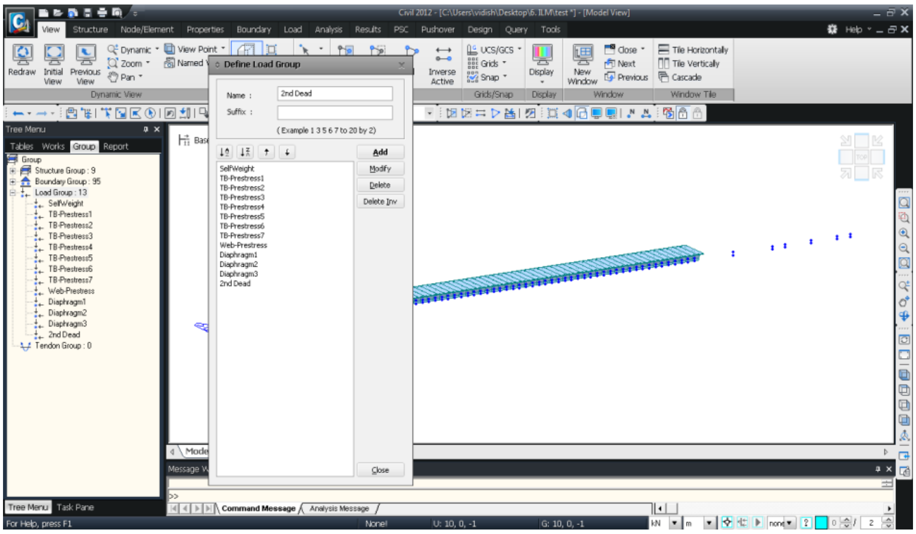


Figure 41 Generate Load Group

Activate load group Diaphragm1 and Diaphragm3 to CS1 and CS7, respectively.

Top Left of model view > *Define Construction Stage*

Name>**CS1** [Modify/Show]

Load tab

Group List>**Diaphragm1**

Activation>Active Day>**First** ; **Group List** [Add] ↵

Name>**CS7** [Modify/Show]

Load tab

Group List>**Diaphragm3**

Activation>Active Day>**First** ; **Group List** [Add] ↵

If the time step for activation is defined at First, load is applied from the 1st date during the duration of specific construction stage.

Among automatically generated construction stages, some construction stages end with "p". At these construction stages, the end of nose is located at piers. But it is considered as cantilever beam because these are the construction stages to confirm the maximum negative reactions.

The names for element group, boundary group and load group that are generated by Bridge Wizard can be referred "Define Structure(Boundary, Load) Group at on-line manual.

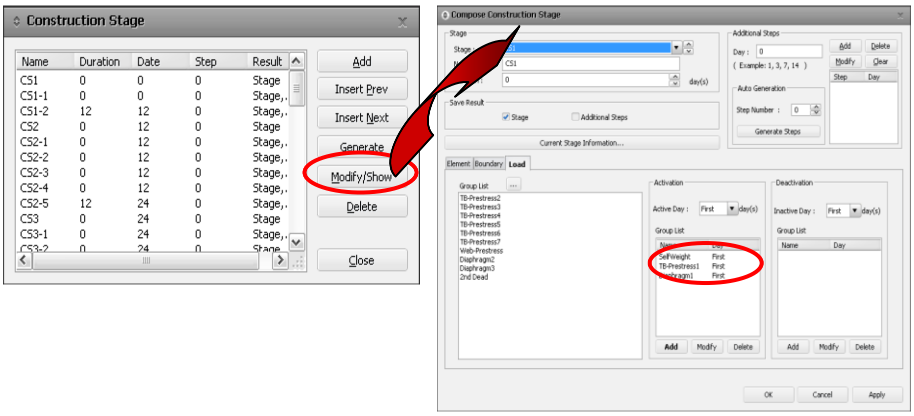


Figure 42 Addition of Diaphragm Self-weight

At CS8, deactivate nose because in this stage diaphragm self-weights and super-imposed dead loads are applied and launching is finished. At CS9, grouting for web tendon is performed and creep, shrinkage and long-term loss for jacking forces are considered until 10,000 days.

Top Left of model view > ***Define Construction Stage***

Name>**CS8** Modify/Show

Element tab

Group List>**Nose**

Deactivation>Element Force Redistribution **(100)** ; Add ↵

Load tab

Group List>**Diaphragm2, 2nd Dead**

Activation>Active Day>**First** ; **Group List** Add ↵

Name>**CS9** Modify/Show

Stage>Duration **(10000)**, Save Result>**Additional Step** ↵

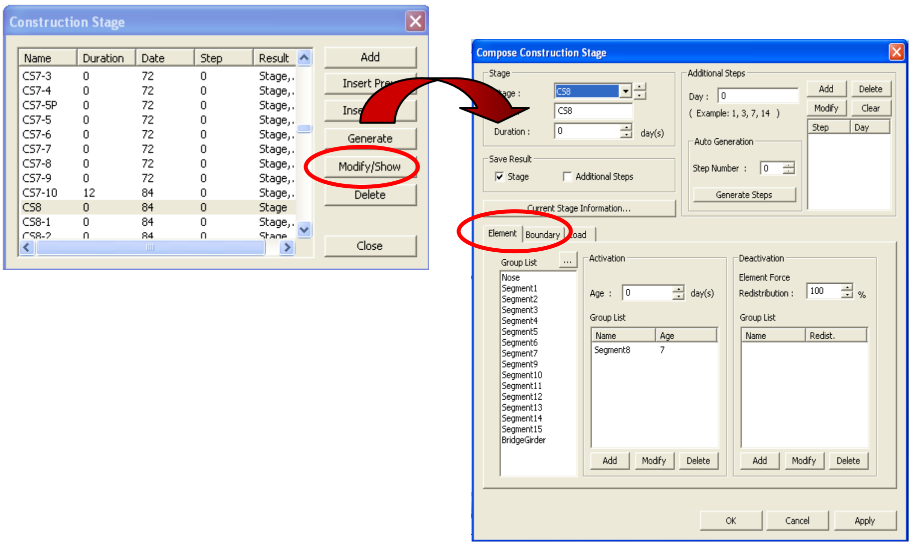


Figure 43 Activation of Load Group and Deactivation of Element Group for CS8

Input load at each construction stages because load group is activated for each Construction stages. First, input self-weight of nose connection part diaphragm at CS1.

