Advanced Application 19

Construction Stage Analysis for FSM (Full Staging Method) using general functions

Civil

Contents

Outline

FSM (Full Staging construction Method) is a very basic method in constructing post-tensioned concrete bridges. Dead weight of concrete, formwork and falsework are fully shored over the full spans of a bridge until the concrete gains a certain level of strength.

FSM can be economical if the horizontal alignment of a bridge is curved or the width of the bridge deck widens, provided that the height of the piers are not too high.

In the case of a bridge with long spans, the use of continuous tendons can be limited, thereby requiring construction joints. Each segment may be constructed sequentially span by span. Structural analysis is carried out on the basis of construction stages defined by the construction joints. Although a bridge is supported by shoring, FSM is generally analyzed with the assumption that effect of support is negated by the effect of prestressing.

When FSM is applied to a bridge with continuous spans, the first stage is a simple span, and it becomes continuous with the progress of the construction stages. In comparison with an analysis that does not consider construction stages, the construction stage analysis results in lower negative support moments higher positive span moments. As such, a bridge constructed by FSM needs to be analyzed with construction stages reflecting both the change in structure, element load and boundary conditions as well as time-dependent material properties, including creep, shrinkage and modulus of elasticity.

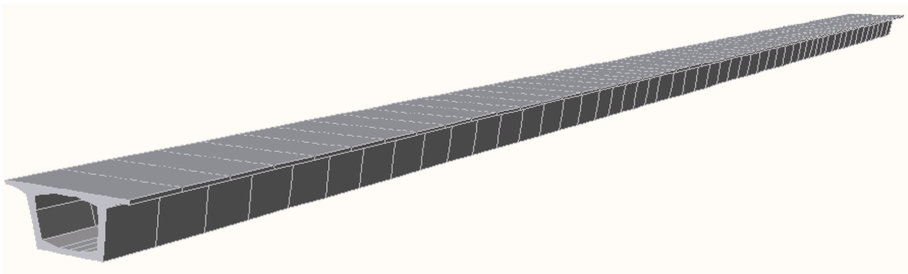


Figure 1. Bridge to be analyzed

Bridge profile and general section

This example has been simplified from an actual project for the purpose of illustrating construction stage analysis using FSM.

The bridge profile is defined as follows:

Structure type: 3 continuous span PSC Box girder bridge (F.S.M)
Spans: L = 40.0 + 45.0 + 40.0 = 125.0m
Bridge width: 8.5m
Skew angle: 90˚

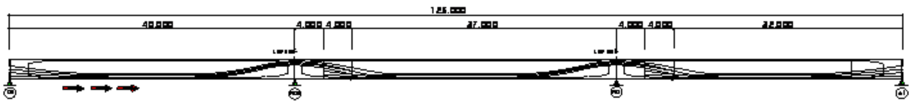


Figure 2. Longitudinal Section

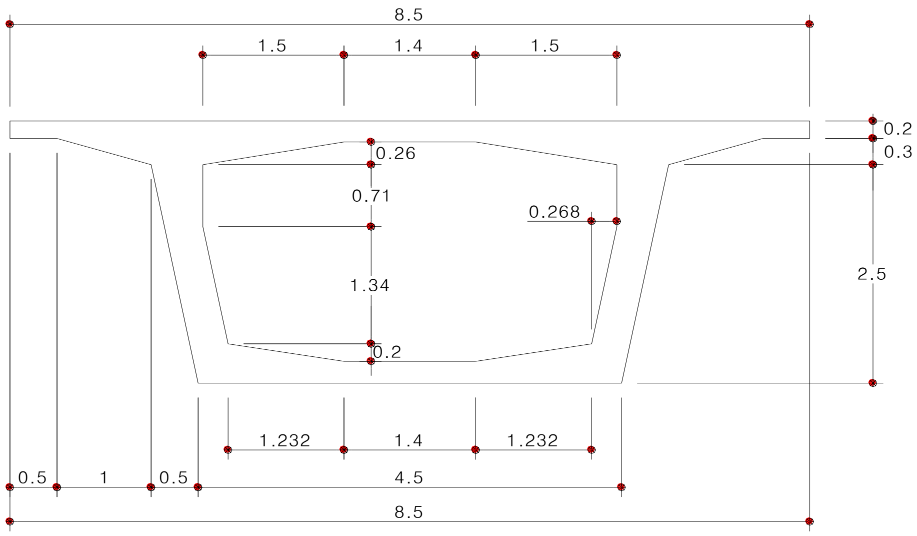


Figure 3. Cross Section

Materials & Strength

▶ **Concrete**

1) Specified Strength: $f_{cu} = 45MPa$
2) Modulus of Elasticity: $E_c = 3.0124 \times 10^4 MPa$

▶ **PS Steel Tendons**

1) Yield Strength: $f_{py} = 1580MPa$
2) Tensile Strength: $f_{pu} = 1860MPa$
3) Nominal Sectional Area: $A_p = 100cm^2$
4) Modulus of Elasticity: $E_p = 1.95 \times 10^5 MPa$
5) Initial Prestressing Force: $f_{pj} = 0.75 f_{pu} = 1395MPa$
6) Anchorage Slip: $\Delta s = 6mm$
7) Coefficient of Curvature Friction : $\mu = 0.25 / rad$
8) Coefficient of Wobble Friction: $k = 0.0066 / m$

Loads

▶ **Primary loads and special loads pertaining to the primary loads**

1) Dead Loads
   A. Reinforced Concrete: $24.52 kN / m^2$
   B. Asphalt Concrete: $22.56 kN / m^2$
   C. Barriers and safety fences
   D. Prestress, creep, shrinkage
2) Live Loads
   A. Vehicle Loads: *Types HA and HB Loading*
3) Differential settlements
   : The worst combination of each pier settlement of 10mm

▶ **Secondary loads**

1) Temperature
   A. For total deformation (±15˚)
   B. Temperature differential between top & bottom chords (±5˚)
2) Wind

Composition of the Construction Stages

This figure below represents the entire construction stage process. Construction stages are generated excluding the erection of the shoring and temporary bents themselves, which have no effect on the structure.

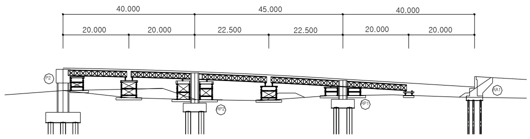
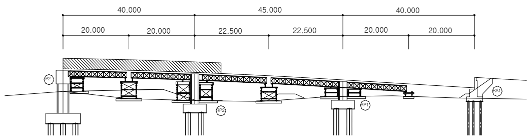
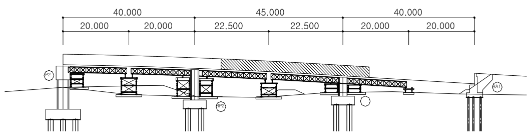
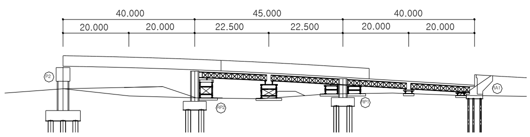
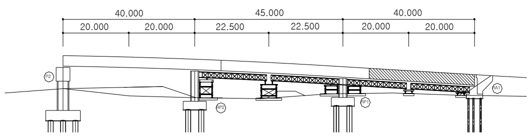
		Install tempor ary
		Install formw ork
		Install formw ork
		Remo ve tempo
		Install formw ork

Figure 4. Construction Stage Chart

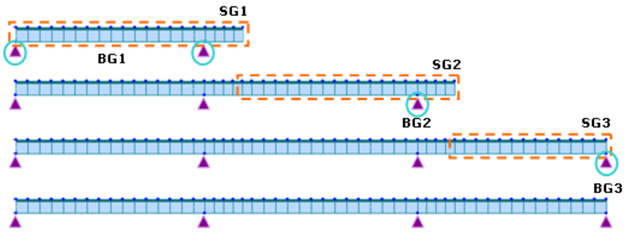
The following construction stages are reflected in the analysis.

CS1 (30 days)

CS2 (30 days)

CS3 (30 days)

CS4 (10,000 days)



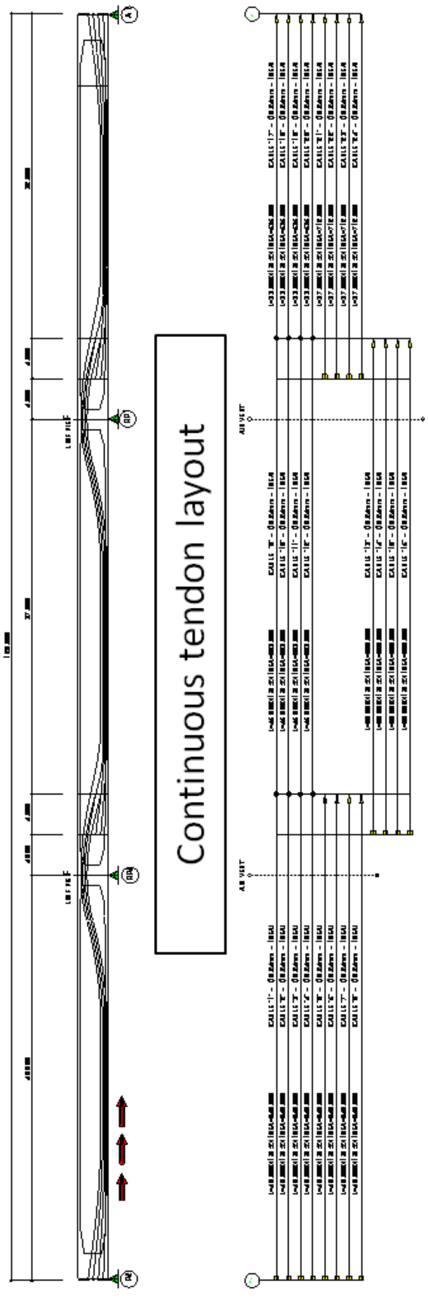


Figure 5. Tendon Placement Layout

Work Environment Settings

For FSM construction stage analysis, open a new file, (***New Project***), and save as (***Save***) **'FSM.mcb'.**

Select '**kN**' and '**m**' for the unit system. The unit system can be conveniently changed at any time later depending on your preferred types of input data.

/ ***New Project***
/ ***Save* (FSM)**

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

The unit system can be changed by clicking "Unit Selection" ( ) on the Status Bar at the bottom of the screen.

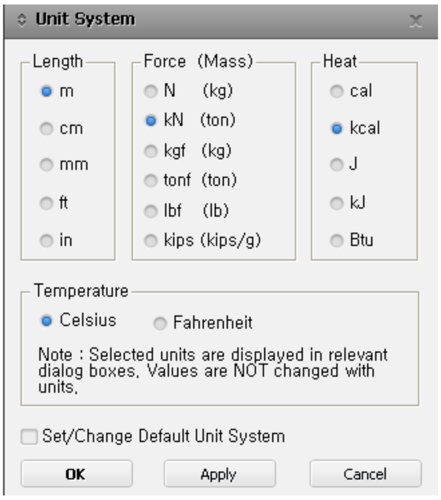


Figure 6. Unit System Setting

Definition of Properties

Definition of Materials

Define the material of the PSC box by selecting one from the built-in database. The material for tendons can be defined using the User Defined function.

Properties / ***Material Properties***

Click Add ↵

Type>**Concrete** ; Standard>**BS(RC)**
DB>**C45** ↵
Apply

Name>**Tendon** ; Type>**User Defined**
Modulus of Elasticity **(1.95e8)**
Weight Density **(78.5)** ↵

The tendon weight is automatically accounted for after grouting.

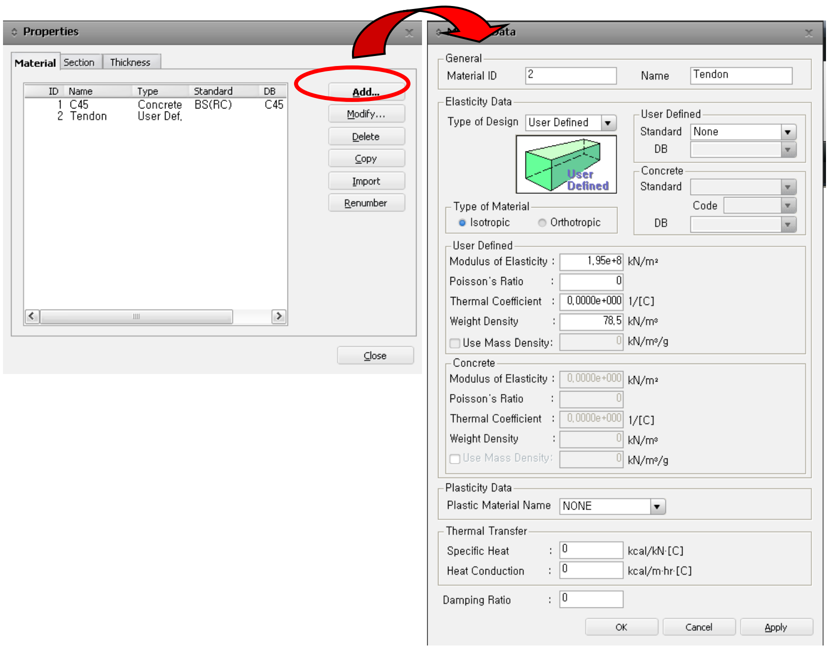


Figure 7. Material Data Input Dialog Box

Definition of Section

Refer to the cross section dimensions in Figure 8 to define the section of the PSC box.

Properties / ***Section Properties***

Click Add ↵

PSC tab

Section ID **(1)** ; Name **(Span)**

PSC-1CELL, 2CELL

Joint On/Off>**JO1** (on), **JI1** (on), **JI3** (on), **JI5** (on)

Web Thick> **for Shear t1** (on), **t2** (on), **t3** (on), **for Torsion(min)** (on)

Offset>**Center-Top**

Outer

HO1 **(0.2)** ; HO2 **(0.3)** ; HO2-1 **(0)** ; HO3 **(2.5)**

BO1 **(1.5)** ; BO1-1 **(0.5)** ; BO2 **(0.5)** ; BO3 **(2.25)**

Inner

HI1**(0.24)** ; HI2**(0.26)** ; HI2-1**(0)** ; HI3**(2.05)** **;** HI3-1**(0.71)**

HI4 **(0.2)** ; HI4-1 **(0)** ; HI5 **(0.25)**

BI1**(2.2)** ; BI1-1**(0.7)** ; BI2-1**(2.2)** ; BI3**(1.932)** **;** BI3-1**(0.7)**

Or click Table Input... to enter the input data in a table.

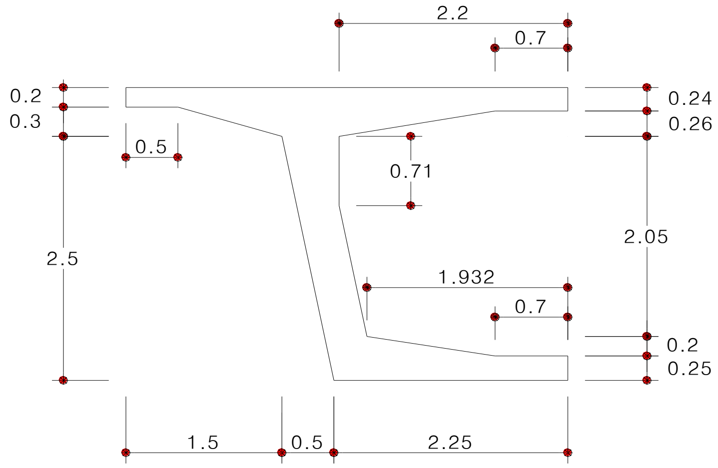


Figure 8. Input Data for the Cross Section

Checking on "Mesh Size for Stiff Calc." enables us to define a maximum size of mesh, which is used to calculate the section properties.

"Consider Shear Deformation" accounts for shear deformation.

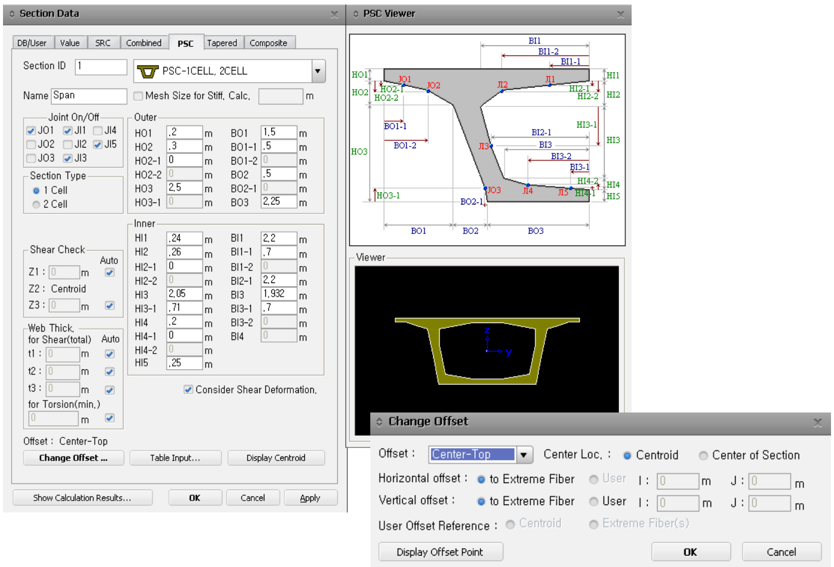
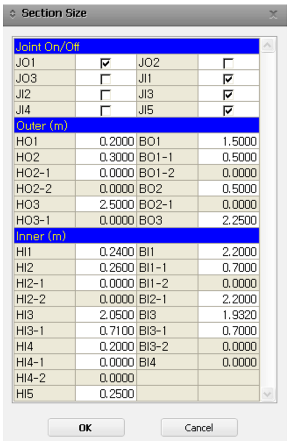


Figure 9. Section Input Dialog Box



Joint On/Off			
JO1	☑	JO2	☐
JO3	☐	JI1	☑
JI2	☐	JI3	☑
JI4	☐	JI5	☑
Outer (m)			
HO1	0.2000	BO1	1.5000
HO2	0.3000	BO1-1	0.5000
HO2-1	0.0000	BO1-2	0.0000
HO2-2	0.0000	BO2	0.5000
HO3	2.5000	BO2-1	0.0000
HO3-1	0.0000	BO3	2.2500
Inner (m)			
HI1	0.2400	BI1	2.2000
HI2	0.2600	BI1-1	0.7000
HI2-1	0.0000	BI1-2	0.0000
HI2-2	0.0000	BI2-1	2.2000
HI3	2.0500	BI3	1.9320
HI3-1	0.7100	BI3-1	0.7000
HI4	0.2000	BI3-2	0.0000
HI4-1	0.0000	BI4	0.0000
HI4-2	0.0000		
HI5	0.2500		

Figure 10. Table Input (PSC)

Cross sectional dimensions can be entered via a table upon clicking Table Input... for the PSC section.

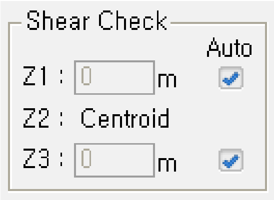
This is faster than directly entering the data in the dialog box for a large amount of dimensional data.

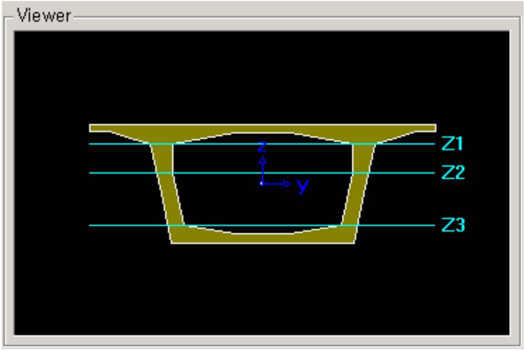
The table is compatible with Excel. Frequently used cross sectional dimensions can be saved to copy & paste later.

The table becomes compatible with Excel by entering "0" for Check Off (☐) and "1" for Check on (☑).

Shear Check

Assign the locations for shear calculations on the PSC section. Numerical data can be entered manually, or if “Auto” is selected, shear calculations take place at the top and bottom of the web(s). The shear results are displayed in no. 5~10 of the Beam Stress (PSC).