Stage>**CS1**

Front View

Load Tab>***Line Beam Loads***

Load Case Name>**SelfWeight** ; Load Group Name>**Diaphragm1**

Options>**Add** ; Load Type>**Uniform Loads**

Direction>**Global Z** ; Projection>**No**

Value>**Absolute** ; x1 **(0)**, x2 **(2)**, w **(-763/2)**

Nodes for Loading Line **(15, 16)**

Because the beam element is modeled as 2.5m spacing, select Absolute at Value, and assign 2m for real load input distance.

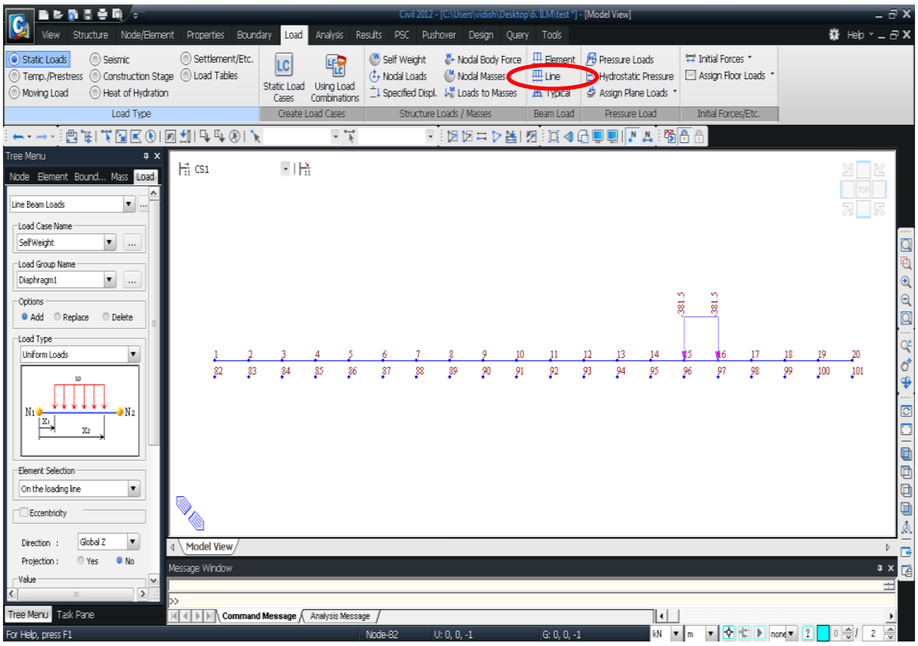


Figure 44 Input Diaphragm weight at Nose Connection Part

Input bridge end part diaphragm load at CS7.

Stage>**CS7**

Display

Load>**All** ; **Beam Load** (on) ↵

Load Tab> Beam Loads Section >***Line***

Load Case Name>**SelfWeight** ; Load Group Name>**Diaphragm3**

Options>**Add** ; Load Type>**Uniform Loads**

Direction>**Global Z** ; Projection>**No**

Value>**Absolute** ; x1 **(0)**, x2 **(2)**, w **(-516.1/2)**

Nodes for Loading Line **(75, 74)**

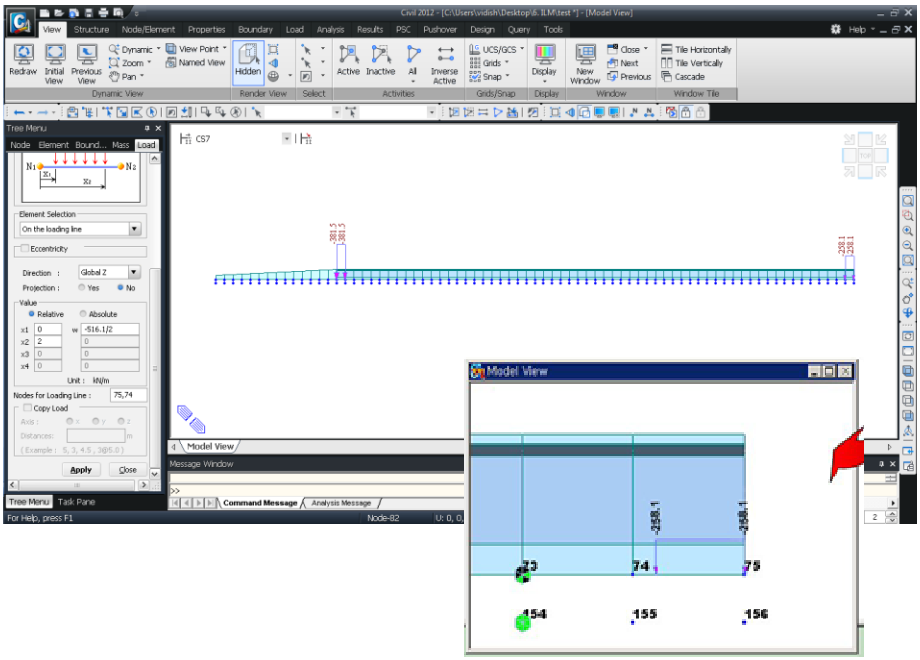


Figure 45 Input Self-weight of Diaphragm at bridge end

Input diaphragm self-weight at CS8 as follows. Pier diaphragm is manufactured as 1m length to both bridge longitudinal directions from the center of pier and defined as 2 elements including boundary conditions. Because same loads are applied to all piers, loads can be copied and then applied.

Stage>**CS8**

Load Tab> Beam Loads Section >***Line***

Load Case Name>**SelfWeight** ; Load Group Name>**Diaphragm2**

Options>**Add** ; Load Type>**Uniform Loads**

Direction>**Global Z** ; Projection>**No**

Value>**Absolute** ; x1 **(1.5)**, x2 **(3.5)**, w **(-516.1/2)**

Copy Load (on)

Axis>**x** ; Distance **(50)**

Nodes for Loading Line **(34, 36)**

Copy load and input because the pier spacing is identical as 50m.