Web Thick.

for Shear(total)
Enter the thicknesses to be used for shear calculations at the locations defined for Shear Check at Z1 through Z3. Enter the sum of web thicknesses at a given location. Check on “Auto” for automatic calculations.

for Torsion(min.)
Enter a minimum thickness for torsion calculation.

Definition of Time-dependent Material Properties

Define the time-dependent properties of the concrete (creep coefficients shrinkage and strength).

Properties / ***Time Dependent Material / Creep/Shrinkage***

Click Add ; Name>**C45** ; Code>**CEB-FIP(1990)**

Compressive strength of concrete at the age of 28 days **(45000)**

Relative Humidity of ambient environment (40-99) **(70)**

Notational size of member **(0.364)**

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

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

Properties / ***Time Dependent Material / Comp. Strength***

Click Add ; Name>**C45** ; Code>**CEB-FIP**

Mean compressive strength of concrete at the age of 28 days **(53000)**

Type of cement>**N, R : 0.25** ↵

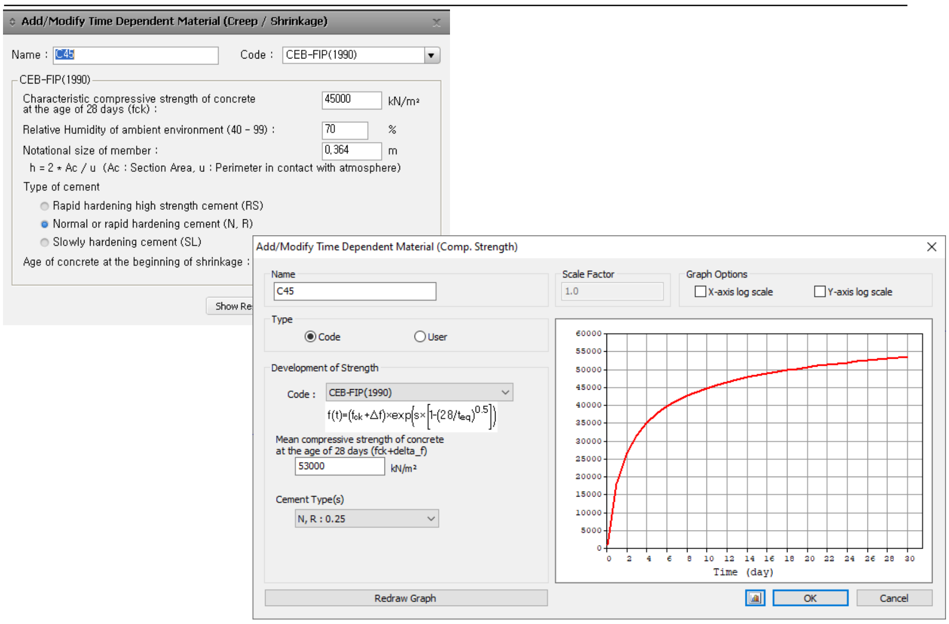


Figure 11. Time Dependent Material Data

Link the time dependent material properties to the material properties. The creep coefficients, shrinkage and concrete strength curves defined earlier need to be linked to the corresponding material property in order to carry out construction stage analysis reflecting their effects.

Properties / ***Time Dependent*** / ***Material Link***
  Time Dependent Material Type
    Creep/Shrinkage>**C45**
    Comp. Strength>**C45**
  Select Material for Assign>Materials>
    1:C45 [>] Selected Materials [Add / Modify]

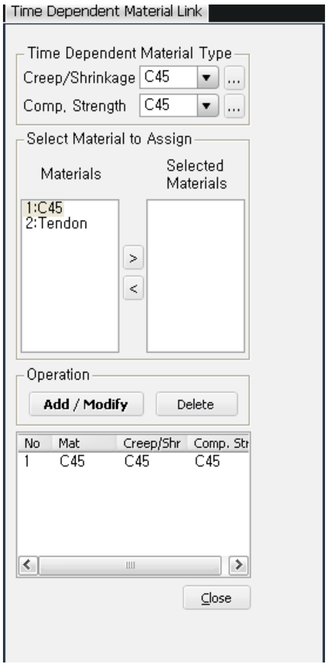


Figure 12. Linking Time Dependent Material Property to the Material Property.

Structural Modeling

Element Generation

Generate a girder using the “Extrude” function.

Node/Element / ***Create Nodes***
Coordinates (x, y, z) **(0, 0, 0)** Apply ↵

Node/Element / ***Extrude***
Select All
Extrude Type>**Node -> Line Element**
Element Attribute
Element Type>**Beam**
Material>**1: C45**
Section>**1: Span**
Translation > Unequal Distance
Axis>**x**
Distances **(16@2.5, 5@2, 14@2.5, 5@2, 12@2.5)** Apply ↵

Zoom Fit

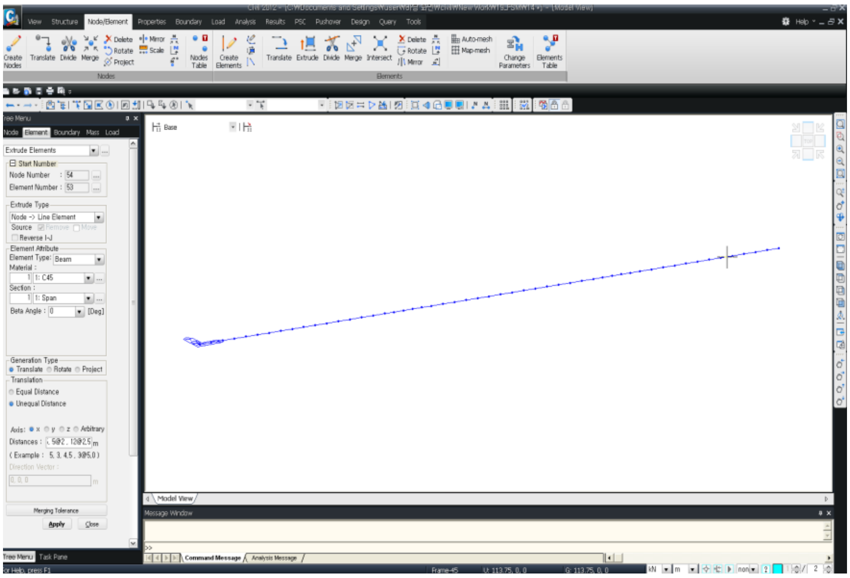


Figure 13. Girder Generation

Support Generation

Considering the spans (40+45+40), create nodes to which boundary conditions will be assigned.

Node/Element / ***Create Nodes***
Start Node Number ... ↵
Node Numbering Option>**User-Defined Number**
Newly Created Number **(61)** OK ↵
Coordinates (x, y, z) **(0, 1.5, -3)**
Copy
Number of Times **(1)**
Distances (dx, dy, dz) **(0, -3, 0)** Apply ↵

Node/Element / ***Translate Node***
Select Recent Entities
Mode>**Copy**
Translation
Unequal Distance
Axis>**x**
Distance **(40, 45, 40)** Apply ↵

Since the depth of the girder is 3m, and the distance between the bearings is 3m with the working point being Center-Top, the supports are created at Z=-3m & Y=±1.5m.

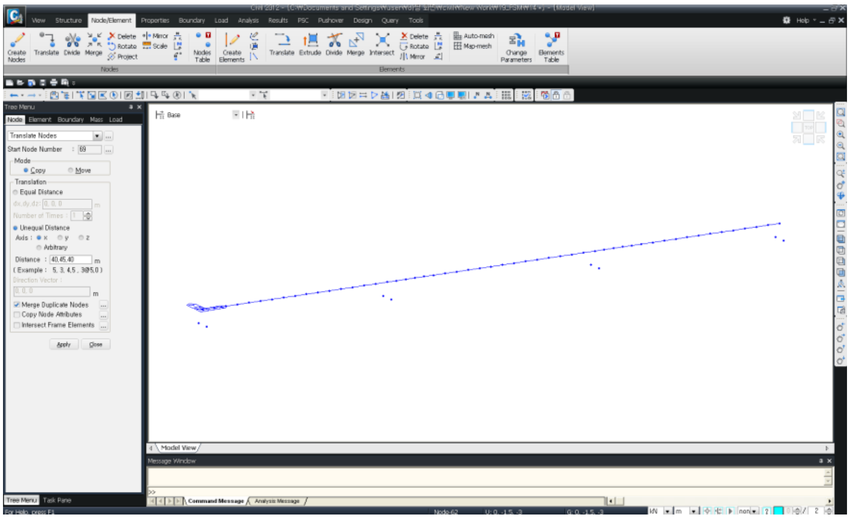


Figure 14. Generation of Support Nodes

Group Definition

Refer to “Construction Stage Configuration” on Figure 4 for the list of the groups to be defined.

Structure / Group / ***Structure***
Name **(SG)** ; Suffix **(1to3)** Add ↵

Structure / Group / B/L/T / ***Define Boundary Group***
Name **(BG)** ; Suffix **(1to3)** Add ↵

Structure / Group / B/L/T / ***Define Load Group***
Name **(Dead)** ; Add ↵ ;
Name **(Superimposed Dead)** ; Add ↵
Name **(PS)** ; Suffix **(1to3)** ; Add ↵
Name **(Diaphragm)** ; Suffix **(1to3)** ; Add ↵

Structure / Group / B/L/T / ***Define Tendon Group***
Name **(A)** ; Suffix **(1to4)** Add ↵
Name **(B)** ; Suffix **(1to4)** Add ↵
Name **(C)** ; Suffix **(1to4)** Add ↵

Tendon Group is not used for composing the construction stages, but is defined to check the results for each group.

Refer to the name assignment of the “Tendon Profile” to see the items of the Tendon Group.

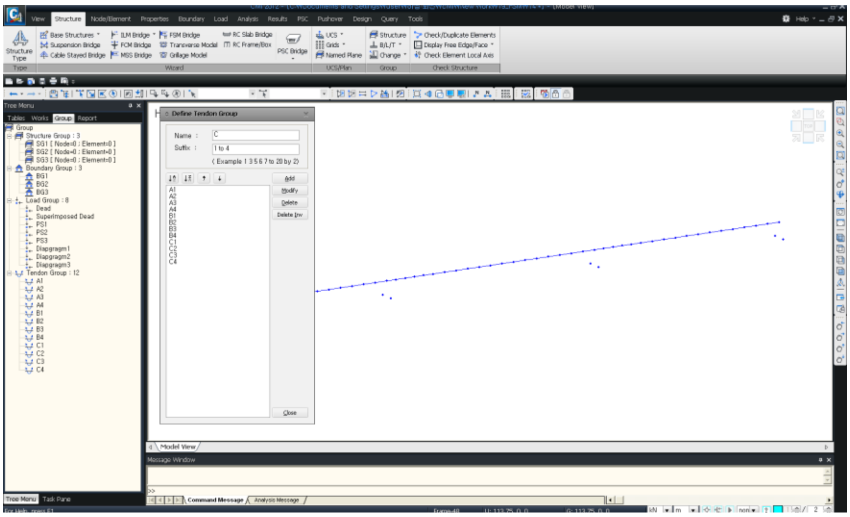


Figure 15. Group Generation

Structure Group Assignment

Assign the elements, which will be activated at each stage, to SG1~3 respectively. Assign the elements to Structure Group by using "Drag & Drop," or by right-clicking and selecting "Assign".

Elements Numbers ; **Front View**

Group Tab in the Tree Menu

Type the numbers of nodes and elements as below

Select Nodes : **61to64** & Elements : **1to20**
Structure Group > SG1

Drag & Drop or (Context Menu) ***Assign***

Select Nodes : **65to66** & Elements : **21to39**
Structure Group > SG2 ***Drag & Drop*** or (Context Menu) ***Assign***

Select Nodes : **67to68** & Elements : **40to52**
Structure Group > SG2 ***Drag & Drop*** or (Context Menu) ***Assign***

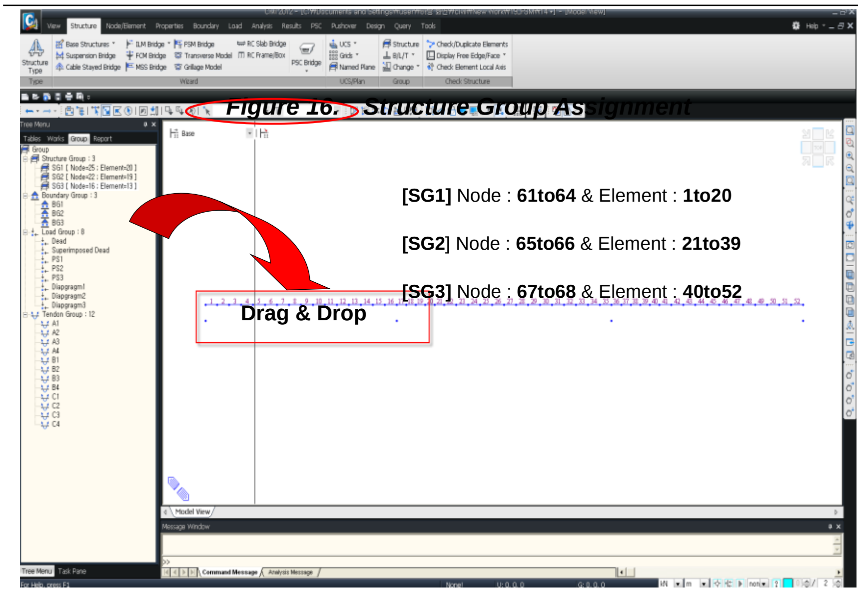


Figure 16. Structure Group Assignment

Boundary Conditions Input

Rigid Links

Considering the centroid of the cross section of the PSC Box, rigid links are connected to the supports.

Iso View

Boundary / ***Elastic Link***
Boundary Group>**BG1**
Link Type>**Rigid Type**
2Nodes **(1, 61) ; (1, 62) ; (18, 63) ; (18, 64)** ↵

Boundary Group>**BG2**
2Nodes **(37, 65) ; (37, 66)** ↵

Boundary Group>**BG3**
2Nodes **(53, 67) ; (53, 68)** ↵

Turn on the node number if necessary when picking up nodes

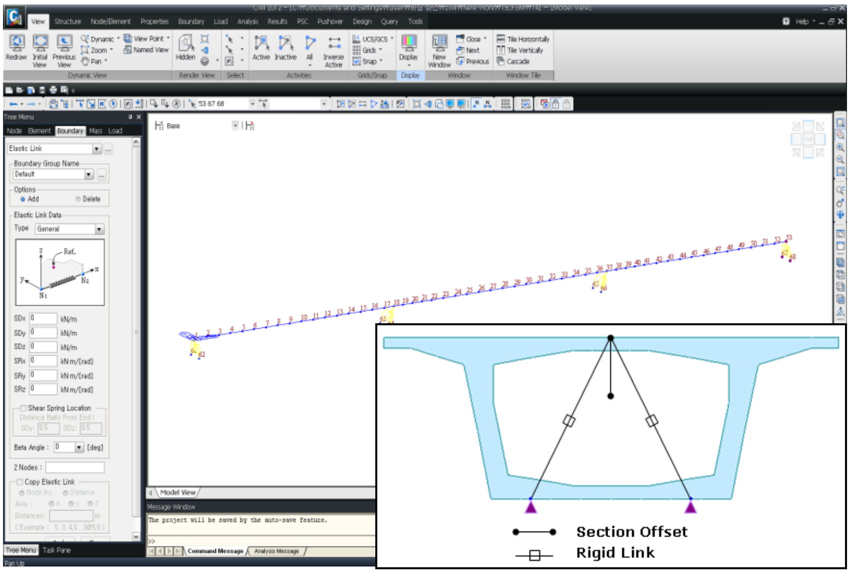


Figure 17. Rigid Links

Supports Input

Considering the construction stages, the supports are defined as below.

Top View ; ***Redraw***

Boundary / ***Define Supports***
Boundary Group>**BG1**
Select Single (Node : **61**) ; Support Type>**Dy** (on), **Dz**(on) ↵
Select Single (Node : **62**) ; Support Type>**Dz**(on) ↵
Select Single (Node : **63**) ; Support Type>**Dx**(on), **Dy**(on), **Dz**(on) ↵
Select Single (Node : **64**) ; Support Type>**Dx**(on), **Dz**(on) ↵

Boundary Group>**BG2**
Select Single (Node : **65**) ; Support Type>**Dy**(on), **Dz**(on) ↵
Select Single (Node : **66**) ; Support Type>**Dz**(on) ↵

Boundary Group>**BG3**
Select Single (Node : **67**) ; Support Type>**Dy**(on), **Dz**(on) ↵
Select Single (Node : **68**) ; Support Type>**Dz**(on) ↵

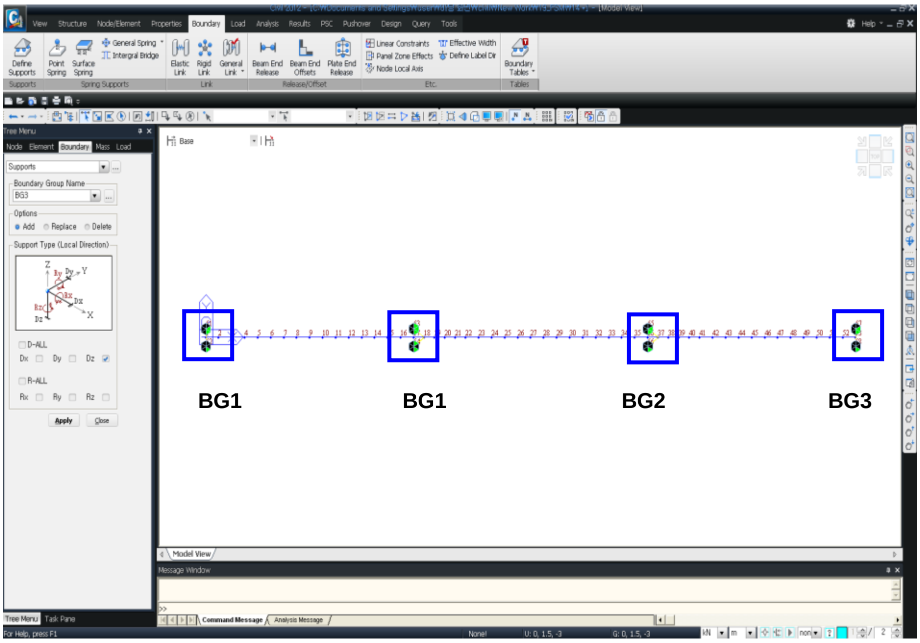


Figure 18. Boundary Condition Input

Construction Stage Loads Input

Define Load Conditions

Define load cases for analysis.

We take the time to define the load “Type” then we can take advantage of the ability to automatically generate load combinations using the "Auto Generate” function. Using these Types of load case we may generate the load combinations after application of the load factors as per the design standard.

Redraw

Load / ***Static Load Cases***

Name **(Self Weight)** ; Type>**Construction Stage Load (CS)** ↵
Name **(Non-Structure Dead)** ; Type>**Construction Stage Load (CS)** ↵
Name **(Prestress)** ; Type>**Construction Stage Load (CS)** ↵
Name **(Superimposed)** ; Type>**Construction Stage Load (CS)** ↵
Name **(Wind)** ; Type>**Wind Load on Structure (W)** ↵
Name **(Temperature (+))** ; Type>**Temperature (T)** ↵
Name **(Temperature (-))** ; Type>**Temperature (T)** ↵
Name **(Top-Bot Temp Diff(+))** ; Type>**Temperature Gradient (TPG)** ↵
Name **(Top-Bot Temp Diff(-))** ; Type>**Temperature Gradient (TPG)** ↵

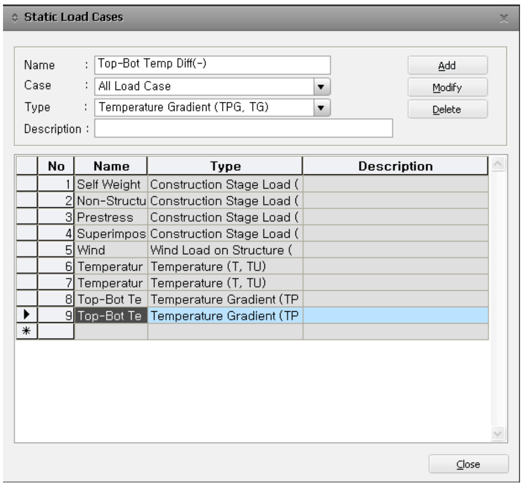


Figure 19. Load Cases Definition

Self Weight

Enter the self weight.

Define the structure's self weight and activate it at the first construction stage. Then the self weights of the elements activated in the subsequent construction stages will automatically be applied.

Load / ***Self Weight***

Load Case Name>**Self Weight**

Load Group Name>**Dead**

Self Weight Factor>**Z (-1)** Add ↵

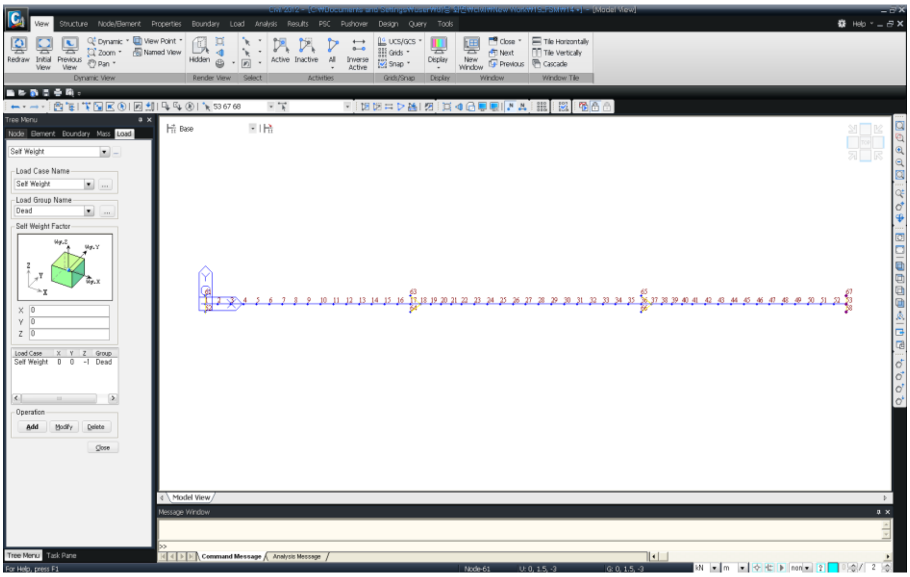


Figure 20. Self Weight Input

Dead Load

Enter diaphragms and construction joint blocks, as loads as they have not been reflected in the model.

Front View

Load / ***Element Beam Loads***

1 52 ***Select Elements by Identifying*** (**1, 52**) ↵

Load Case Name>**Non-Structure Dead**

Direction>**Global Z**

Relative

x1**(0)** ; x2 **(1)** ; w**(-220.34)** Apply ↵

19to21 38to40 ***Select Elements by Identifying*** (**19to21, 38to40**) ↵

Relative

x1**(0)** ; x2 **(1)** ; w**(-63.0)** Apply ↵

16 35 ***Select Elements by Identifying*** (**16, 35**) ↵

Absolute

x1**(1.5)** ; x2 **(2.5)** ; w**(-220.34)** Apply ↵

17 36 ***Select Elements by Identifying*** (**17, 36**) ↵

Absolute

x1**(0)** ; x2 **(1)** ; w**(-220.34)** Apply ↵

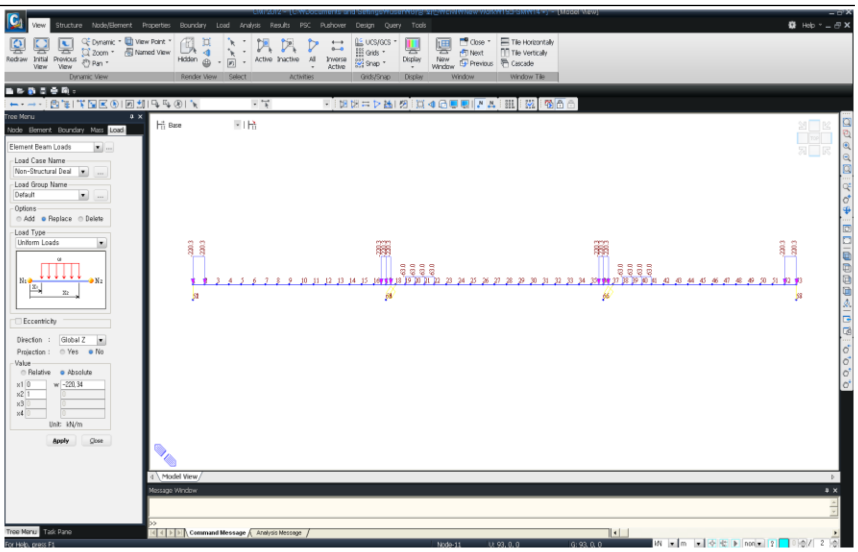


Figure 21. Miscellaneous Dead Loads

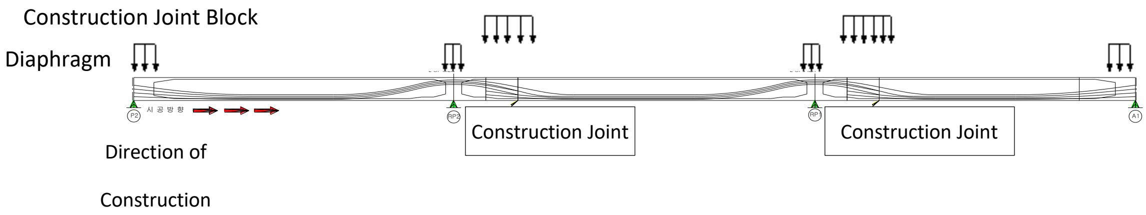
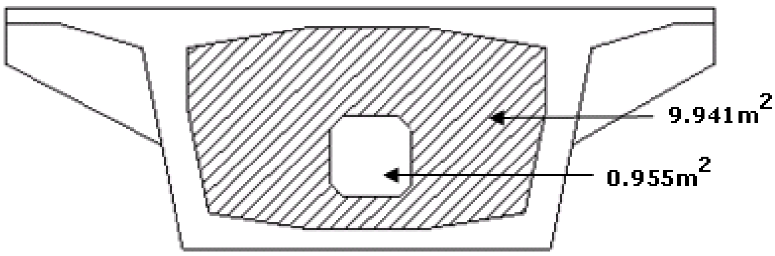


Figure 22. Dead Load Layout

The diaphragms at the supports and the construction joint blocks have not been considered as structural elements in this longitudinal analysis and are thus treated as loads. Their cross sectional areas are calculated and converted into Beam Load over the corresponding lengths. Other additional dead loads may exist, but are ignored in this Tutorial.

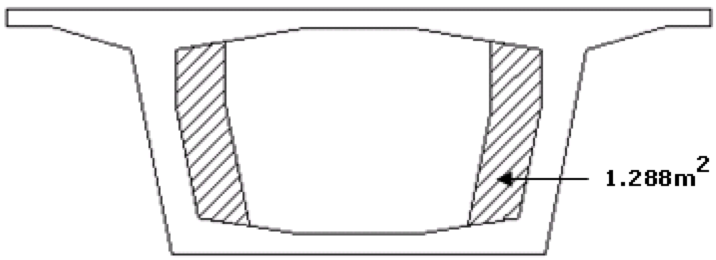
Diaphragm (End: 2m, Intermediate Support: 2.5m)

$$Area = 9.941m^2 - 0.955m^2 = 8.986m^2$$
$$P = 8.986m^2 \times 24.52kN/m^3 = 220.34kN/m$$



Construction Joint Block

$$Area = 1.288m^2 \times 2EA = 2.576m^2$$
$$P = 2.576m^2 \times 24.52kN/m^3 = 63kN/m$$



We need to assign the loads to Load Groups and activate the Load Groups in the corresponding construction stages.

Because the magnitudes of the Beam Loads are the same, setting the Load Group to Default is convenient for input. We will now see how to modify the Load Group using the Table Tab.

By selecting the desired columns, we can adjust the locations in Beam Load Table. The row column containing the Group information is located at the end of the Table. For convenience, we will select the entire column, and move it next to the Element numbers.

Assign Load Group: Diaphragm1 to 3 to the loads in order to activate them in Stages 1 through 3.

Load / Load Tables / Static Load / ***Beam Loads***

Assign

Element 1~20> **Diaphragm1**

Element 21~39>**Diaphragm2**

Element 40~52>**Diaphragm3**

Note that the Group column is found at the last column in the table as shown in the first figure of the three figures below. In the second figure, the Group column was relocated to the front for convenience. The third figure depicts how Diaphram is applied to the elements.

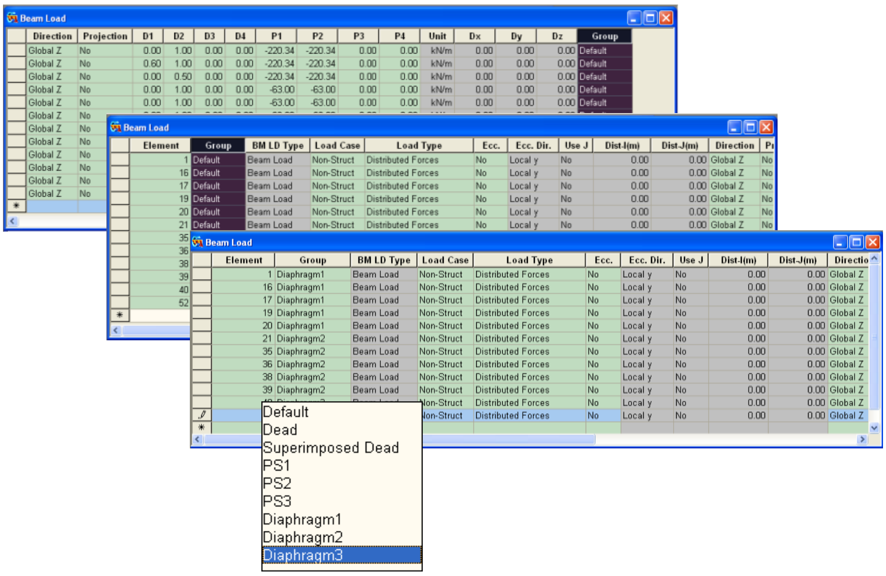


Figure 23. Changing Load Group using Table

Tendon Prestress Load

Define the properties of the Tendon related to the material, strength, losses. etc

---

Load / Temp./Prestress / *Tendon Property*

Click Add ↵

Tendon Name **(Tendon)**
Tendon Type> **Internal(Post-Tension)** ; Material> **2: Tendon**
Total Tendon Area> **0.0016112**
or Strand Diameter> **12.9mm(1χ3)** ; Number of Strands> **19**
Duct Diameter **(0.1)**
Relaxation Coefficient> **CEB-FIP(2.5%)**
Ultimate Strength> **(1860000)** ; Yield Strength> **(1580000)**
Curvature Friction Factor> **(0.25)** ; Wobble Friction Factor> **(0.0066)**
Anchorage Slip(Draw in)> Begin**(0.006)** , End**(0.006)**
Bond Type> **Bonded**

Relaxation Coefficient can be defined by selecting Magura equation, JTG04 or CEB-FIP Code.

If "Unbonded" is selected, the section stiffness is calculated on the basis of the net cross section. "Bonded" reflects the composite stiffness reflecting the tendons.

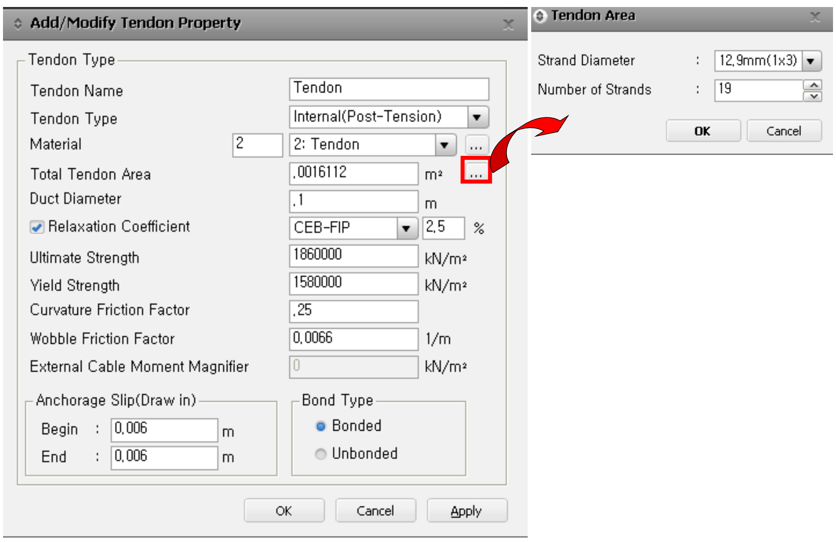


Figure 24. Tendon Property Dialog

The Tendon Profile can be defined in many ways such as defining the inflection points, but this example uses a common approach often used in practice, using the Tendon ordinates from drawings.

Referring to the values in the attached Excel file (*TD profile.xls*), prepared on the basis of the tendon drawings the ordinates of the tendon at every 2m are pasted into the software.

Load / Temp./Prestress / ***Tendon Profile***
Tendon Name **(A1L)** **;** Group **(A1)**
Tendon Property> **Tendon** **;** Assigned Elements **(1to20)**
Input Type> **3-D** **;** Curve Type> **Spline**
Profile
1> x (0), y (0), z (-1)
2> x (2), y (0), z (-1.2590)
…
25> x (48), y (0), z (-1.25)
Profile Inserton Point> **End-I of Elem.1**
x Axis Direction> **I->J of Elem.1** **;** x Axis Rot. Angle **(-11.3)**
Offset y : **(2.666)**

Copy & Paste the values from the Excel file to enter the Profile. We may also copy the Profile after creating an MCT file.

Transfer Length may be specified to consider the unstressed length of the anchorage.

Checking on "Typical Tendon" and entering the number of tendons can be used to represent a number of tendons of the same profile. This is also handy when preliminary analysis is

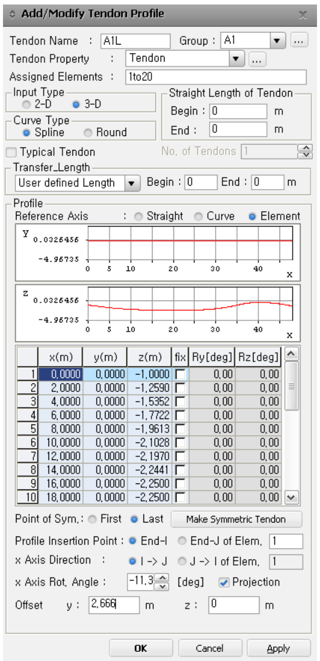


Figure 25. Tendon Profile Input Dialog

From the tendon profile drawings, x-z coordinates are obtained at every 2m. The result (TD Profile.xls) contains the values as if the tendons were placed in the centroidal 2-D plane, each side. We need to translate the layout using y-Offset and rotate the layout using x-Rotation to properly position them in the webs of the PSC section.

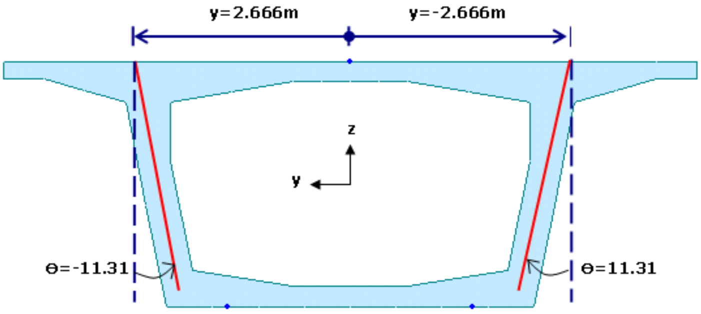
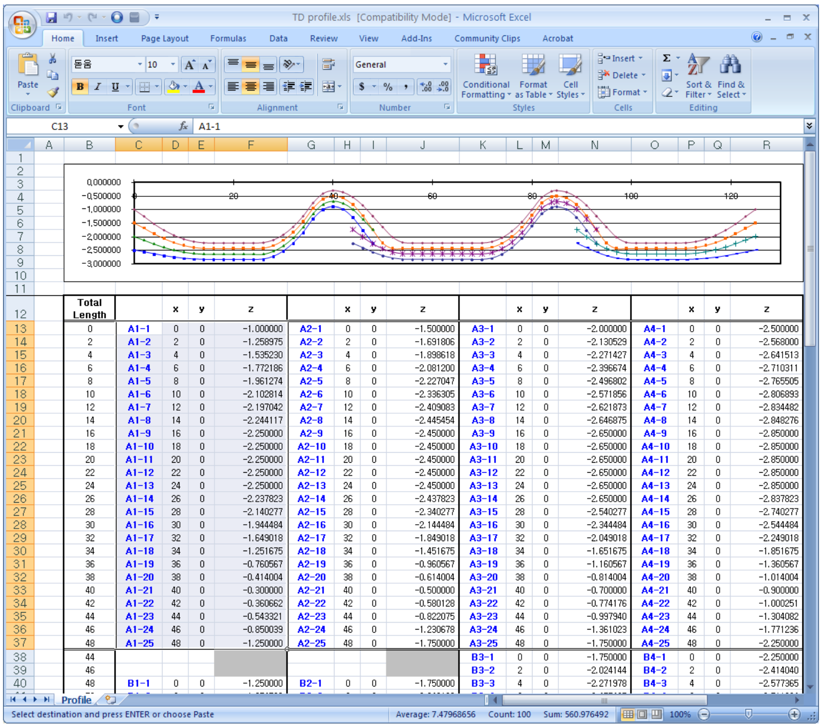


Figure 26. 3-Dimentional Tendon Profile Input

Copy and paste the values of x, y and z from the Excel file as below, and position the tendons in the webs by y-Offset and x-Rotation depending on the "left" or "right" tendon.