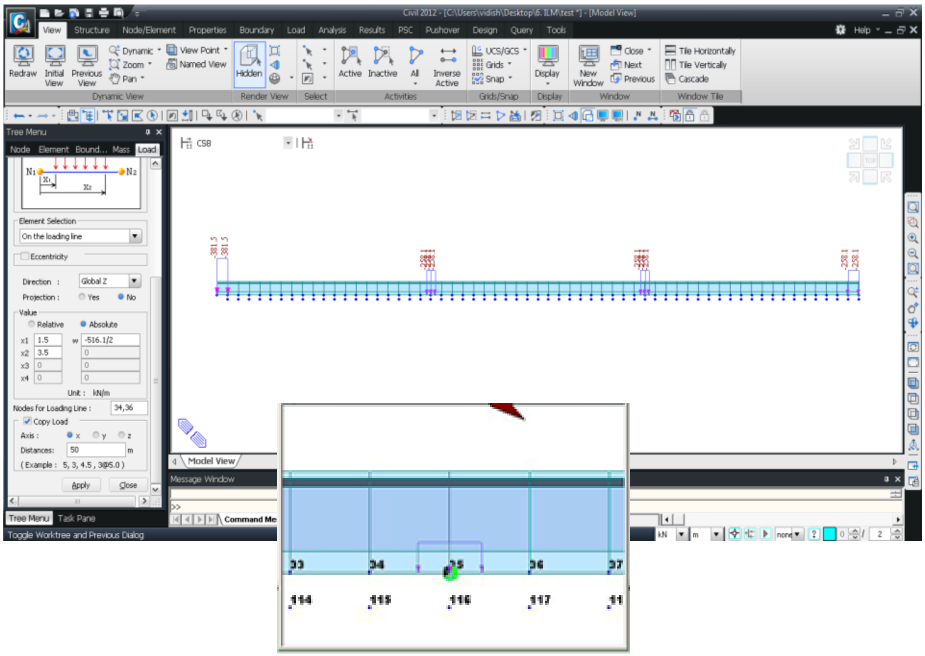


Figure 46 Input Diaphragm self-weight at CS8

Input superimposed dead load at CS8.

Hidden (off)

Load Tab > Beam Loads Section > ***Element***

Select All

Load Case Name>**SelfWeight** ; Load Group Name>**2nd Dead**

Options>**Add** ; Load Type>**Uniform Loads**

Direction>**Global Z** ; Projection>**No**

Value>**Relative** ; x1 **(0)**, x2 **(1)**, w **(-31.5)** ↲

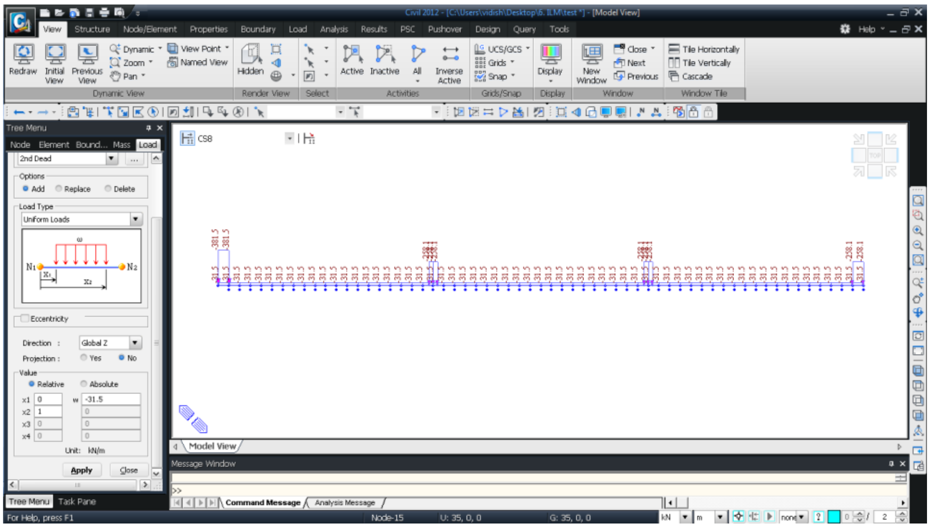


Figure 47 Input Superimposed Dead Load at CS8

Stage>**Base**

Analysis Tab >***Construction Stage***

Final Stage>**Last Stage**

Analysis Option>**Include Time Dependent Effect** (on)

Time Dependent Effect

Creep Shrinkage (on) ; Type>**Creep & Shrinkage**

Convergence for Creep Iteration

Number of Iteration **(5)** ; Tolerance **(0.01)**

Internal Time Step for Creep **(1)**

Auto Time Step Generation for Large Time Gap (on)

Tendon Tension Loss Effect (Creep & Shrinkage) (on)

Variation of Comp. Strength (on)

Tendon Tension Loss Effect (Elastic Shortening) (on) ↵

Define convergence condition for the creep and shrinkage analysis.

The long term load effect can be considered by generating time steps internally for construction stages that have adequate duration by checking on “Auto Time Step for Generation for Large Time Gap”.

This is the item to consider tendon prestress losses by elastic shortening, creep and shrinkage.

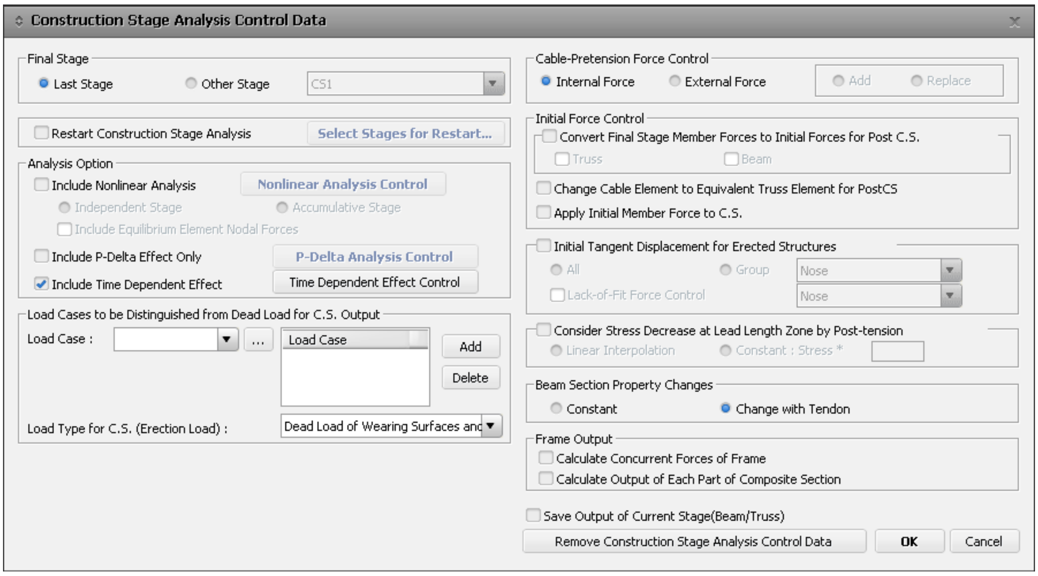


Figure 48 Dialog Box for Construction Stage Analysis Control Data

Perform Structural Analysis

All input for construction stage analysis of ILM bridge is finished, perform structural analysis.