Total Length		x	y	z		x	y	z		x	y	z		x	y	z
0	A1-1	0	0	-1.000000	A2-1	0	0	-1.500000	A3-1	0	0	-2.000000	A4-1	0	0	-2.500000
2	A1-2	2	0	-1.258975	A2-2	2	0	-1.691806	A3-2	2	0	-2.130529	A4-2	2	0	-2.568000
4	A1-3	4	0	-1.535230	A2-3	4	0	-1.898618	A3-3	4	0	-2.271427	A4-3	4	0	-2.641513
6	A1-4	6	0	-1.772186	A2-4	6	0	-2.081200	A3-4	6	0	-2.396674	A4-4	6	0	-2.710311
8	A1-5	8	0	-1.961274	A2-5	8	0	-2.227047	A3-5	8	0	-2.496802	A4-5	8	0	-2.765505
10	A1-6	10	0	-2.102814	A2-6	10	0	-2.336305	A3-6	10	0	-2.571856	A4-6	10	0	-2.806893
12	A1-7	12	0	-2.197042	A2-7	12	0	-2.409083	A3-7	12	0	-2.621873	A4-7	12	0	-2.834482
14	A1-8	14	0	-2.244117	A2-8	14	0	-2.445454	A3-8	14	0	-2.646875	A4-8	14	0	-2.848276
16	A1-9	16	0	-2.250000	A2-9	16	0	-2.450000	A3-9	16	0	-2.650000	A4-9	16	0	-2.850000
18	A1-10	18	0	-2.250000	A2-10	18	0	-2.450000	A3-10	18	0	-2.650000	A4-10	18	0	-2.850000
20	A1-11	20	0	-2.250000	A2-11	20	0	-2.450000	A3-11	20	0	-2.650000	A4-11	20	0	-2.850000
22	A1-12	22	0	-2.250000	A2-12	22	0	-2.450000	A3-12	22	0	-2.650000	A4-12	22	0	-2.850000
24	A1-13	24	0	-2.250000	A2-13	24	0	-2.450000	A3-13	24	0	-2.650000	A4-13	24	0	-2.850000
26	A1-14	26	0	-2.237823	A2-14	26	0	-2.437823	A3-14	26	0	-2.650000	A4-14	26	0	-2.837823
28	A1-15	28	0	-2.140277	A2-15	28	0	-2.340277	A3-15	28	0	-2.540277	A4-15	28	0	-2.740277
30	A1-16	30	0	-1.944484	A2-16	30	0	-2.144484	A3-16	30	0	-2.344484	A4-16	30	0	-2.544484
32	A1-17	32	0	-1.649018	A2-17	32	0	-1.849018	A3-17	32	0	-2.049018	A4-17	32	0	-2.249018
34	A1-18	34	0	-1.251675	A2-18	34	0	-1.451675	A3-18	34	0	-1.651675	A4-18	34	0	-1.851675
36	A1-19	36	0	-0.760567	A2-19	36	0	-0.960567	A3-19	36	0	-1.160567	A4-19	36	0	-1.360567
38	A1-20	38	0	-0.414004	A2-20	38	0	-0.614004	A3-20	38	0	-0.814004	A4-20	38	0	-1.014004
40	A1-21	40	0	-0.300000	A2-21	40	0	-0.500000	A3-21	40	0	-0.700000	A4-21	40	0	-0.900000
42	A1-22	42	0	-0.360662	A2-22	42	0	-0.580128	A3-22	42	0	-0.774176	A4-22	42	0	-1.000251
44	A1-23	44	0	-0.543321	A2-23	44	0	-0.822075	A3-23	44	0	-0.997940	A4-23	44	0	-1.304082
46	A1-24	46	0	-0.850039	A2-24	46	0	-1.230678	A3-24	46	0	-1.361023	A4-24	46	0	-1.771236
48	A1-25	48	0	-1.250000	A2-25	48	0	-1.750000	A3-25	48	0	-1.750000	A4-25	48	0	-2.250000
44									B3-1	0	0	-1.750000	B4-1	0	0	-2.250000
46									B3-2	2	0	-2.024144	B4-2	2	0	-2.414040
48	B1-1	0	0	-1.250000	B2-1	0	0	-1.750000	B3-3	4	0	-2.271978	B4-3	4	0	-2.577365

The Name and the Assigned Elements for all Tendon Profiles are as follows:

Ex) A1L ➔ X coordinate (A, B, C), Z coordinate (1, 2, 3, 4), Y coordinate (Left, Right)

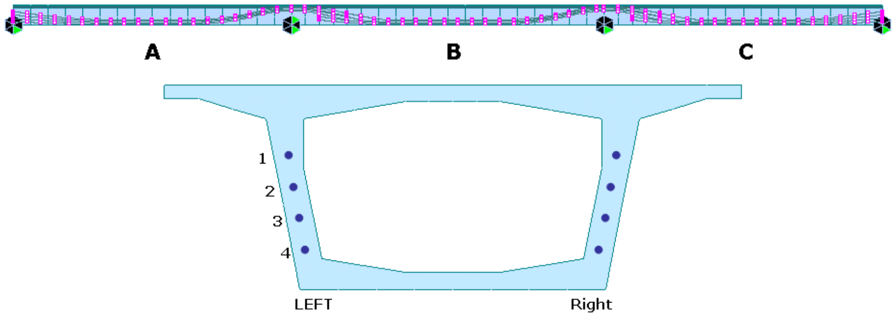


Figure 27.   Name Assignment for Tendon Profile

Tendon Profile	Assigned Element	Tendon Profile	Assigned Element
A1, A2	1 ~ 20	A3, A4	1 ~ 20
B1, B2	21 ~ 39	B3, B4	19 ~ 39
C1, C2	40 ~ 52	C3, C4	38 ~ 52

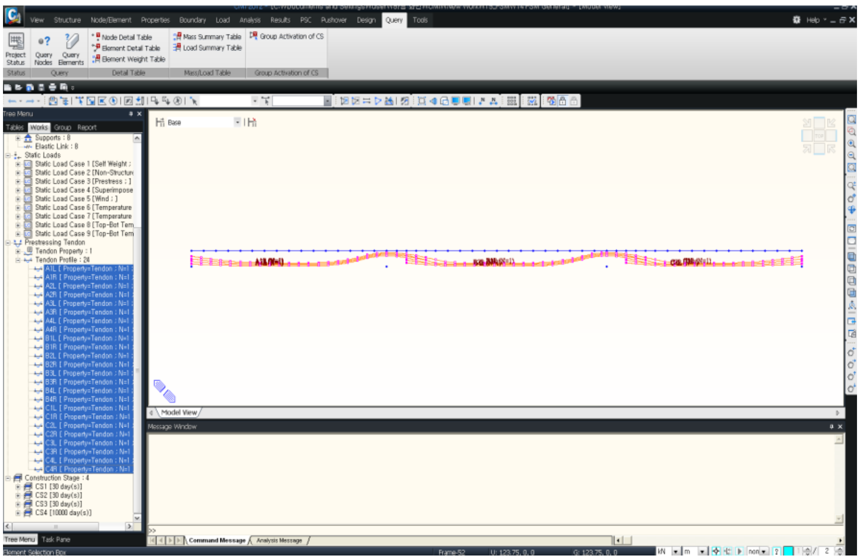


Figure 28.   Result of Tendon Profile Input

After defining all the Tendon Profiles, assign the Load Groups (PS1~3) and then apply prestress loads so that the defined Tendon Profiles can be applied to each construction stage.

Load / Temp.Prestress / ***Tendon Prestress Loads***
  Load Case Name> **Prestress**
  Load Group Name> **PS1**
  Select Tendon for Loading
    Tendon> **A1L~A4R** [>]
  Stress Value
    Begin **(1395000)** ; End **(0)**
  Grouting: after **(1)** Stage [Add] ↵

Prestress is applied one stage after the stage at which the load is entered.

  Load Group Name> **PS2**
  Select Tendon for Loading
    Selected> **A1L~A4R** [<] ; Tendon> **B1L~B4R** [>] [Add] ↵

  Load Group Name> **PS3**
  Select Tendon for Loading
    Selected> **B1L~B4R** [<] ; Tendon> **C1L~C4R** [>] [Add] ↵

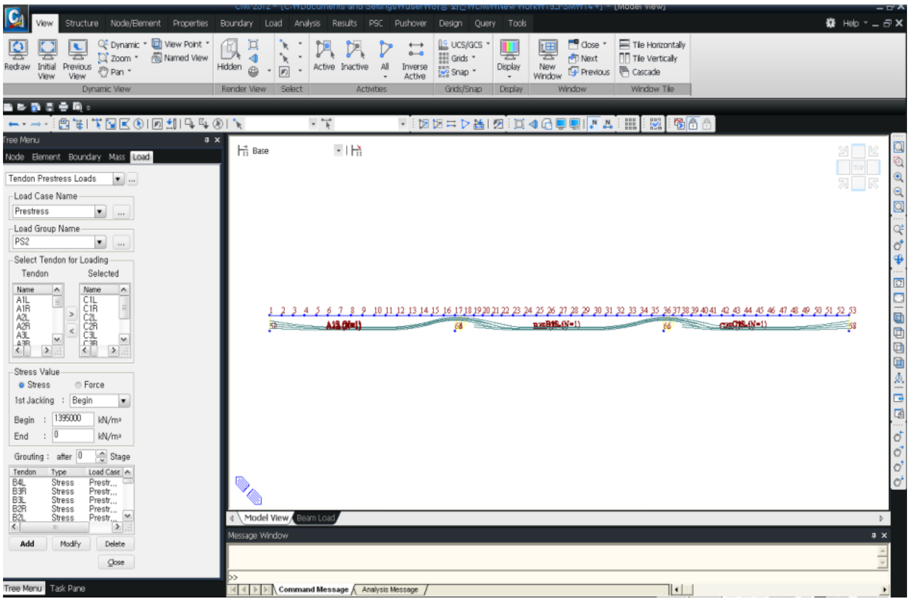


Figure 29. Loading Tendon Prestress

Superimposed Dead Loads

Superimposed Dead Loads are applied as Beam Load onto the superstructure.

Barriers	$(0.3075m^2 + 0.4975m^2) \times 24.52kN/m^3$	$19.74kN/m$
Safety Fences		$1kN/m$
Asphalt concrete pavement	$7.5m \times 8cm \times 22.56kN/m^3$	$13.536kN/m$
Noise barriers		$1.52kN/m$
Total		$35.796kN/m$

Load / Static Loads / Beam Loads/ ***Element***

Select All

Load Case Name>**Superimposed**

Load Group Name> **Superimposed dead**

Load Type>**Uniform Loads**

Value

Relative ; x1(**0**) ; x2 (**1**) ; w(**-35.796**) Apply ↵

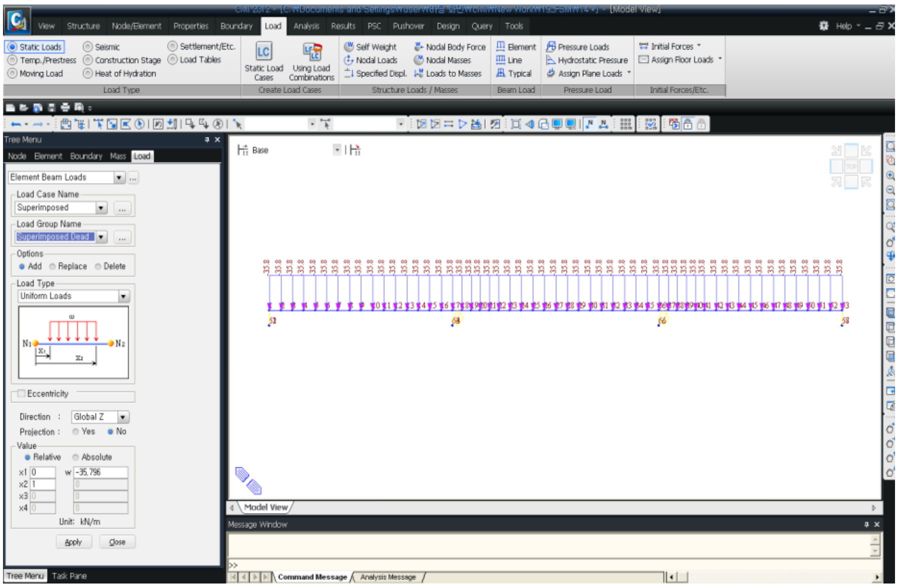


Figure 30. Loading Superimposed dead Loads

Loading Input on the Completed Structure

Wind Loading

wind loading of 3 kN/m2

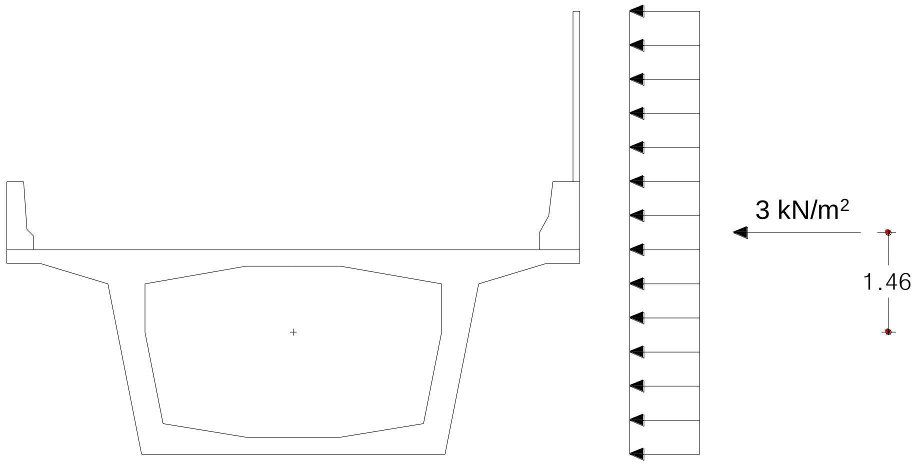


Figure 31. Wind Load Distribution

Total Height = Section Depth + Barriers + Noise barriers = 3 + 1 + 2.5 = 6.5m
Wind Pressure = $3kN/m^2$
Wind Load = $6.5m \times 3kN/m^2 = 19.5kN/m$ (Horizontal Load)
= $19.5kN/m \times -1.46m = -28.47kN \cdot m/m$ (Eccentricity Moment)

Loading pertaining to the Load Groups, which are not activated during the construction stages, are loaded in PostCS.

Enter the wind loads.

Load / Static Loads/ Beam Loads / ***Element***

Select All ***Iso View***

Load Case Name>**Wind**

Load Group Name>**Default**

Load Type>**Uniform Loads**

Direction>**Global Y**

Value

Relative

x1(**0**) ; x2 (**1**) ; w(**19.5**) Apply ↵

Select All

Load Type>**Uniform Moments/Torsion**

Direction>**Global X**

Value

Relative

x1(**0**) ; x2 (**1**) ; w(**-28.47**) Apply ↵

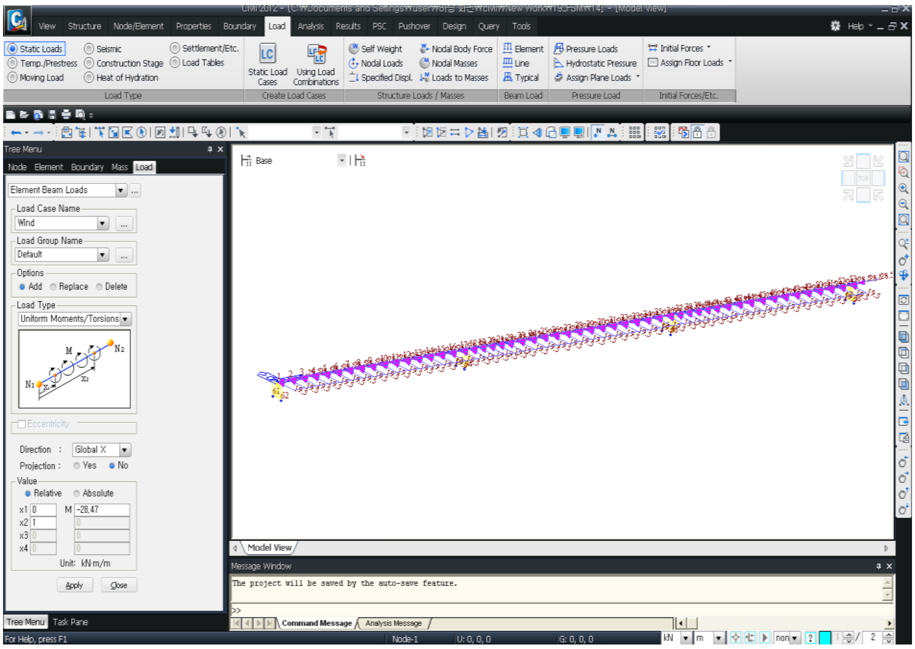


Figure 32. Wind Loading Input

Temperature

Specify the temperature loading acting on the entire structure.

The System Temperature function allows us to specify strain, $\varepsilon_t = \alpha(T_2 - T_1)$, over the entire structure as temperature loads.

Load / Temp./Prestress / Temperature Loads / ***System Temp.***

Redraw
Load Case Name>**Temperature (+)**
Load Group Name>**Default**
Temperature > Final Temperature **(15)** Add ↵

Load Case Name>**Temperature (-)**
Load Group Name>**Default**
Temperature > Final Temperature **(-15)** Add ↵

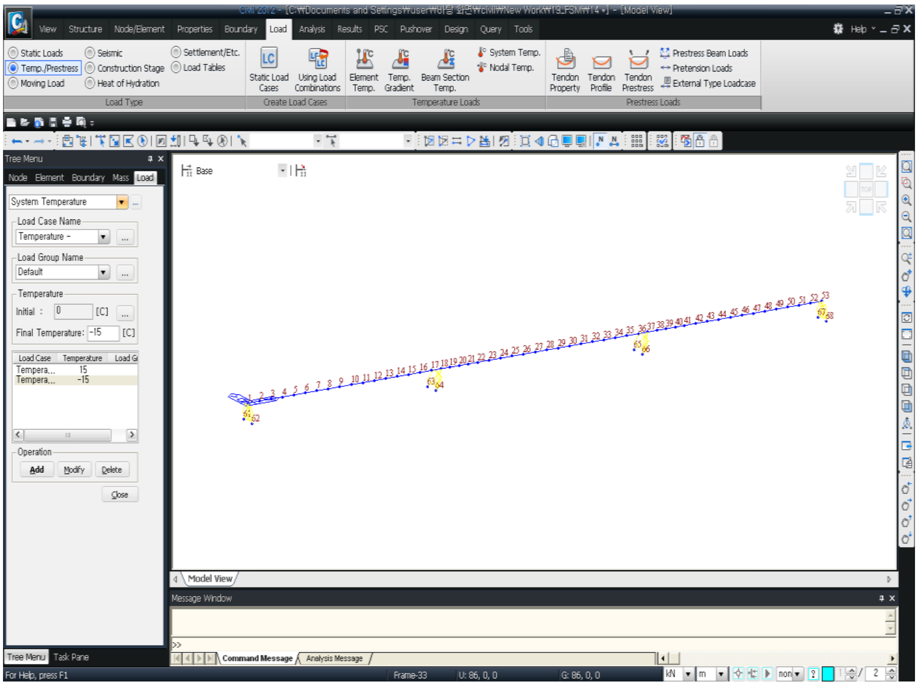
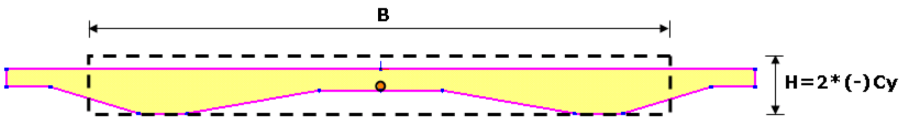


Figure 33. Temperature Loading Input

Specify the differential temperature between the top and bottom chords.

The Beam Section Temperature function generates a temperature differential between top and bottom chords on a part of a rectangle. Since PSC sections are not rectangular sections, they need to be converted into equivalent rectangular sections to be able to specify temperature differential loads.

Where temperature differentials exist as shown below, the parts experiencing the temperature differentials are converted into a rectangle defined by dotted lines having the same area and centroid.



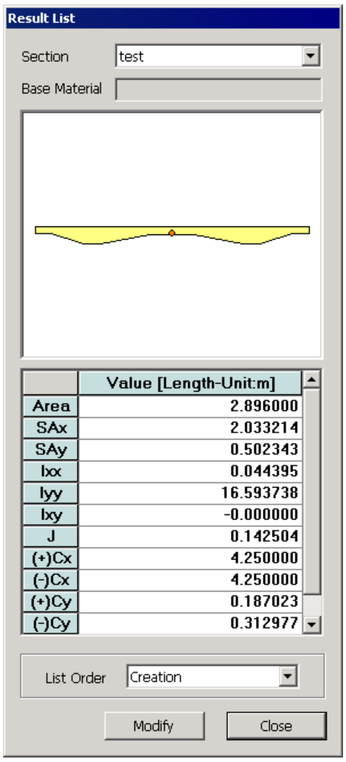


Figure 34. Section Properties calculated by SPC

Beam Section Temperature can be defined as either General Type or PSC Type. General Type assumes the section as a rectangle. When PSC Type is specified, the sections defined as PSC Type in defining Section Data are automatically converted into rectangles and loaded on the parts experiencing temperature differentials.

Although the Beam Section is defined as PSC Type in this example, which results in a simple input process for loading for a temperature differential between the top and bottom chords, input is carried out as General Type after converting into a rectangle.

Figure 34 shows the calculations for cross sectional area and centroid of the top part of the PSC Box section using SPC (Section Property Calculator). The instruction for using SPC is separately documented in user’s manual.