Analysis> ***Perform Analysis***

Review Analysis Result

There are two methods for reviewing analysis results for construction stage analysis results. The one is to review accumulated stresses and displacements until specified construction stage and the other is to review for a specified construction stage. In MIDAS / Civil the construction stage analysis results can be reviewed using graph and table by either of above method.

Stress review by graph

Stress graph at CS9 is reviewed.

If Axial, Bending My and Bending Mz are all selected, stresses at each extreme outer fiber location can be reviewed.

In ILM wizard, Structure Group, that is required to review section stresses, are automatically generated.

If "Draw Allowance Stress Line" is checked on, the allowable stressed for compression and tension are plotted by dashed lines on stress graph.

Stage>**CS9**

Results Tab > ***Bridge Girder Diagram***

Load Cases/Combinations>**CS:Summation ;** Step>**Last Step**

Components>**Combined** (on) ; **2(+y, +z), 3(+y, -z)**

Bridge Girder Elem. Group>**Bridge Girder**

Draw Allowable Stress Line (on)>Comp. **(16000)** ; Tens. **(3200)**

General Option > Current Stage/Step (on) ↵

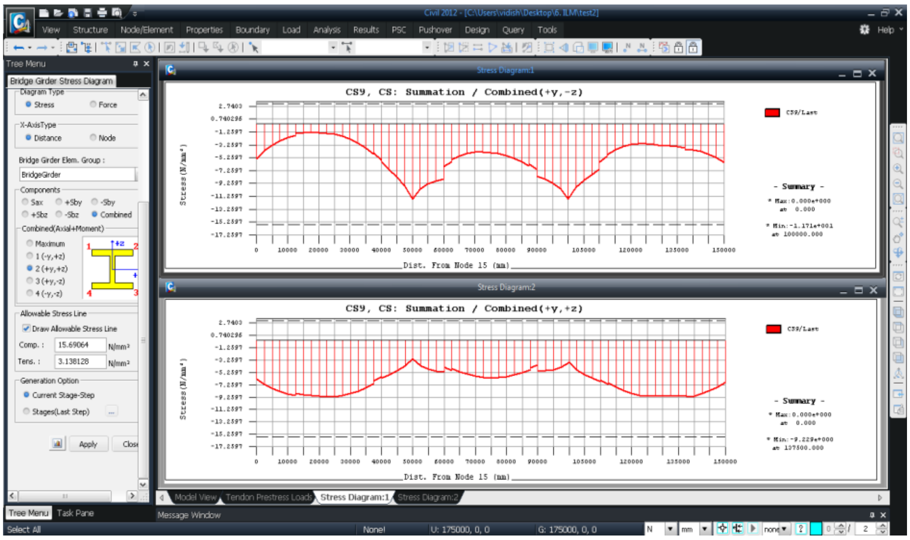


Figure 49 Stress Graph at Top/Bottom fiber at CS9

For detailed review for specific range of graph, the graph can be magnified by locating mouse on the graph and dragging. After reviewing, it can be returned to its original status by selecting “Zoom Out All”, clicking mouse right button.

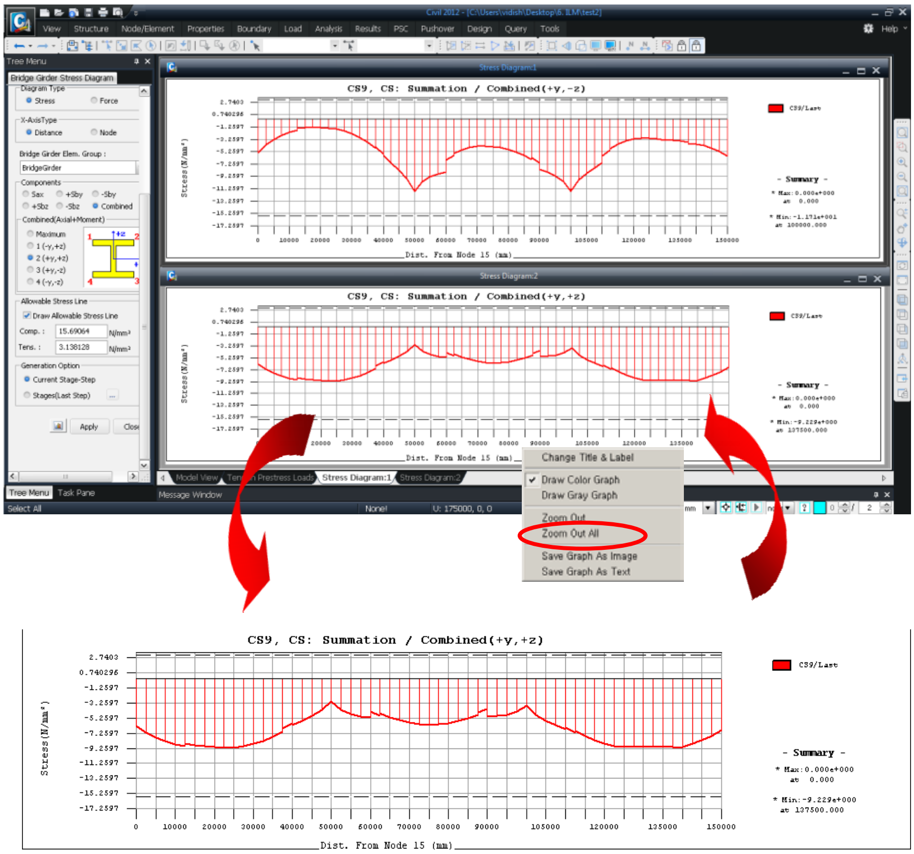


Figure 50 Stress Graph Magnification

The "Min/Max" stage produces the maximum and minimum values out of the analysis results of all the construction stages. Min/Max Stage is used to review maximum and minimum stresses during construction along the girder.

Stage>**Min/Max**

Results Tab > ***Bridge Girder Diagram***

Load Cases/Combinations>**CSmax:Summation, CSmin:Summation**

Diagram Type>**Stress**; X-Axis Type>**Distance**

Components> **Combined** (on) ; **2(+y, +z), 3(+y, -z)**

Bridge Girder Elem. Group>**Bridge Girder**

Draw Allowable Stress Line (on)>Comp. (**16000**) ; Tens. (**3200**) ↵

Figure 51 Maximum / Minimum Stress Graph

Min/Max Stage is used to review maximum and minimum moments during construction alongthe girder.

Stage>**Min/Max**

Results Tab > ***Bridge Girder Diagram***

Load Cases/Combinations>**CSmax:Summation, CSmin:Summation**

Diagram Type>**Force** ; X-Axis Type>**Node**

Bridge Girder Elem. Group>**Bridge Girder**

Components>**My**↵

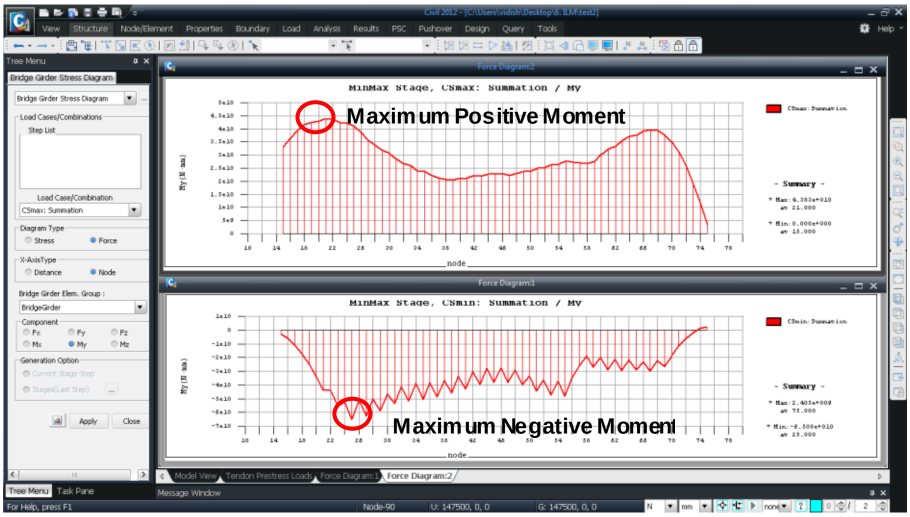


Figure 52 Maximum / Minimum Moment Graph

The maximum and minimum moments diagram can be also displayed using "BeamDiagrams".

Stage>**Min/Max**
Click on Model View Tab (Figure 53)
Results Tab > Forces / ***Beam Diagrams***
Load Cases/Combination> **CSmin:Summation, CSmax:Summation**
Components> **My**
Output Options> **5 Points ; Line Fill**
Type of Display> **Contour** (on) ↵

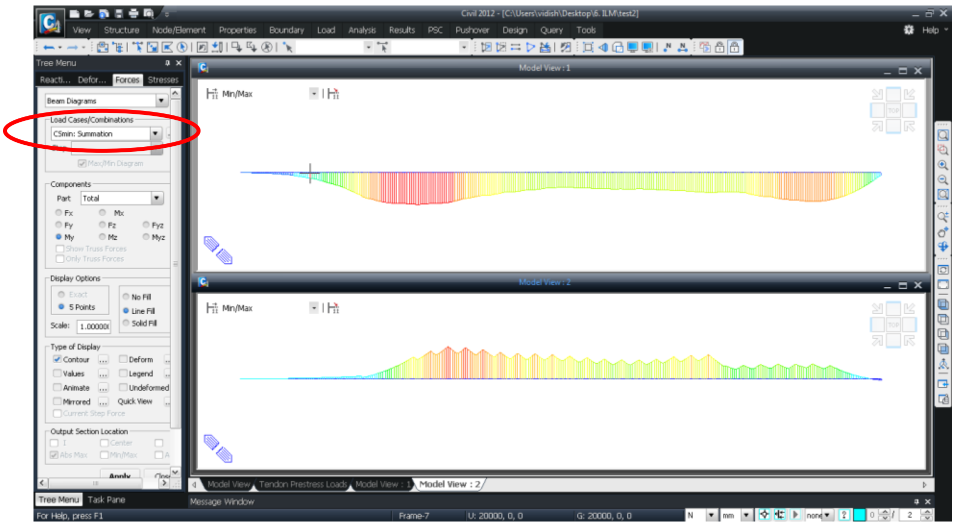


Figure 53 Maximum / Minimum Moment Diagram

Moment history graph by stages at a specific point, for example, I end, element 67 and I end, element 25 at which maximum positive and negative moment occur, respectively, can be displayed.

Stage>**CS9**
Results Tab > ***Stage/Step History Graph***
Define Function>**Beam Force/Stress** Add New Function
Beam Force/Stress>Name (**67-Moment**) ; Element No. (**67**) ; **Force Point>I-Node** ; Components>**Moment-y** ↵

Define Function>**Beam Force/Stress** Add New Function
Beam Force/Stress>Name (**25-Moment**) ; Element No. (**25**) ; **Force Point>I-Node** ; Components>**Moment-y**
Mode>**Multi Func.** ; Step Options>**All Step**

Check Functions To Plot>**25-Moment**, **67-Moment** (on)
Load Cases/Combinations>**Summation** ; Graph Title (**Moment**) Graph

---

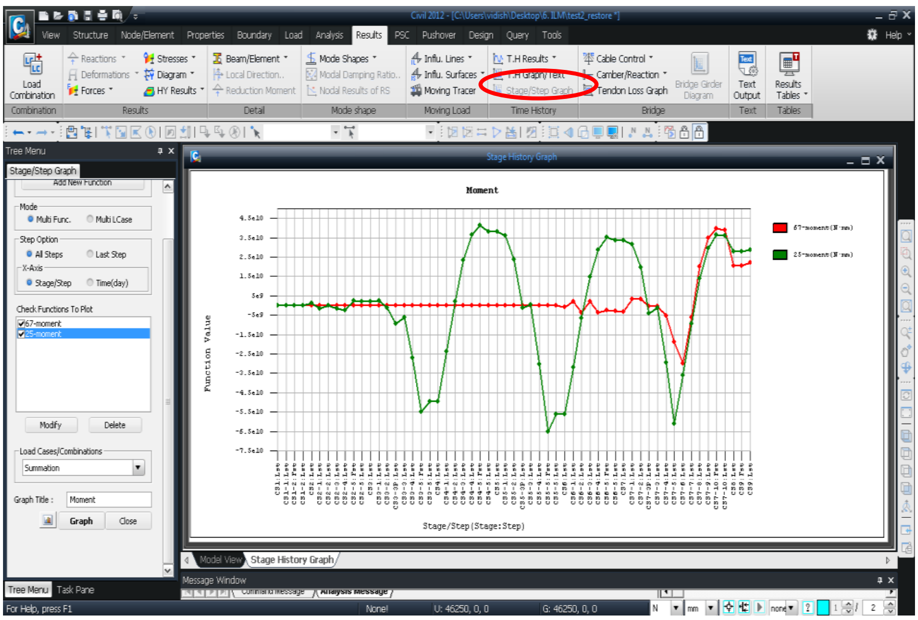


Figure 54 Moment History Graph for elements 67 and 25

Deflection Graph at Nose End

Deflections at nose end for each construction stage are reviewed. Displacements, section forces and stresses can be traced by construction stages and they can be reviewed by graph using Stage/Step History Graph function.