Using the above calculation results in conversion into an equivalent rectangle, which will be loaded, as follows:

Area = $2.896m^2$

H = $2 \times 0.312977m = 0.625954m$

$$\therefore B = \frac{Area}{H} = \frac{2.896}{0.625954} = 4.626m$$

Load / Temp./Prestress / Temperature Loads / ***Beam Section Temp.***

Load Case Name> **Top-Bot Temp Diff (+)**
Load Group Name>**Default**
Direction > **Local-z** ; Ref. Position > **Centroid**
B **(4.626)** ; H1 **(0.71)** ; H2 **(1.336)** ; T1 **(5)** ; T2 **(5)** Add ↵
Select All Apply ↵
Delete the defined Section Temperatures (select ①and delete)
Load Case Name> **Top-Bot Temp Diff (-)**
B **(4.626)** ; H1 **(0.71)** ; H2 **(1.336)** ; T1 **(-5)** ; T2 **(-5)** Add ↵
Select All Apply ↵

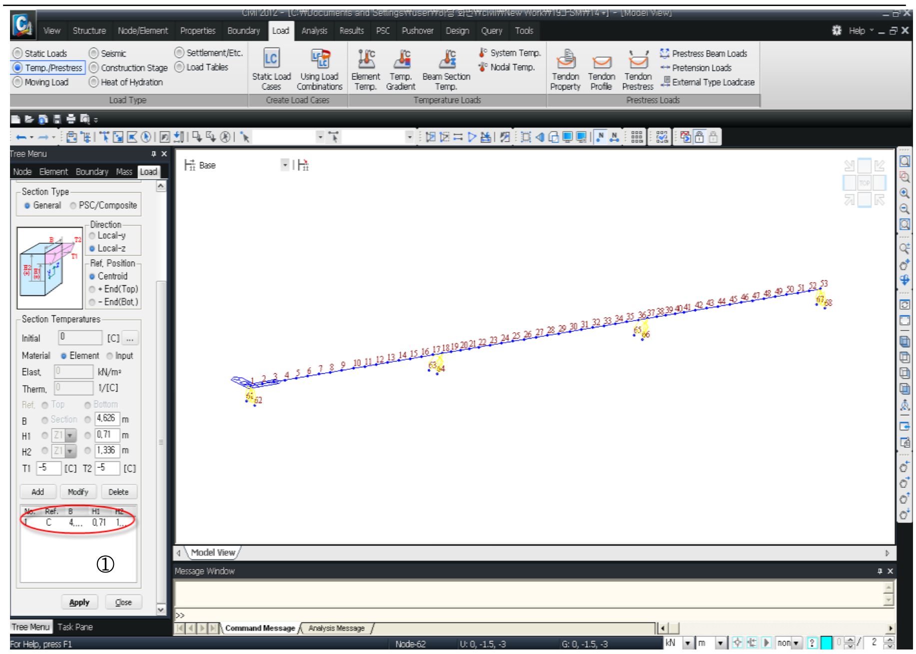


Figure 35. Input for Temperature Differential between Top & Bottom Chords

Live Load

The sequence of defining the live load is as follows:

Select a Code defining live load: Define Moving Load Code
Define lanes: Traffic Line Lanes
Define vehicles: Vehicles
Define live load cases: Moving Load Cases

▶ Select a Code, which specifies live load

The input process and the parameters are tailored to the selected Code.

Load / Moving Load Load Type / ***Moving Load Code***

Moving Load Code>**BS**

▶ **Define traffic lanes**

Eccentric and symmetrical loading can be considered for the transverse position of traffic lanes. In this tutorial, we specify only a symmetrical loading case as described below.

The eccentricity is positive (+) if the traffic lane (center) is on the right side of the elements in the direction of traffic, and vice versa.

Since this example bridge is straight and symmetrical, only the wind loading in the +Y direction has been applied. For the worst condition, only the eccentric live load in the +Y direction is entered.

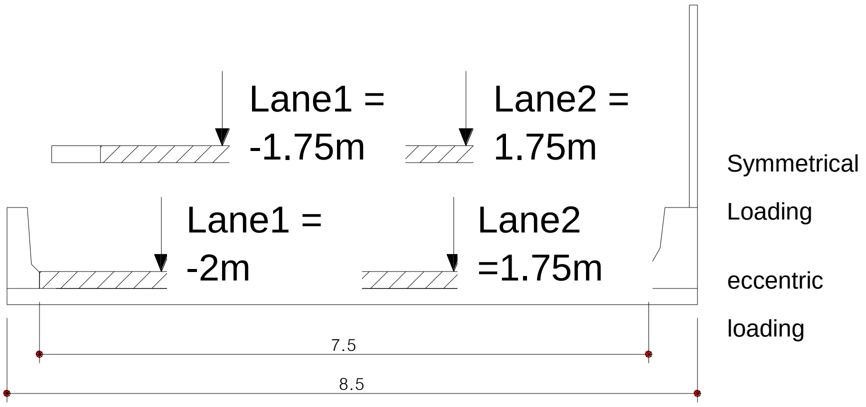


Figure 36. Traffic Lanes & Eccentricities

Refer to the Figure 36 for the traffic lanes and eccentricities to define 2 traffic lanes.

Top View

Load / Moving Load / ***Traffic Line Lanes***

Click Add ↵
Lane Name **(Lane 1 left)**
Traffic Lane Properties
Eccentricity **(-1.75)** ; Wheel Spacing **(1)** ; Lane Width**(3.5)**
Vehicular Load Distribution > **Lane Element**
Selection by > **2 Points ((0, 0, 0)(125, 0, 0))**
Click **OK**

When a traffic lane is curved or when the lane data entry with 2 Points becomes awkward due to discontinuity, select "Number" and directly type in the element numbers. (In this case, even if you select "Number" and input "1 to 53", the same traffic lanes are selected)

Click Add ↵
Lane Name **(Lane 2 right)**
Traffic Lane Properties
Eccentricity **(1.75)** ; Wheel Spacing **(1)** ; Lane Width**(3.5)**
Vehicular Load Distribution > **Lane Element**
Selection by > **2 Points ((0, 0, 0)(125, 0, 0))**
Click **OK**

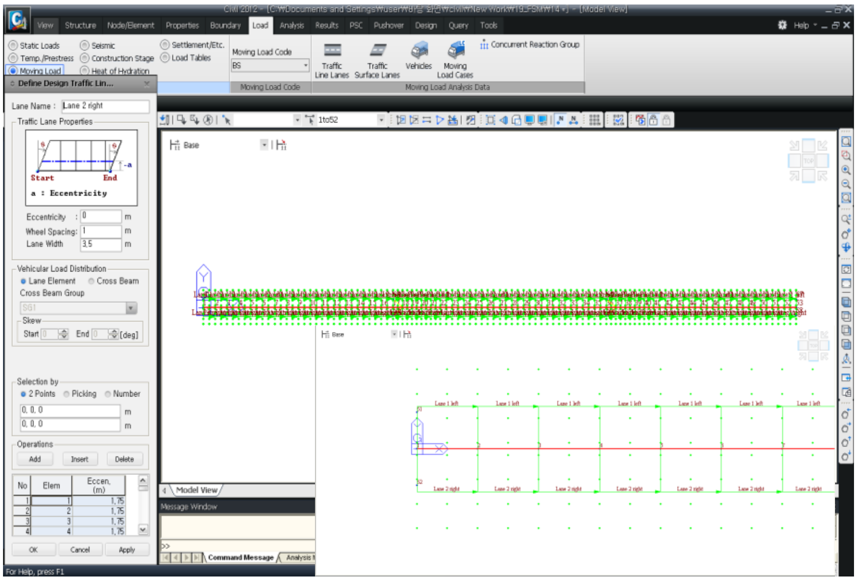


Figure 37. Traffic Lane Input Dialog & Input Result

▶ Definition of Vehicle Loads

Define the vehicles for live loads.

Load / Moving Load Analysis Data / ***Vehicles***

Standard Name > **BD 37/01 Standard Load**
Vehicular Load Name > **HA & HB(Auto)**
Vehicular Load Type > **HA & HB(Auto)** OK ↵

MIDAS/Civil contains the standard vehicle loads such as BS 5400, BS BD 37/01, AASHTO Standard, AASHTO LRFD, Caltrans, etc.

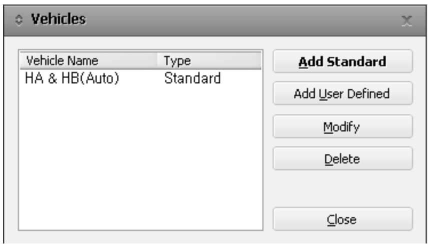


Figure 38. Definition of Vehicle Loads

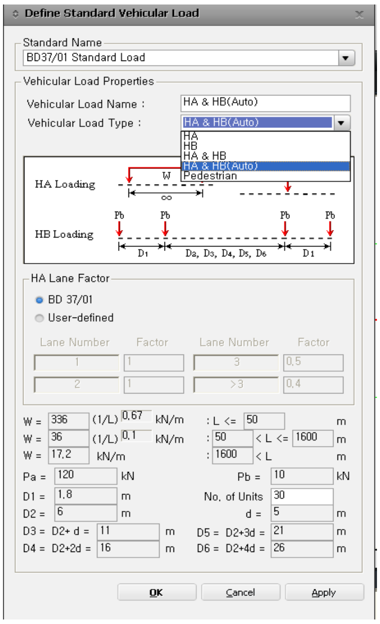


Figure 39. Definition of BD37/01 Standard Vehicular Load

▶ **Conditions for applying live loads**

To consider Load Cases, which combines the effects of HA and HB vehicle, Load case name MV U 1, MV U 2 3, MV S 1 and MV S 2 3 are created as below.

Load factors for HA loading for ULS, SLS, Combination 1 and Combinations 2 & 3 are taken from Section 6.2.7 of BD 37/01. Load factors for HB loading for ULS, SLS, Combination 1 and Combinations 2 & 3 are taken from Section 6.3.4 of BD 37/01. These load factors are automatically incorporated into moving load analysis results. Therefore, to avoid duplication, the user should not apply the load factors for moving loads while generating the Load Combinations.

Load Case Name		Type of Design Combination Factor	
		Ultimate Limit State	Serviceability Limit State
Combination of Loads	Combination 1	MV U 1	MV S 1
	Combination 2 & 3	MV U 2 3	MV S 2 3

Table 1. Definition of Load Case Name

Load / Moving Load Analysis Data / ***Moving Load Cases***

Click Add ↵

Load Case Name **(MV U 1)**
Check on **Auto Live Load Combination**
Type of Design Combination Factor>**Ultimate Limit State**
Combination of Loads>**Combination 1** Add ↵

Load Case Data
Scale Factor field **(1)** ; Number of Loaded Lanes **(2)**
Vehicle>**HA & HB (Auto)**

Assignment Lanes
List of Lanes **(Lane 1 left, Lane 2 right)** -> **Selected Lanes**
OK ↵

Apply ↵

Load Case Name **(MV U 2 3)**
Type of Design Combination Factor>**Ultimate Limit State**
Combination of Loads>**Combination 2 or 3** Apply ↵

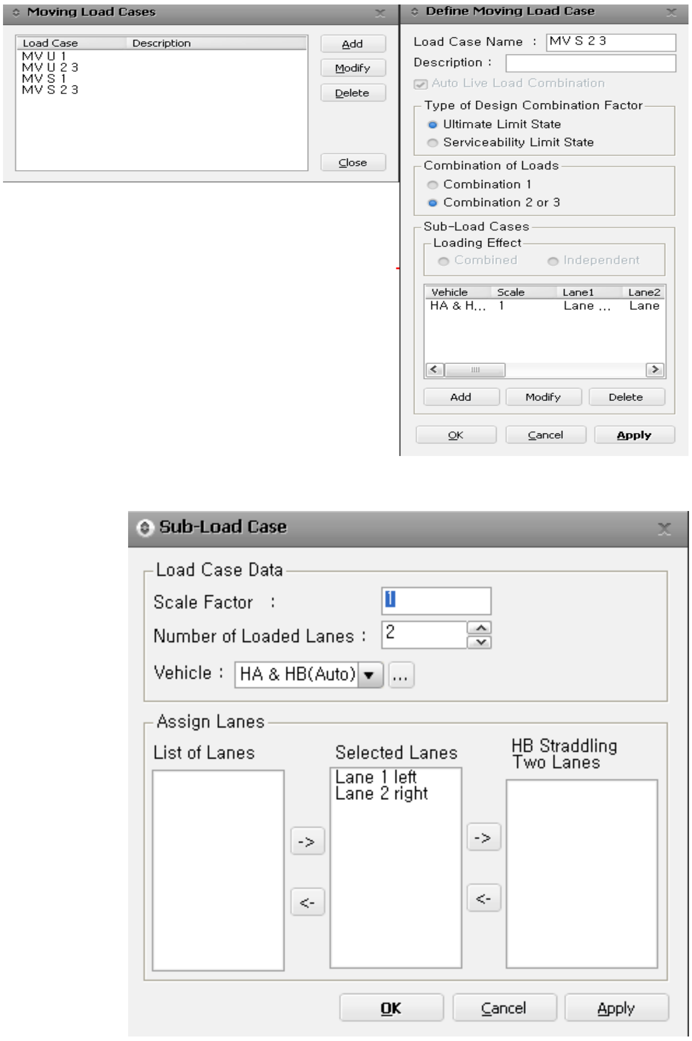


Figure 40. Definition of Live Load

Differential Settlement

▶ Definition of Differential Settlement Groups

Select the nodes, which can settle simultaneously, representing the abutments and piers, to individually define them as a Settlement Group.

Load / Settlement/Etc. / ***Settlement Group***
Group Name > **A1** ; Settlement Displacement **(-0.01)**
Select By Window (**61, 62**) Add ↵
Group Name > **P1** ; Settlement Displacement **(-0.01)**
Select By Window (**63, 64**) Add ↵
Group Name > **P2** ; Settlement Displacement **(-0.01)**
Select By Window (**65, 66**) Add ↵
Group Name > **A2** ; Settlement Displacement **(-0.01)**
Select By Window (**67, 68**) Add ↵

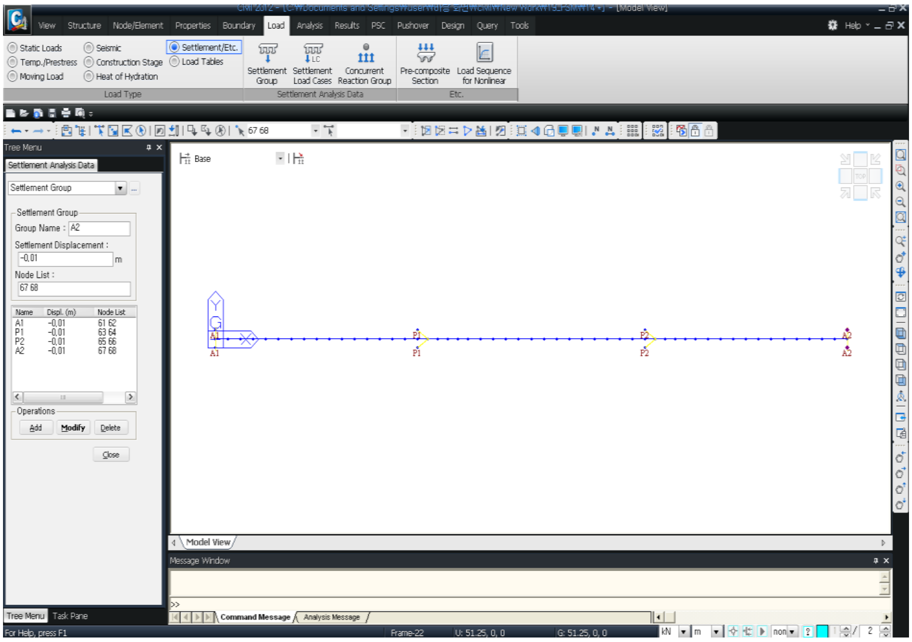


Figure 41. Definition of Differential Settlement Groups

▶ Conditions for Differential Settlement Loads

Using the data for differential settlement groups, the loading condition is defined.

Maximum/Minimum numbers of differential settlement groups are specified. Min: 1 support and Max: 3 supports are specified to investigate all the possible combinations of simultaneous settlements from which Min/Max results are produced.

Load / Settlement/Etc. / ***Settlement Load Case***

Load Case Name **(SM)**

Select Settlement Group

Settlement Group (**A1, P1, P2, A2)** -> **Selected Group**

Smin **(1)** ; Smax **(3)** Add ↵

Since the magnitude of the settlements of all 4 groups is identical, only a maximum of 3 combinations is used.

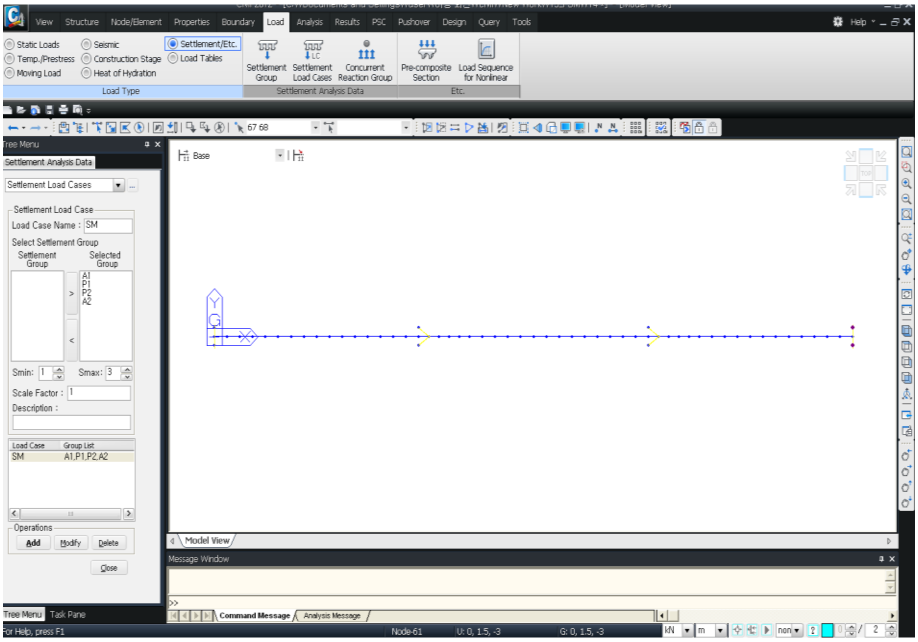


Figure 42. Definition of Loading Conditions for Differential Settlements

Definition of Construction Stages

We refer to the composition of construction stages outlined earlier to define the stages.

Load / Construction Stage / ***Define C.S***

Add ↵

Name> **CS1**
Duration>**30**

Element tab

Group List>**SG1** ; Activation>Age **(5)** Add

Boundary tab

Group List>**BG1**

Activation>Spring/Support Position>**Deformed** (on) Add

Load tab

Group List>**Dead, PS1, Diaphragm1**

Activation>Active Day>**First** Add ↵

Concrete maturity (age) of 5 days is activated.

Stage (days)		**Element**	**Boundary**	**Load**
CS1	30	SG1	BG1	Dead, PS1, Diaphragm1
CS2	30	SG2	BG2	PS2, Diaphragm2
CS3	30	SG3	BG3	PS3, Diaphragm3
CS4	10,000	-	-	Superimposed dead

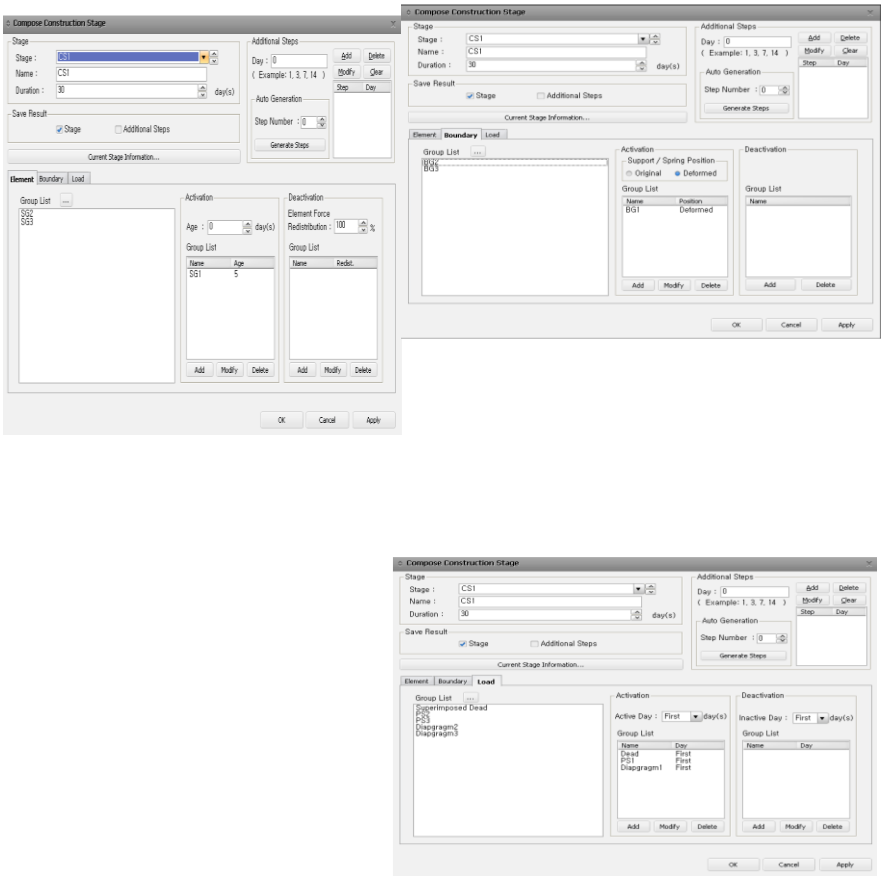


Figure 43.   Dialog Boxes for defining Construction Stages

Performing Structural Analysis

Select the analysis options for construction stage analysis and moving load analysis and perform analysis.

Construction Stage Analysis

All the dead loads applied during the construction stages are included in CS:Dead Load. If results for other Load Cases need to be separated from CS:Dead Load, such Load Cases need to be selected in “Load Cases to be Distinguished from Dead Load for C.S. Output”. Separate results are then produced in CS:Erection Load.

Analysis / Construction Stage Analysis Control

Load Cases to be Distinguished from Dead Load for C.S. Output
Load Case>**Superimposed** Add
Save Output of Current Stage(Beam/Truss) (on) ↵

Check on “Save Output of Current Stage (Beam /Truss)” to produce member forces generated only from each (current) stage. That is, not the member forces accumulated up to that (current) stage.

Checking on “Change with Tendon” in “Beam Section Property Change” will reflect the effect of tendons for calculating section properties by construction stages.

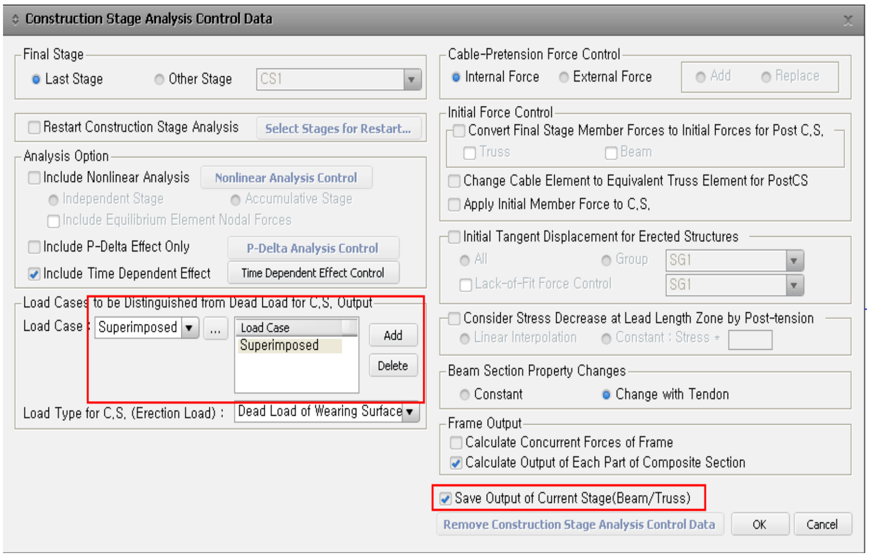


Figure 44. Construction Stage Analysis Control Data

Moving Load (Live Load) Analysis

Select the method of influence line calculation and the options for generation of analysis results.

Analysis / Moving Load Analysis Control

Influence Generating Method > Number/Line Element **(2)**
Analysis Results
Frame>**Normal + Concurrent Force**
Combined Stress Calculation (off) ↵

Specify the number of points per beam element on which influence line is calculated. A number between 1 to 10 can be specified.

"Concurrent Force" will generate member forces, which take place simultaneously under the same loading.

Check on "Combined Stress" to generate combined stress results.

A substantial amount of results are generated from moving load analysis. Only the desired parts should be selected in groups for output generation.

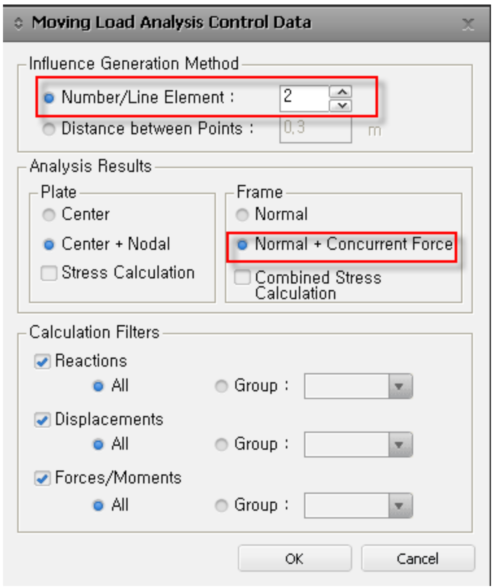


Figure 45. Moving Load Analysis Control Dialog

Execution of Structural Analysis

We have completed the process of structural modeling and defining the analysis options, so analysis can begin now.

Analysis / Perform Analysis

Checking Analysis Results

Construction stage analysis results will be reviewed via the versatile functionality of midas Civil.

Element Properties & Section Properties for each Construction Stage

The properties of each element used during the construction stages are produced in a table.

Select a stage to see the corresponding data; initial (Start) age, final (End) age, initial (Start) modulus of elasticity, final (End) modulus of elasticity, shrinkage accumulated up to the end of the corresponding stage and creep coefficient.

When a construction stage is selected, only the results pertaining to the corresponding stage are produced. The Post CS construction stage is selected followed by pressing the Apply button to change the result values as below.

Results / Result Table / Construction Stage / Element Properties at Each Stage

PostCS (Post construction stage)

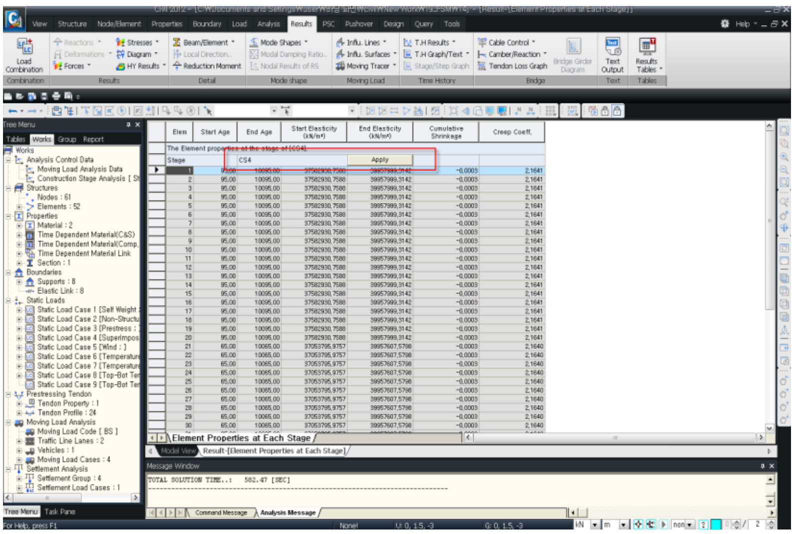
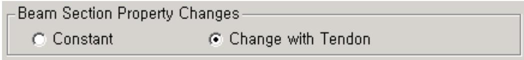


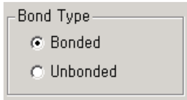
Figure 46. Element Properties at each Construction Stage

Transformed section properties used in the last stage of the construction stage analysis are produced in a table. The properties may change with change in modulus of elasticity (if a time dependent material is used). And if tendons are included in sections, the tendon properties and the timing of grouting will affect the section properties.

* In order to reflect the Tendon in section property calculations, "Change with Tendon" needs to be selected in Construction Stage Analysis Control.



* If "Change with Tendon" is selected, and "Bonded" type in "Tendon Property" is selected, the Tendon will be reflected in the section property calculations. Otherwise (in case of "Unbonded"), the Tendon is excluded and the net section is used in the calculations.



The section properties at the last stage are used for calculating stresses due to additional loads applied at the completed stage such as moving load, temperature load, wind load, etc.

Results / Result Table / Construction Stage / Beam Section Properties at Last Stage

---

In the *.out file, we can see the section properties for all the stages in addition to those for the final stage.

Elem	Part	Area (m^2)	Ixx (m^4)	Iyy (m^4)	Izz (m^4)	Cyp (m)	Cym (m)	Czp (m)	Czm (m)	WArea (m^2)	Translational Distance Local-y (m)	Translational Distance Local-z (m)
1	I	6,2587	15,9796	7,8969	29,8433	4,2500	4,2500	1,2137	1,7863	6,2087	0,0000	−0,0043
1	J	6,2587	15,9796	7,9055	29,8336	4,2500	4,2500	1,2154	1,7846	6,2087	0,0000	−0,0060
2	I	6,2587	15,9796	7,9055	29,8336	4,2500	4,2500	1,2154	1,7846	6,2087	0,0000	−0,0060
2	J	6,2587	15,9796	7,9195	29,8239	4,2500	4,2500	1,2171	1,7829	6,2087	0,0000	−0,0077
3	I	6,2587	15,9796	7,9195	29,8239	4,2500	4,2500	1,2171	1,7829	6,2087	0,0000	−0,0077
3	J	6,2587	15,9796	7,9345	29,8164	4,2500	4,2500	1,2184	1,7816	6,2087	0,0000	−0,0090
4	I	6,2587	15,9796	7,9345	29,8164	4,2500	4,2500	1,2184	1,7816	6,2087	0,0000	−0,0090
4	J	6,2587	15,9796	7,9472	29,8111	4,2500	4,2500	1,2194	1,7806	6,2087	0,0000	−0,0099
5	I	6,2587	15,9796	7,9472	29,8111	4,2500	4,2500	1,2194	1,7806	6,2087	0,0000	−0,0099
5	J	6,2587	15,9796	7,9556	29,8080	4,2500	4,2500	1,2199	1,7801	6,2087	0,0000	−0,0105
6	I	6,2587	15,9796	7,9556	29,8080	4,2500	4,2500	1,2199	1,7801	6,2087	0,0000	−0,0105
6	J	6,2587	15,9796	7,9585	29,8070	4,2500	4,2500	1,2201	1,7799	6,2087	0,0000	−0,0107
7	I	6,2587	15,9796	7,9585	29,8070	4,2500	4,2500	1,2201	1,7799	6,2087	0,0000	−0,0107
7	J	6,2587	15,9796	7,9585	29,8070	4,2500	4,2500	1,2201	1,7799	6,2087	0,0000	−0,0107
8	I	6,2587	15,9796	7,9585	29,8070	4,2500	4,2500	1,2201	1,7799	6,2087	0,0000	−0,0107
8	J	6,2587	15,9796	7,9585	29,8070	4,2500	4,2500	1,2201	1,7799	6,2087	0,0000	−0,0107
9	I	6,2587	15,9796	7,9585	29,8070	4,2500	4,2500	1,2201	1,7799	6,2087	0,0000	−0,0107
9	J	6,2587	15,9796	7,9584	29,8070	4,2500	4,2500	1,2201	1,7799	6,2087	0,0000	−0,0107
10	I	6,2587	15,9796	7,9584	29,8070	4,2500	4,2500	1,2201	1,7799	6,2087	0,0000	−0,0107
10	J	6,2587	15,9796	7,9587	29,8069	4,2500	4,2500	1,2201	1,7799	6,2087	0,0000	−0,0107
11	I	6,2587	15,9796	7,9587	29,8069	4,2500	4,2500	1,2201	1,7799	6,2087	0,0000	−0,0107
11	J	6,2587	15,9796	7,9488	29,8102	4,2500	4,2500	1,2195	1,7805	6,2087	0,0000	−0,0101
12	I	6,2587	15,9796	7,9488	29,8102	4,2500	4,2500	1,2195	1,7805	6,2087	0,0000	−0,0101
12	J	6,2587	15,9796	7,9225	29,8203	4,2500	4,2500	1,2177	1,7823	6,2087	0,0000	−0,0083
13	I	6,2587	15,9796	7,9225	29,8203	4,2500	4,2500	1,2177	1,7823	6,2087	0,0000	−0,0083
13	J	6,2587	15,9796	7,8904	29,8376	4,2500	4,2500	1,2146	1,7854	6,2087	0,0000	−0,0052
14	I	6,2587	15,9796	7,8904	29,8376	4,2500	4,2500	1,2146	1,7854	6,2087	0,0000	−0,0052
14	J	6,2587	15,9796	7,8698	29,8638	4,2500	4,2500	1,2102	1,7898	6,2087	0,0000	−0,0008
15	I	6,2587	15,9796	7,8698	29,8638	4,2500	4,2500	1,2102	1,7898	6,2087	0,0000	−0,0008
15	J	6,2587	15,9796	7,8785	29,8896	4,2500	4,2500	1,2060	1,7940	6,2087	0,0000	0,0034
16	I	6,2587	15,9796	7,8785	29,8896	4,2500	4,2500	1,2060	1,7940	6,2087	0,0000	0,0034
16	J	6,2587	15,9796	7,8877	29,8987	4,2500	4,2500	1,2045	1,7955	6,2087	0,0000	0,0049

Beam Section Properties at Last Stage

Model View / Result-[Element Properties at Each Stage] / Result-[Beam Section Properties at Last Stage]

Figure 47. Section Property Data at the Last Stage

Checking Construction Stage Member Forces & Stresses

Member forces can be checked in a diagram using the Beam Diagram function. If a beam element is selected after invoking Quick View, member forces at any particular point on the selected element can be checked in detail.

Results / Forces / Beam Diagrams…

CS4

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

Components>**My**

Display Options> **Solid Fill**

Type of Display

Contour (on) ; **Legend** (on) Apply ↵

Type of Display>**Quick View**

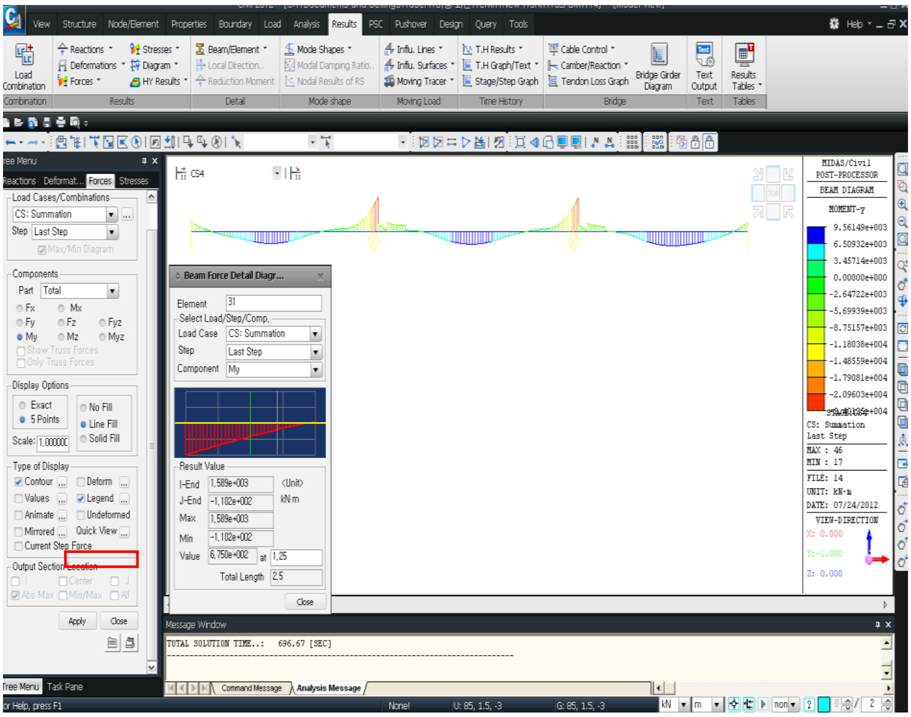


Figure 48. Checking Member Forces at CS4

Using the Beam Stresses(PSC) function, the stresses in a PSC section can be checked in a diagram. A total of 10 locations, Top/Bot vetices (1 to 4), Center (7 & 8) and shear checking points (5, 6, 9 & 10) defined at the time of defining the PSC section, can be checked.

Let us check the bottom chord stress for CS:Summation at the last construction stage.

Results / Stresses / Beam Stresses(PSC)…

CS4

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

Section Position>**Position 3**

Components>**Sig-xx(Summation)**

Type of Display

Contour (on) ; **Legend** (on)

Top-Bot chord stresses for each construction stage can be also checked in Bridge Girder Diagram. In case of a PSC section, Beam Stresses (PSC) can be used to check Contour in the Model View state.

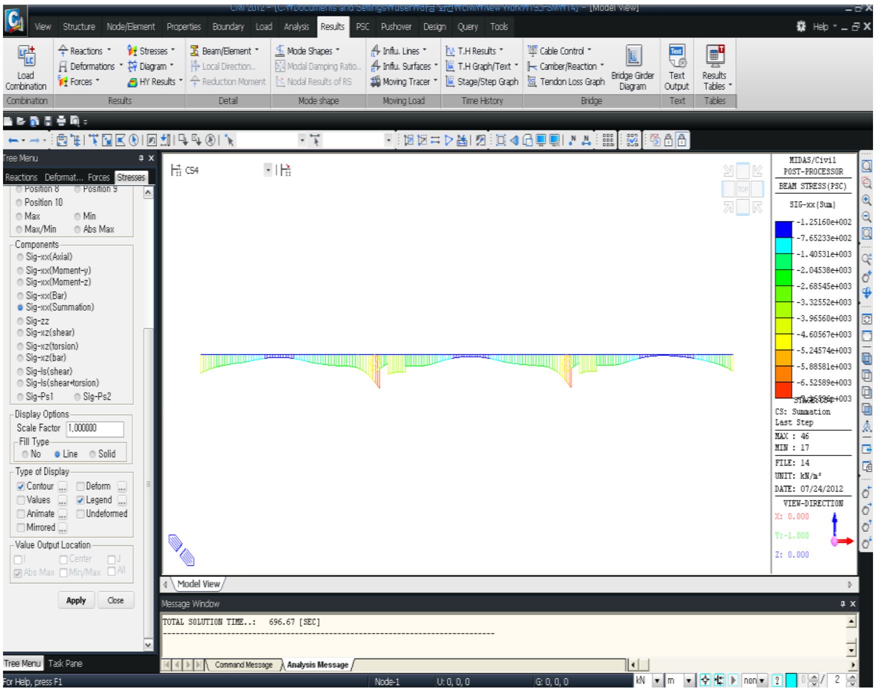


Figure 49. Bottom Chord Stresses at the Last Stage

Using User Defined Diagram, different results (displacements / member forces / stresses) for different elements/groups can be produced.

We will generate results for displacements in the left span, bending moments in the middle span and stresses in the right span in a single diagram simultaneously. Let us check displacements / member forces / stresses for CS:Summation at the last construction stage.

Combined results can be produced only in the same construction stage.