To review deflections at nose end by graph, first change construction stage to the stage at which the nose is activated, and then review the results.

Stage>**CS7-10**

Model View (Fig 55)

Results Tab > ***Stage/Step History Graph***

Define Function>**Displacement** [Add New Function]

Displacement>Name (**Nose**) ; Node Number (**1**)

Components>**DZ** ↵

Mode>**Multi Func.** ; Step Options>**All Steps**

X-Axis>**Stage/Step**

Check Functions To Plot>**Nose** (on)

Load Cases/Combinations>**Summation**

Graph Title (**Displacement**) [Graph]

After completion of graph, the name of graph or axis can be changed by clicking mouse right button on graph.

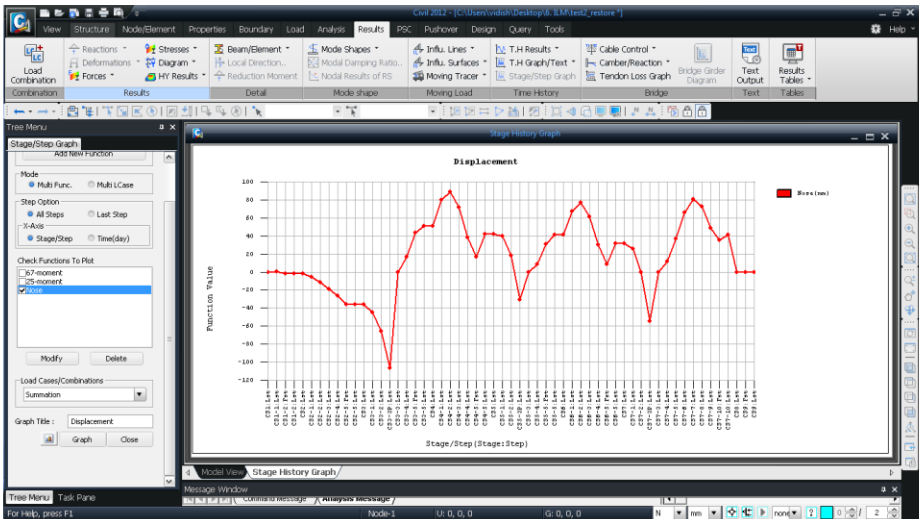


Figure 55 Node End Deflection at Each Construction Stage

Reviewing Results by Table

The construction stage analysis results can be reviewed by table using Records Activation Dialog function. In this function, various filtering options like element, load, construction stage and stress output location could be applied. Stress changes at I end, element 25 at which maximum stresses occur can be reviewed by table.

Tree Menu >Tables Tab > Results Table / Beam / ***Stress***

Node or Element (**25**)

Loadcase/Combination>**Summation(CS)** (on)

Stage/Step>**CS1:001(last) ~ Min/Max:min** (on)

Part Number>**Part i** (on) ↵

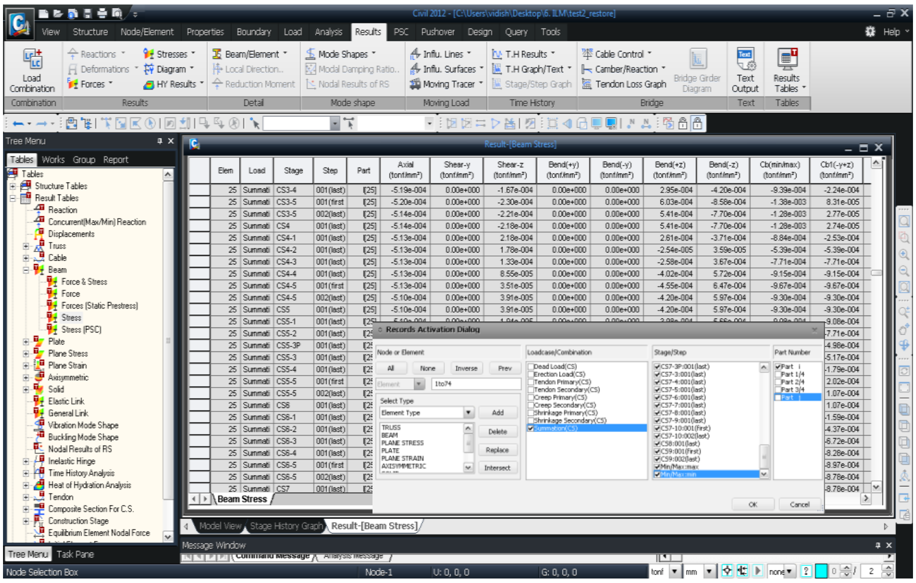


Figure 56 Stress Table at I end, Element 25

Reviewing Prestress Loss

Tensioning changes by construction stages due to prestress loss are reviewed. Change construction stage to the stage that includes the tendons, which will be reviewed, for only the tendons that are included at current construction stage can be reviewed. And then select Tendon Time-dependent Loss Graph menu. By clicking Animate button, the tensioning changes according to the construction stages can be reviewed by animation.

Results Tab > ***Tendon Time-Dependent Loss Graph***

Loss Graph Tendon>**BOT1-1** ; Stage>**CS9** ; Step>**Last Step**

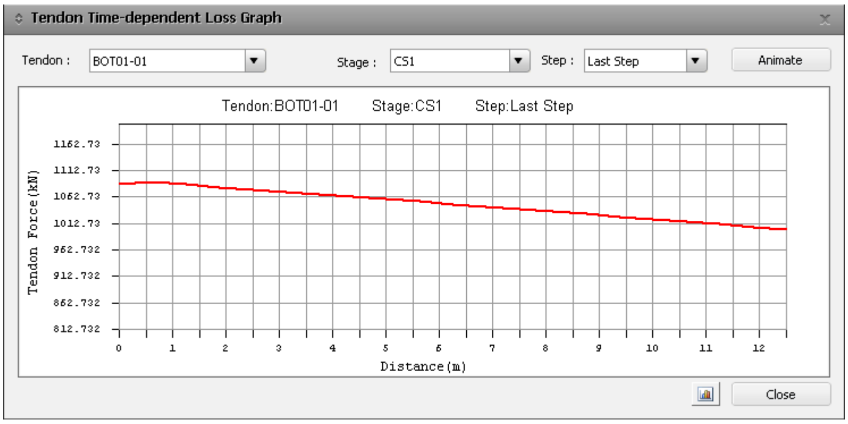


Figure 57 Tendon Force Loss Graph

Reviewing Tendon Coordinates

In MIDAS/Civil, tendon coordinates can be reviewed by table for every quarter points within each element. And tendon profile can be verified by clicking mouse right button and selecting Show Graph.

Tree Menu /Tables Tab / Result Tables / Tendon / ***Tendon Coordinates***

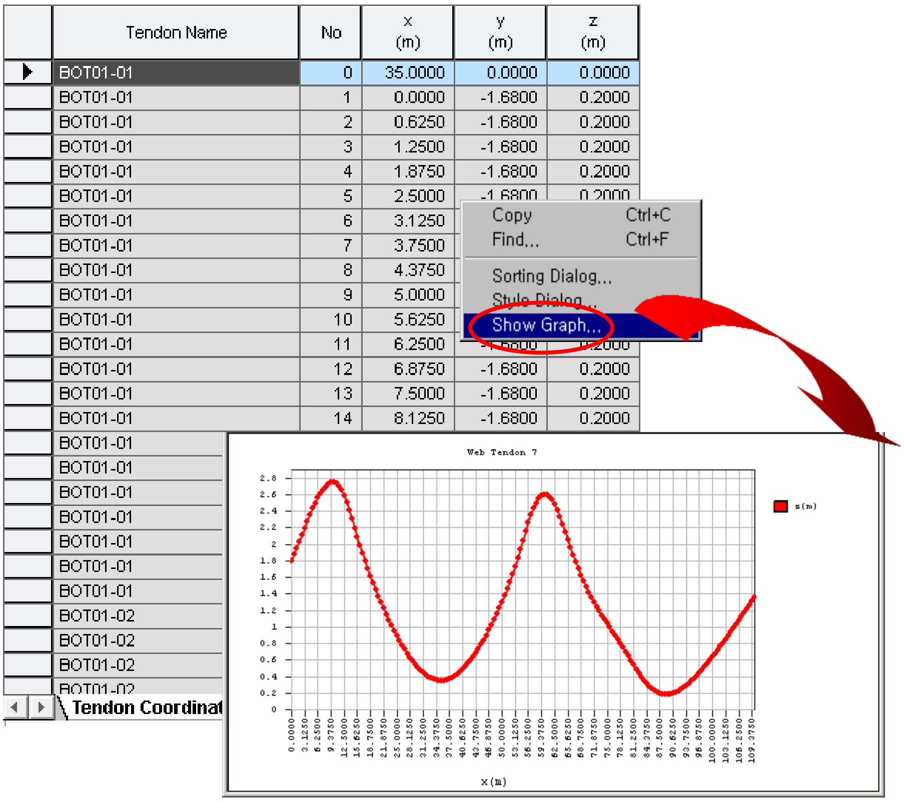


Figure 58 Prestress Force Loss Table

Reviewing Tendon Elongation

Tendon elongation and girder shortening are reviewed by table.

Results / Result Tables / Tendon / ***Tendon Elongation***

Tendon Name	Stage	Step	Tendon Elongation		Element Elongation		Summation	
			Begin (m)	End (m)	Begin (m)	End (m)	Begin (m)	End (m)
BOT01-01	CS1	001(last)	0.0000	0.0798	0.0000	0.0001	0.0000	0.0798
BOT01-02	CS3	001(last)	0.0000	0.2832	0.0000	0.0002	0.0000	0.2834
BOT01-03	CS5	001(last)	0.0000	0.2832	0.0000	0.0002	0.0000	0.2834
BOT01-04	CS7	001(last)	0.0000	0.2209	0.0000	0.0001	0.0000	0.2211
BOT02-01	CS2	001(last)	0.0000	0.2209	0.0000	0.0002	0.0000	0.2211
BOT02-02	CS4	001(last)	0.0000	0.2832	0.0000	0.0002	0.0000	0.2834
BOT02-03	CS6	001(last)	0.0000	0.2832	0.0000	0.0002	0.0000	0.2834
BOT02-04	CS7	001(last)	0.0000	0.0798	0.0000	0.0001	0.0000	0.0798
BOT03-01	CS1	001(last)	0.0000	0.0798	0.0000	0.0001	0.0000	0.0798
BOT03-02	CS3	001(last)	0.0000	0.2832	0.0000	0.0002	0.0000	0.2834
BOT03-03	CS5	001(last)	0.0000	0.2832	0.0000	0.0002	0.0000	0.2834
BOT03-04	CS7	001(last)	0.0000	0.2209	0.0000	0.0001	0.0000	0.2211
BOT04-01	CS2	001(last)	0.0000	0.2209	0.0000	0.0002	0.0000	0.2211
BOT04-02	CS4	001(last)	0.0000	0.2832	0.0000	0.0002	0.0000	0.2834
BOT04-03	CS6	001(last)	0.0000	0.2832	0.0000	0.0002	0.0000	0.2834
BOT04-04	CS7	001(last)	0.0000	0.0798	0.0000	0.0001	0.0000	0.0798
BOT05-01	CS1	001(last)	0.0000	0.0798	0.0000	0.0001	0.0000	0.0798
BOT05-02	CS3	001(last)	0.0000	0.2832	0.0000	0.0002	0.0000	0.2834
BOT05-03	CS5	001(last)	0.0000	0.2832	0.0000	0.0002	0.0000	0.2834
BOT05-04	CS7	001(last)	0.0000	0.2209	0.0000	0.0001	0.0000	0.2211
BOT06-01	CS2	001(last)	0.0000	0.2209	0.0000	0.0002	0.0000	0.2211
BOT06-02	CS4	001(last)	0.0000	0.2832	0.0000	0.0002	0.0000	0.2834
BOT06-03	CS6	001(last)	0.0000	0.2832	0.0000	0.0002	0.0000	0.2834
BOT06-04	CS7	001(last)	0.0000	0.0798	0.0000	0.0001	0.0000	0.0798
BOT07-01	CS1	001(last)	0.0000	0.0798	0.0000	0.0001	0.0000	0.0798