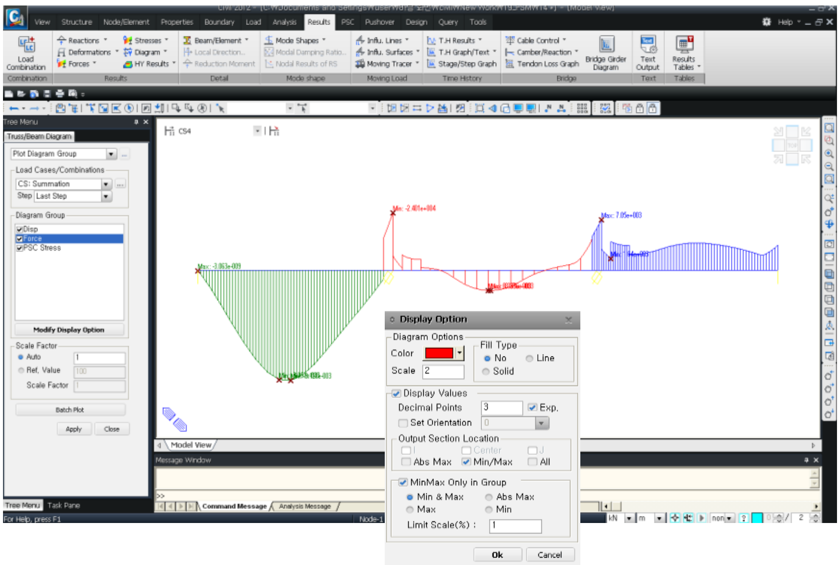
Output option can be selected in Modify Display Option

Results / Diagram / Define Diagram…

CS4

Element>**1to16** ; Type of Result>**Displacement**
Component>**DZ** ; Group Name>**Disp** Add ↵
Element>**17to35** ; Type of Result>**Beam Force/Moments**
Component>**My** ; Group Name>**Force** Add ↵
Element>**36to52** ; Type of Result>**Beam Stresses(PSC)**
Section Position>**Abs Max** ; Components>**Sig-xx(Summation)**
Group Name>**PSC Stress** Add ↵

Results / Diagram / Plot Diagram…
Load Cases/Combination>**CS: Summation**
Diagram Group>**Disp**(on), **Force**(on), **PSC Stress**(on) ↵



Note that the three plots above have different scale factors to properly display in this figure. In order to check the results, you may enlarge the figure and compare the values.

Figure 50. User Defined Diagram Output Display

Checking Results using Graphs

The change in stresses with the progress of construction stages in the support element (No.36) will be checked in a Graph.

Results / Stage/Step History Graph
CS4
Define Function>**Beam Force/Stress** [Add New Function] ↵
Name**(36_ax)** ; Element No.**(36)** ; **Stress**
Point>**I-Node** ; Components>**Axial** ↵

Name**(36_b(+y))** ; Components>**Bend(+y)** ↵
Name**(36_b(-y))** ; Components>**Bend(-y)** ↵
Name**(36_b(+z))** ; Components>**Bend(+z)** ↵
Name**(36_b(-z))** ; Components>**Bend(-z)** ↵

Mode >**Multi Func.**
Step Option>**All Steps**
Load Cases/Combinations>**Summation** ↵

If "Multi LCase" is selected, the results history of the component of the corresponding element for a number of Load Cases can be checked.

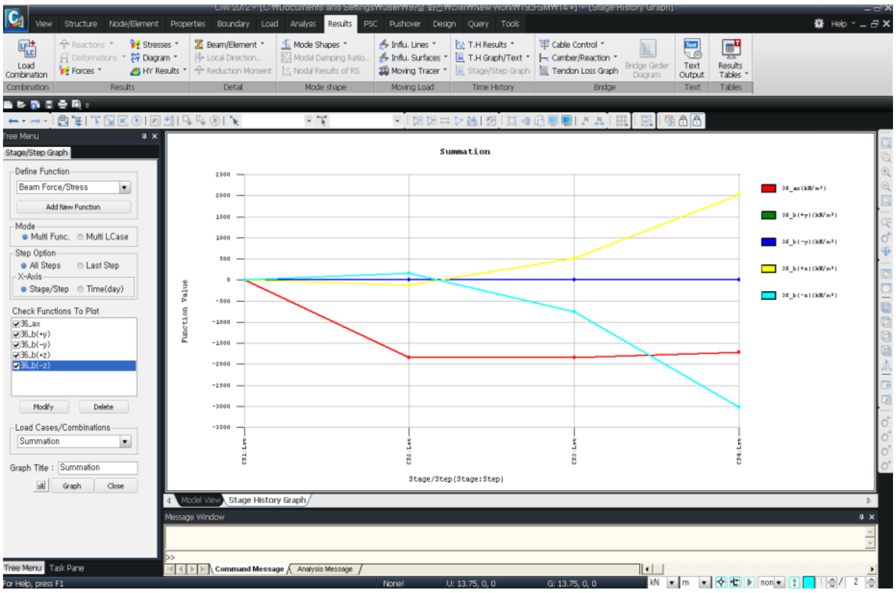


Figure 51. Change in Stresses with Construction Stages

Checking Results using Tables

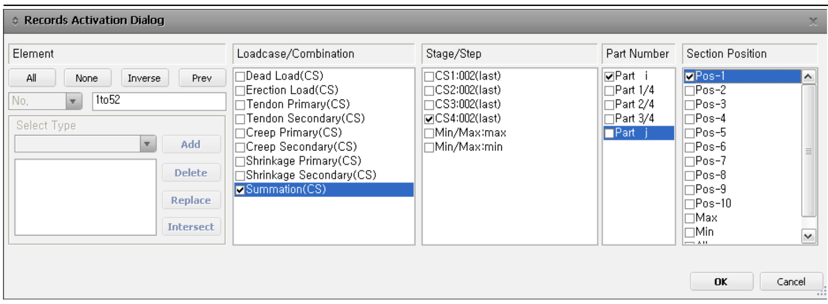
Tables are also useful in checking construction stage analysis results. Tables can be manipulated in various ways by right-clicking on the tables.

From “Records Activation Dialog”, tables can be generated by selecting elements to be checked for stresses, load cases, construction stages (steps), elements on which points of stress output are required, load cases, construction stages (steps), stress output locations on elements, stress output locations on a section, etc.

The Sorting Dialog allows us to sort/arrange the data based on the sorting criteria. The Style Dialog allows us to change the data type and produce results.

Let us check top vertex stresses for CS:Summation at the last construction stage.

Results / Result Tables / Beam / Stresses(PSC)



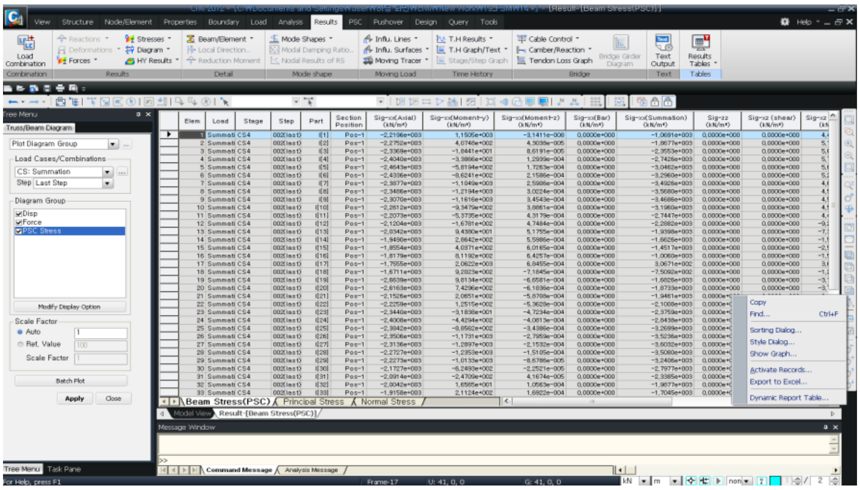
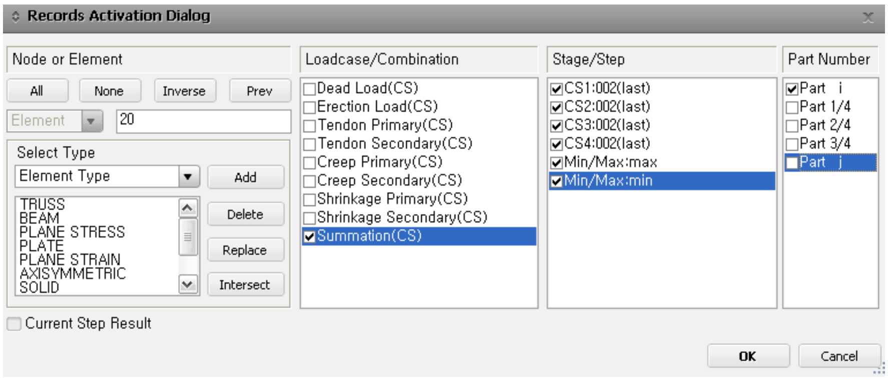


Figure 52. Checking Top Chord Stresses using Table

In “Construction Stage Analysis Control” dialog box, if “**Save Output of Current Stage (Beam/Truss)”** ☑ Save Output of Current Stage(Beam/Truss) option has been checked on, we can generate the member forces resulting only from the corresponding construction stage (not the member forces accumulated up to that stage). So in order to produce results for the un-accumulated effects of one given construction stage, check “Current Step Result” for all the stages.

Results / Result Tables / Beam / Force



Elem	Load	Stage	Step	Part	Axial (kN)	Shear-y (kN)	Shear-z (kN)	Torsion (kN·m)	Moment-y (kN·m)	Moment-z (kN·m)
20	Summati	CS1	002(last)	I[20]	-10296,16	0,00	1802,79	0,00	-1447,41	0,00
20	Summati	CS2	002(last)	I[20]	-17587,18	-0,00	253,98	0,00	-1650,80	-0,00
20	Summati	CS3	002(last)	I[20]	-17507,52	-0,00	319,25	0,00	-1514,36	-0,01
20	Summati	CS4	002(last)	I[20]	-16096,67	-0,00	-485,96	0,00	-4833,07	-0,00
20	Summati	Min/Max	max	I[20]	-10296,16	0,00	1848,67	0,00	-1083,72	0,00
20	Summati	Min/Max	min	I[20]	-17786,08	[illegible]	[illegible]	[illegible]	[illegible]	-0,01

Accumulated member forces at each construction stage

Elem	Load	Stage	Step	Part	Axial (kN)	Shear-y (kN)	Shear-z (kN)	Torsion (kN·m)	Moment-y (kN·m)	Moment-z (kN·m)
20	Summati	CS1	002(last)	I[20]	97,94	0,00	-21,12	0,00	10,11	0,00
20	Summati	CS2	002(last)	I[20]	68,91	-0,00	-15,28	0,00	-104,60	-0,00
20	Summati	CS3	002(last)	I[20]	41,75	-0,00	-4,99	0,00	-92,17	-0,00
20	Summati	CS4	002(last)	I[20]	125,97	0,00	-21,98	-0,00	-43,41	0,00
20	Summati	Min/Max	max	I[20]	0,00	0,00	0,00	-0,00	0,00	0,00
20	Summati	Min/Max	min	I[20]	0,00	[illegible]	[illegible]	[illegible]	[illegible]	0,00

Member forces solely due to the corresponding stage

Figure 53. Member Forces due to the sole effect of Current Stage (below)

Prestress Losses

We can check the change in tendon tension at each construction stage due to prestress losses.

In the "Tendon Time Dependent Loss Graph" dialog box, only the tendons included in the stage selected in the "Stage" selection window can be checked. A Graph is generated for selected tendons, selected Stage and selected Step. Click Animate to check the results in an animation.

Results / Tendon Time-dependent Loss Graph

Tendon>**A1L** Animate ↵

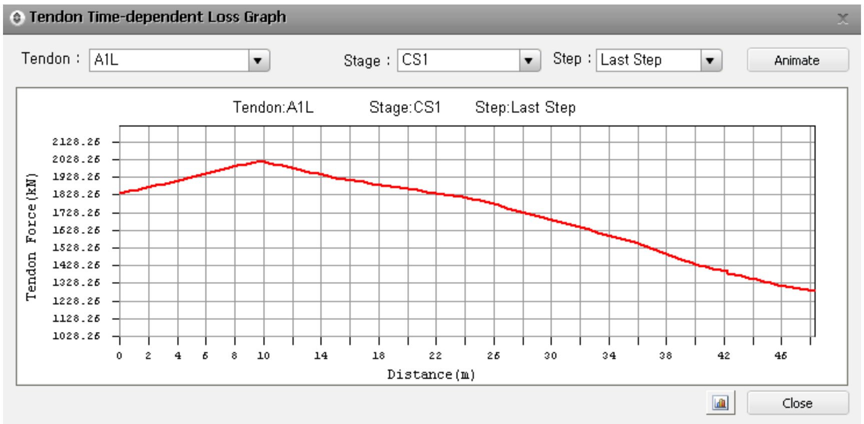


Figure 54. Graph showing Loss of Prestress Forces

Checking Tendon Information

The tendon information used in construction stage analysis can be produced in a table.

The coordinates of the tendons placed in elements are produced.

Results / Result Tables / Tendon / Tendon Coordinates…

Tendon Name	No	x (m)	y (m)	z (m)
A1L	0	0,0000	0,0000	0,0000
A1L	1	0,0000	2,4660	-1,0000
A1L	2	0,6250	2,4502	-1,0790
A1L	3	1,2500	2,4341	-1,1593
A1L	4	1,8750	2,4176	-1,2420
A1L	5	2,5000	2,4004	-1,3281
A1L	6	3,1250	2,3829	-1,4157
A1L	7	3,7500	2,3656	-1,5018
A1L	8	4,3750	2,3493	-1,5837
A1L	9	5,0000	2,3340	-1,6602
A1L	10	5,6250	2,3197	-1,7316
A1L	11	6,2500	2,3063	-1,7984
A1L	12	6,8750	2,2939	-1,8607

Figure 55. Tendon Coordinates Table

Elongation of tendons is produced. Timing of tensioning each tendon, elongation of tendons and elements at the start and end points of the tendons and their sum are produced.

Results / Result Tables / Tendon / Tendon Elongations…

Tendon Name	Stage	Step	Tendon Elongation Begin (m)	Tendon Elongation End (m)	Element Elongation Begin (m)	Element Elongation End (m)	Summation Begin (m)	Summation End (m)
A1L	CS1	001(first	0,2766	0,0000	0,0005	0,0000	0,2770	0,0000
A1R	CS1	001(first	0,2766	0,0000	0,0005	0,0000	0,2770	0,0000
A2L	CS1	001(first	0,2785	0,0000	0,0005	0,0000	0,2789	0,0000
A2R	CS1	001(first	0,2785	0,0000	0,0005	0,0000	0,2789	0,0000
A3L	CS1	001(first	0,2799	0,0000	0,0005	0,0000	0,2803	0,0000
A3R	CS1	001(first	0,2799	0,0000	0,0005	0,0000	0,2803	0,0000
A4L	CS1	001(first	0,2825	0,0000	0,0005	0,0000	0,2829	0,0000
A4R	CS1	001(first	0,2825	0,0000	0,0005	0,0000	0,2829	0,0000
B1L	CS2	001(first	0,2569	0,0000	0,0004	0,0000	0,2573	0,0000
B1R	CS2	001(first	0,2569	0,0000	0,0004	0,0000	0,2573	0,0000
B2L	CS2	001(first	0,2509	0,0000	0,0004	0,0000	0,2513	0,0000
B2R	CS2	001(first	0,2509	0,0000	0,0004	0,0000	0,2513	0,0000
B3L	CS2	001(first	0,2818	0,0000	0,0005	0,0000	0,2822	0,0000
B3R	CS2	001(first	0,2818	0,0000	0,0005	0,0000	0,2822	0,0000
B4L	CS2	001(first	0,2848	0,0000	0,0005	0,0000	0,2853	0,0000

Figure 56. Tendon Elongation Table

The effective stresses and effective prestressing force in the tendons can be checked by group and construction stage. Vertical and horizontal force components of the tendons can be readily obtained from the distance from the centroid of the section to the tendon group and the orientation of the tendon (direction cosine).

Results / Result Tables / Tendon Arrangement…

Select a construction stage and click Apply to produce the results corresponding to the stage.

Elem	Part	Tendon Number	Yp (m)	Zp (m)	Average Sin θ ([deg])	Average Cos θ ([deg])	Average Stress (kN/m²)	Average Force (kN)
The arrangement data for tendon group [A1] at the stage of [CS1]								
Tendon Group	A1			Stage	CS1	Apply		
1	I	2	0,0000	0,2094	-0,1301	-0,9915	1140860,7379	1838,1548
1	J	2	0,0000	-0,1187	-0,1301	-0,9915	1166973,4564	1880,2276
2	I	2	0,0000	-0,1187	-0,1317	-0,9913	1166973,7832	1880,2282
2	J	2	0,0000	-0,4507	-0,1317	-0,9913	1196465,8749	1927,7458
3	I	2	0,0000	-0,4507	-0,1028	-0,9947	1196466,3047	1927,7465
3	J	2	0,0000	-0,7090	-0,1028	-0,9947	1228038,9033	1978,6163
4	I	2	0,0000	-0,7090	-0,0736	-0,9973	1228039,2614	1978,6169
4	J	2	0,0000	-0,8934	-0,0736	-0,9973	1250906,0441	2015,4598
5	I	2	0,0000	-0,8934	-0,0442	-0,9990	1250906,3049	2015,4602
5	J	2	0,0000	-1,0039	-0,0442	-0,9990	1222161,0060	1969,1458
6	I	2	0,0000	-1,0039	-0,0147	-0,9999	1222161,1722	1969,1461
6	J	2	0,0000	-1,0406	-0,0147	-0,9999	1194626,3057	1924,7819
7	I	2	0,0000	-1,0406	0,0000	1,0000	1194626,3532	1924,7820
7	J	2	0,0000	-1,0405	0,0000	1,0000	1174713,4298	1892,6983
8	I	2	0,0000	-1,0405	-0,0000	-1,0000	1174713,4298	1892,6983
8	J	2	0,0000	-1,0406	-0,0000	-1,0000	1155564,6599	1861,8458
9	I	2	0,0000	-1,0406	0,0001	1,0000	1155564,6595	1861,8458
9	J	2	0,0000	-1,0404	0,0001	1,0000	1136158,1002	1830,5779

Figure 57. Tendon Arrangement Table

The effective stresses & forces in the table above are the results reflecting both immediate and long-term losses of the tendon. If the effective prestress forces for the immediate losses (friction, anchorage slip & elastic shortening) other than the long-term losses are of interest, right-click on the table and check the forces from "Tendon Immediate Loss Graph".

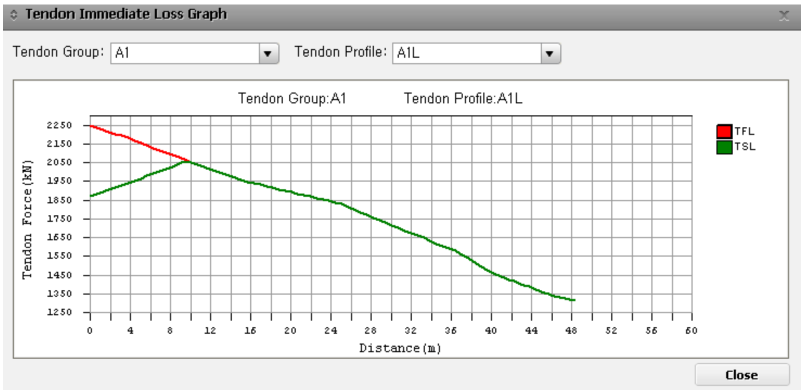


Figure 58. Tendon Force due to Immediate Loss

For each tendon group, losses to due friction, anchorage slip, elastic shortening, creep, shrinkage, relaxation, etc. are separately classified in a table.

Results Tab / Result Tables / Tendon Loss…

Select a construction stage and click Apply to produce the results corresponding to the stage.