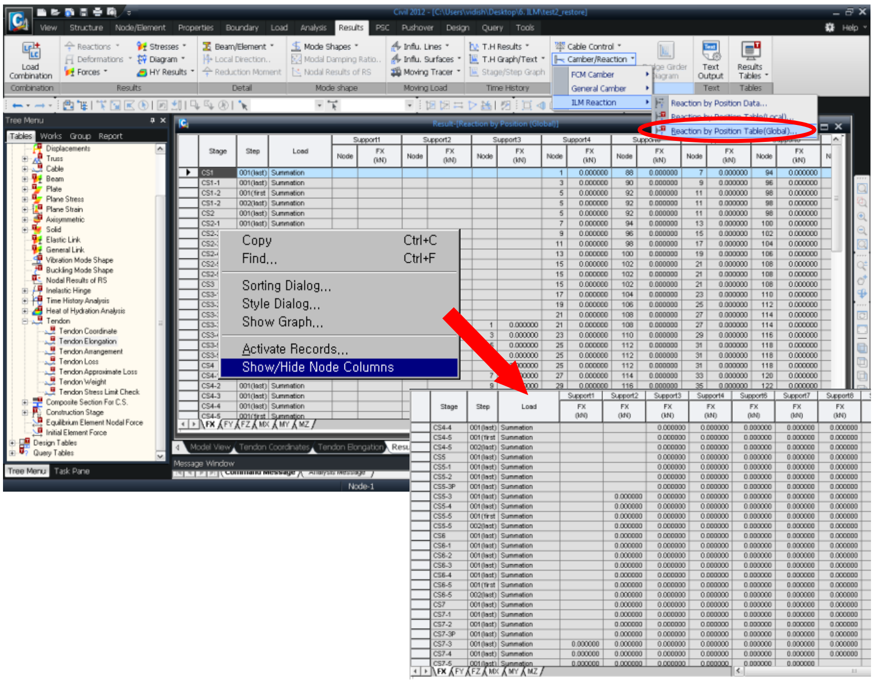
Tendon Elongation

Figure 59 Tendon Elongation and Girder Shortening Table

Reviewing reactions at each construction stage

If the ILM Wizard is used, the support points are auto-generated. Otherwise, the information on the reaction points should be defined in Reaction by Position data in the results menu. In the case of a bridge constructed by the ILM, the reactions in GCS are produced, according to the change of construction stages at the piers and abutments, in the table in Spread Sheet format.
The reactions at each construction stage for the summation load case are reviewed are reviewed.

Results Tab / ILM Reaction / ***Reaction by Position Table (Global)***



Node Columns can be shown or hidden by clicking Show/Hide Node Columns in the text menu.

Figure 60 Reaction Table at each construction stage

Reviewing Section Forces by Factored Load

Ultimate Strength for post-construction stage should be checked. The factored load includes live load, temperature load, support settlement and dead loads. The structural analysis results for loads that are defined other than Construction Stage Load Type is performed at PostCS Stage and it can be combined with the construction stage analysis results.

Stage>**PostCS**
Results Tab >***Load Combination***
Active (on) ; Name (**Dead**) ; Type>**Add**
Load Case>**Dead Load (CS)** ; Factor (**1.3**)
Load Case>**Tendon Secondary (CS)** ; Factor (**1**)
Load Case>**Creep Secondary (CS)** ; Factor (**1.3**)
Load Case>**Shrinkage Secondary (CS)** ; Factor (**1.3**)

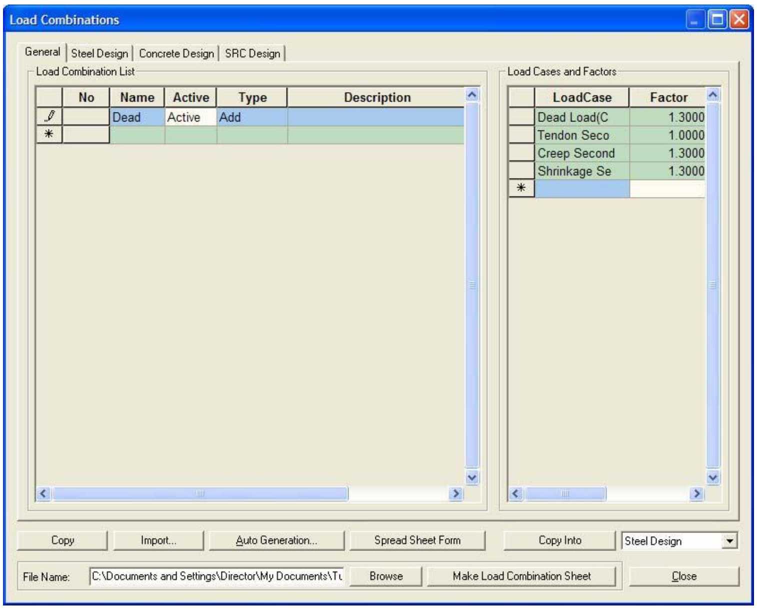


Figure 61 Load Combination Dialog Box

Moment diagram due to factored load combination is reviewed.

Results Tab / Forces / ***Beam Diagrams***

Load Cases/Combination>**CB:Dead**

Components>**My**

Output Options>**5 Points** ; **Line Fill**

Type of Display>**Contour** (on) ↵

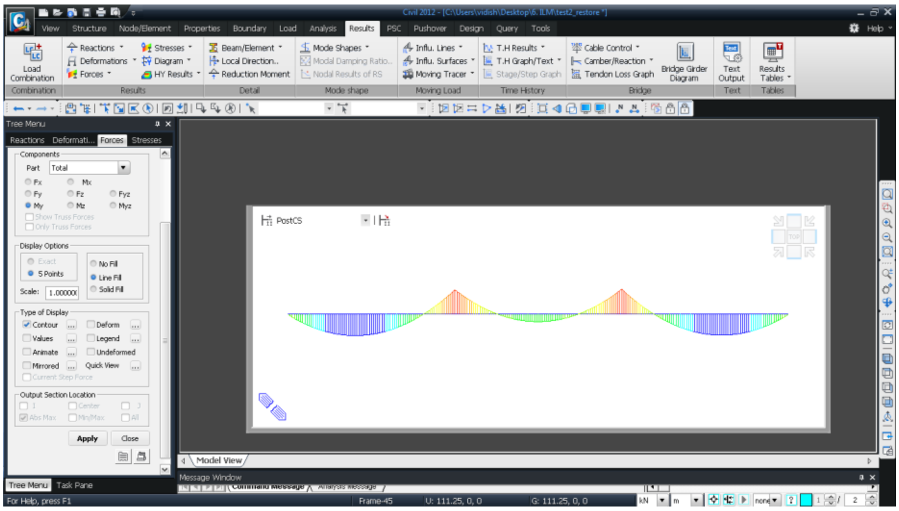


Figure 62 Beam Moment Diagram