Elem	Part	Stress (After Immediate Loss) : A (kN/m²)	Elastic Deform. Loss : B (kN/m²)	Stress(Elastic Loss)/ Stress(Immediate Loss)	Creep/Shrinkage Loss (kN/m²)	Relaxation Loss (kN/m²)	Stress(After All Loss)/ Stress(After Immediate Loss)	Effective Num.
The Loss of tendon group [A1] at the stage of [CS1]								
Tendon Group		B3	Stage	CS4	Apply			
1	I	1161959,6511	267,1150	CS2	-13576,4332	-7789,5949	0,9818	2,0000
1	J	1189636,2538	315,7131	CS3	-15003,3760	-7975,1345	0,9809	2,0000
2	I	1189636,2538	315,7089	CS4	-15003,0450	-7975,1345	0,9810	2,0000
2	J	1219836,6206	396,7800	1,0003	-15589,9327	-8177,5930	0,9808	2,0000
3	I	1219836,6206	396,7729	1,0003	-15589,4958	-8177,5930	0,9808	2,0000
3	J	1251591,3959	479,3410	1,0004	-15641,3615	-8390,4721	0,9812	2,0000
4	I	1251591,3959	479,3338	1,0004	-15640,9962	-8390,4721	0,9812	2,0000
4	J	1274265,1849	544,6933	1,0004	-15361,3604	-8542,4736	0,9817	2,0000
5	I	1274265,1849	544,6872	1,0004	-15361,0936	-8542,4736	0,9817	2,0000
5	J	1244113,8475	552,5986	1,0004	-14165,0963	-8340,3438	0,9824	2,0000
6	I	1244113,8475	552,5943	1,0004	-14164,9259	-8340,3438	0,9824	2,0000
6	J	1215069,1930	521,4844	1,0004	-12818,7387	-8145,6330	0,9832	2,0000
7	I	1215069,1930	521,4831	1,0004	-12818,6900	-8145,6330	0,9832	2,0000
7	J	1194131,8609	488,5435	1,0004	-11901,7021	-8005,2724	0,9837	2,0000
8	I	1194131,8609	488,5435	1,0004	-11901,7022	-8005,2724	0,9837	2,0000
8	J	1174479,8756	473,5575	1,0004	-11515,2447	-7873,5285	0,9839	2,0000
9	I	1174479,8756	473,5575	1,0004	-11515,2451	-7873,5285	0,9839	2,0000
9	J	1155066,6314	475,5742	1,0004	-11640,7201	-7743,3852	0,9836	2,0000
10	I	1155066,6314	475,5740	1,0004	-11640,7152	-7743,3852	0,9836	2,0000
10	J	1134147,6817	494,6497	1,0004	-12265,3886	-7603,1479	0,9829	2,0000
11	I	1134147,6817	494,6557	1,0004	-12265,6237	-7603,1479	0,9829	2,0000
11	J	1099333,9226	465,1451	1,0004	-12557,5779	-7369,7619	0,9823	2,0000
12	I	1099333,9226	465,1616	1,0004	-12558,2697	-7369,7619	0,9823	2,0000
12	J	1064511,0072	367,9269	1,0003	-12444,9432	-7136,3145	0,9820	2,0000
13	I	1064511,0072	367,9462	1,0003	-12445,9457	-7136,3145	0,9820	2,0000
13	J	1031565,9112	262,1179	1,0003	-12472,6840	-6915,4558	0,9815	2,0000
14	I	1031565,9112	262,1380	1,0003	-12474,1168	-6915,4558	0,9815	2,0000
14	J	996879,1124	220,5014	1,0002	-13507,6980	-6682,9209	0,9800	2,0000
15	I	996879,1124	220,5053	1,0002	-13508,0415	-6682,9209	0,9800	2,0000

Tendon Loss (Stress) / Tendon Loss (Force)

Figure 59. Tendon (Tension) Loss Table

Right-click on the table and select "Tendon Time-dependent Loss Graph" to check the effective prestress forces after accounting for tension losses.

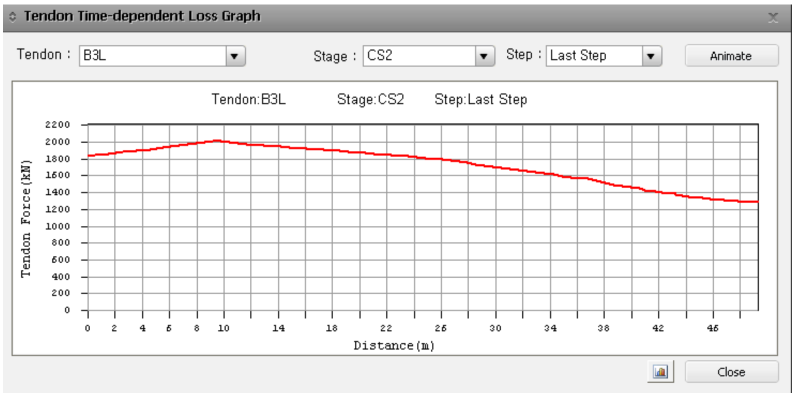


Figure 60. Tendon Time-dependent Loss Graph

Tendon type, property and weight for each group can be tabulated.

Results / Result Tables / Tendon / Tendon Weight…

PostCS

Tendon Weight can be produced only in the PostCS stage.

Tendon Name	Tendon Num	Area (m²)	Length (m)	Weight/Length (kN/m)	Weight (kN)	Total Weight (kN)
A1L	1,00	0,001611	48,320358	0,126479	6,111520	6,111520
A1R	1,00	0,001611	48,320358	0,126479	6,111520	6,111520
A2L	1,00	0,001611	48,345183	0,126479	6,114660	6,114660
A2R	1,00	0,001611	48,345183	0,126479	6,114660	6,114660
A3L	1,00	0,001611	48,282773	0,126479	6,106767	6,106767
A3R	1,00	0,001611	48,282773	0,126479	6,106767	6,106767
A4L	1,00	0,001611	48,323420	0,126479	6,111908	6,111908
A4R	1,00	0,001611	48,323420	0,126479	6,111908	6,111908
B1L	1,00	0,001611	45,341275	0,126479	5,734728	5,734728
B1R	1,00	0,001611	45,341275	0,126479	5,734728	5,734728
B2L	1,00	0,001611	45,381107	0,126479	5,739766	5,739766
B2R	1,00	0,001611	45,381107	0,126479	5,739766	5,739766
B3L	1,00	0,001611	49,312943	0,126479	6,237062	6,237062
B3R	1,00	0,001611	49,312943	0,126479	6,237062	6,237062
B4L	1,00	0,001611	49,339207	0,126479	6,240383	6,240383
B4R	1,00	0,001611	49,339207	0,126479	6,240383	6,240383
C1L	1,00	0,001611	32,159596	0,126479	4,067520	4,067520
C1R	1,00	0,001611	32,159596	0,126479	4,067520	4,067520
C2L	1,00	0,001611	32,115616	0,126479	4,061957	4,061957
C2R	1,00	0,001611	32,115616	0,126479	4,061957	4,061957
C3L	1,00	0,001611	36,068159	0,126479	4,561872	4,561872
C3R	1,00	0,001611	36,068159	0,126479	4,561872	4,561872
C4L	1,00	0,001611	36,026608	0,126479	4,556617	4,556617
C4R	1,00	0,001611	36,026608	0,126479	4,556617	4,556617
SUM	24,00	-	1038,032491	0,126479	-	131,289519

Figure 61. Tendon Weight Table

Checking Moving Load Analysis Results

The member forces produced in moving load analysis are the results of maximum values for each component in the corresponding element. As such, the locations of the loads causing each maximum force component may be different.

In order to obtain the concurrent member forces, right-click on the table and use the "View by Max Value Item" function. We can then check the corresponding force components associated with one maximum force component.

Results / Result Tables / Beam / Force

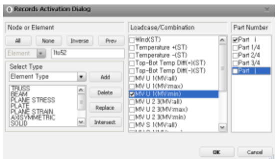
Loadcase/Combination>**MV U 1(MV:min)** ; Part Number>**Part I** ↵

(Context Menu) View by Max Value Item

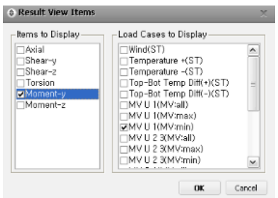
Items to Display>**Moment-y**

Load Cases to Display> **MV U 1(MV:min)** ↵

When Moment-y is maximum, other force components occurring at the same time are produced.



	Elem	Load	Part	Axial (kN)	Shear-y (kN)	Shear-z (kN)	Torsion (kN·m)	Moment-y (kN·m)	Moment-z (kN·m)
▶	1	MV U 1(	I[1]	-0,16	-0,09	-2152,40	-2442,67	-0,00	-1,49
	2	MV U 1(	I[2]	-0,17	-0,09	-1950,83	-2311,11	-603,54	-1,55
	3	MV U 1(	I[3]	-0,15	-0,09	-1756,05	-2200,67	-1207,08	-1,33
	4	MV U 1(	I[4]	-0,13	-0,09	-1567,91	-2104,52	-1810,61	-1,20
	5	MV U 1(	I[5]	-0,11	-0,09	-1386,10	-2016,92	-2414,15	-1,18
	6	MV U 1(	I[6]	-0,05	-0,09	-1211,33	-1934,47	-3017,69	-1,23
	7	MV U 1(	I[7]	-0,00	-0,09	-1044,29	-1856,44	-3621,23	-1,37
	8	MV U 1(	I[8]	-0,00	-0,09	-885,66	-1782,33	-4224,77	-1,60
	9	MV U 1(	I[9]	-0,00	-0,09	-736,14	-1711,74	-4828,30	-1,82
	10	MV U 1(	I[10]	-0,01	-0,09	-597,48	-1648,06	-5431,84	-2,05
	11	MV U 1(	I[11]	-0,09	-0,09	-468,90	-1588,41	-6035,38	-2,52
	12	MV U 1(	I[12]	-0,18	-0,09	-351,24	-1680,35	-6638,92	-3,66
	13	MV U 1(	I[13]	-0,19	-0,09	-245,05	-1783,88	-7242,46	-4,85
	14	MV U 1(	I[14]	-0,15	-0,09	-150,98	-1895,26	-7850,23	-6,14
	15	MV U 1(	I[15]	-0,11	-0,09	-81,48	-2014,69	-8900,70	-6,10
	16	MV U 1(	I[16]	-0,03	-0,09	-66,25	-2145,80	-11084,62	-4,20
	17	MV U 1(	I[17]	-0,75	-0,09	-60,23	-2292,01	-14005,03	-3,55
	18	MV U 1(	I[18]	-0,26	-0,19	-2267,31	-2477,43	-11707,35	-8,40
	19	MV U 1(	I[19]	-0,54	-0,19	-2149,72	-2355,54	-9753,36	-10,03
	20	MV U 1(	I[20]	-0,61	-0,19	-2029,18	-2242,36	-8316,63	-10,07
	21	MV U 1(	I[21]	-0,36	-0,19	-1906,15	-2137,61	-7541,56	-7,79
	22	MV U 1(	I[22]	-0,25	-0,19	-1781,76	-2043,50	-7024,62	-6,43
	23	MV U 1(	I[23]	-0,07	-0,19	-1624,04	-1933,28	-6504,59	-4,87
	24	MV U 1(	I[24]	-0,00	-0,19	-1464,94	-1832,42	-6103,68	-4,24
	25	MV U 1(	I[25]	-0,00	-0,19	-1305,78	-1738,87	-5830,90	-3,98
	26	MV U 1(	I[26]	-0,00	-0,19	-1147,81	-1652,56	-5558,05	-3,73
	27	MV U 1(	I[27]	-0,00	-0,19	-992,30	-1573,42	-5292,60	-3,48
	28	MV U 1(	I[28]	-0,00	-0,19	-840,54	-1506,96	-5573,32	-3,24
	29	MV U 1(	I[29]	-0,00	-0,19	-696,67	-1581,49	-5854,04	-3,00
	30	MV U 1(	I[30]	-0,11	-0,19	-560,53	-1663,18	-6134,75	-3,13



	Elem	Load	Part	Component	Axial (kN)	Shear-y (kN)	Shear-z (kN)	Torsion (kN·m)	Moment-y (kN·m)	Moment-z (kN·m)
▶	1	MV U 1(	I[1]	Moment-y	1,03	0,01	-1607,04	-255,72	-0,00	-0,22
	2	MV U 1(	I[2]	Moment-y	-0,16	0,00	241,42	2,30	-603,54	-0,00
	3	MV U 1(	I[3]	Moment-y	-0,13	0,00	241,42	2,30	-1207,08	-0,00
	4	MV U 1(	I[4]	Moment-y	-0,09	0,00	241,42	2,30	-1810,61	-0,00
	5	MV U 1(	I[5]	Moment-y	-0,06	0,00	241,42	2,30	-2414,15	-0,01
	6	MV U 1(	I[6]	Moment-y	-0,02	0,00	241,42	2,30	-3017,69	-0,01
	7	MV U 1(	I[7]	Moment-y	0,00	0,00	241,42	2,30	-3621,23	-0,01
	8	MV U 1(	I[8]	Moment-y	-0,00	0,00	241,42	2,30	-4224,77	-0,01
	9	MV U 1(	I[9]	Moment-y	0,00	0,00	241,42	2,30	-4828,30	-0,01
	10	MV U 1(	I[10]	Moment-y	-0,00	0,00	241,42	2,30	-5431,84	-0,01
	11	MV U 1(	I[11]	Moment-y	0,06	0,00	241,42	2,30	-6035,38	-0,02
	12	MV U 1(	I[12]	Moment-y	0,18	0,00	241,42	2,30	-6638,92	-0,02
	13	MV U 1(	I[13]	Moment-y	0,30	0,00	241,42	2,30	-7242,46	-0,02
	14	MV U 1(	I[14]	Moment-y	0,48	0,00	270,23	41,15	-7850,23	-0,19
	15	MV U 1(	I[15]	Moment-y	1,26	-0,01	753,13	-99,12	-8900,70	0,36
	16	MV U 1(	I[16]	Moment-y	0,59	-0,01	1028,96	-300,48	-11084,62	0,62
	17	MV U 1(	I[17]	Moment-y	-0,58	0,03	1806,69	148,43	-14005,03	-1,17
	18	MV U 1(	I[18]	Moment-y	1,01	0,03	-1063,61	509,56	-11707,35	1,80
	19	MV U 1(	I[19]	Moment-y	1,81	0,02	-916,48	247,77	-9753,36	1,37
	20	MV U 1(	I[20]	Moment-y	0,99	-0,03	-452,66	-392,62	-8316,63	-2,43
	21	MV U 1(	I[21]	Moment-y	0,41	-0,02	-334,20	-168,74	-7541,56	-1,00
	22	MV U 1(	I[22]	Moment-y	0,15	0,03	-208,67	-3,71	-7024,62	0,14
	23	MV U 1(	I[23]	Moment-y	0,04	0,03	-208,12	-1,80	-6504,59	0,08
	24	MV U 1(	I[24]	Moment-y	0,00	-0,01	-109,23	0,60	-6103,68	0,01
	25	MV U 1(	I[25]	Moment-y	0,00	-0,01	-109,21	0,57	-5830,90	0,04
	26	MV U 1(	I[26]	Moment-y	-0,00	0,01	-109,20	-0,55	-5558,05	-0,06
	27	MV U 1(	I[27]	Moment-y	-0,00	0,04	112,28	2,00	-5292,60	-0,26
	28	MV U 1(	I[28]	Moment-y	0,00	0,04	112,28	1,97	-5573,32	-0,35
	29	MV U 1(	I[29]	Moment-y	-0,00	0,04	112,28	1,96	-5854,04	-0,44
	30	MV U 1(	I[30]	Moment-y	0,03	0,04	112,28	1,94	-6134,75	-0,53
	31	MV U 1(	I[31]	Moment-y	0,15	0,03	210,57	2,37	-6549,31	-0,45

Figure 62. Moving Load Results

Checking Stresses due to Combined Loads

Create load combinations.

Results / Combinations
PostCS
Name**(Temperature)** ; Type>**Envelop**
LoadCase> **Temperature (+)(ST)** ; Factor**(1.0)**
LoadCase> **Temperature (-)(ST)** ; Factor**(1.0)**
Name**(Top-Bot Temp Diff)** ; Type>**Envelop**
LoadCase> **Top-Bot Temp Diff (+)(ST)** ; Factor**(1.0)**
LoadCase> **Top-Bot Temp Diff (-)(ST)** ; Factor**(1.0)**

Name**(ULS 1)** ; Type>**Add**
LoadCase>**Summation(CS)** ; Factor**(1.15)**
LoadCase>**Erection Load(CS)** ; Factor**(1.2)**
LoadCase>**SM(SM)** ; Factor**(1.2)**
LoadCase>**MV U 1** ; Factor**(1.0)**
Name**(SLS 2)** ; Type>**Add**
LoadCase>**Summation(CS)** ; Factor**(1.0)**
LoadCase>**Erection Load(CS)** ; Factor**(1.0)**
LoadCase>**Wind(ST)** ; Factor**(1.0)**
LoadCase>**SM(SM)** ; Factor**(1.0)**
LoadCase>**MV S 2 3** ; Factor**(1.0)**
Name**(SLS 3)** ; Type>**Add**
LoadCase>**Summation(CS)** ; Factor**(1.0)**
LoadCase>**Erection Load(CS)** ; Factor**(1.0)**
LoadCase>**Temperature(CB)** ; Factor**(1.0)**
LoadCase>**Top-Bot Temp Diff(CB)** ; Factor**(0.8)**
LoadCase>**SM(SM)** ; Factor**(1.0)**
LoadCase>**MV S 2 3(MV)** ; Factor**(1.0)**

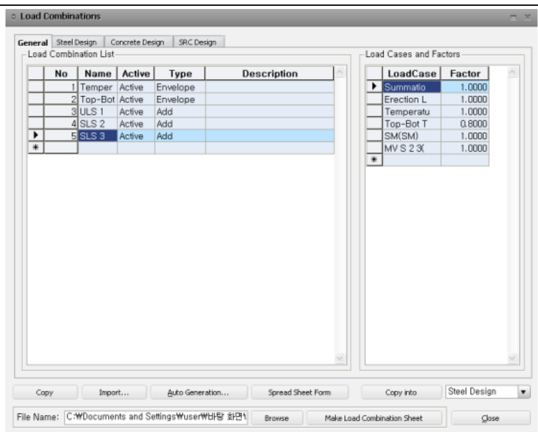


Figure 63. Creating Load Combinations

Check stress results due to load combinations.

Results / Stresses / Beam Stresses(PSC)…
Load Cases/Combinations>**CBall:SLS 3**
Section Position>**Position 1**
Components>**Sig-xx(Summation)**
Type of Display
Contour (on) ; **Legend** (on)

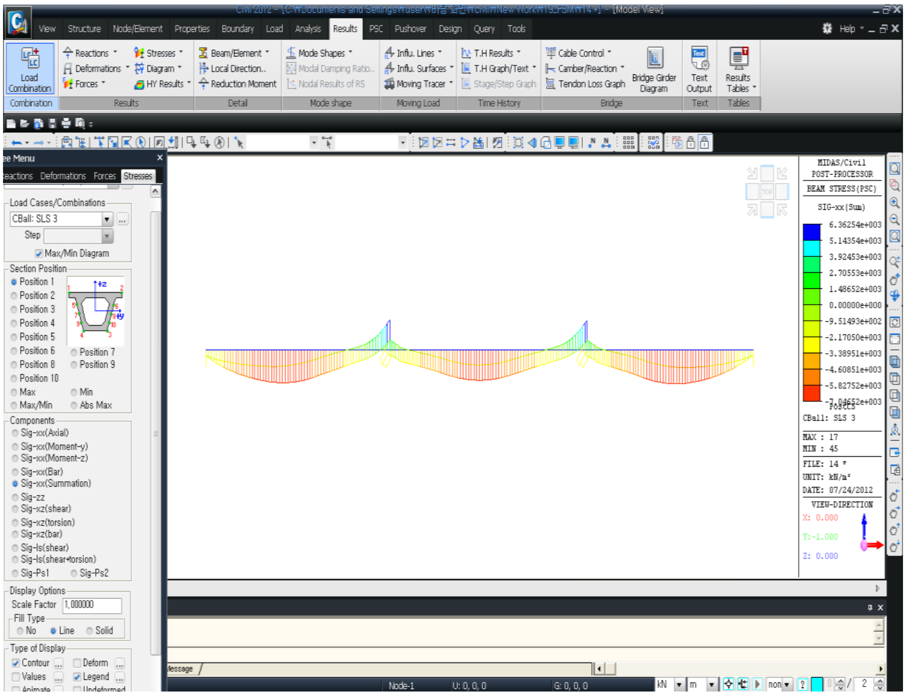


Figure 64. Stress Results due to Serviceability Limit State Combination 3