Advanced Application 12

Final and Forward Construction Stage Analysis for a PC Cable-Stayed Bridge (Part I)

Civil

Table of Contents

Summary

Cable-stayed bridges are structural systems effectively composed of cables, main girders and towers. This bridge form has a beautiful appearance and easily blends in with the surrounding environment due to the fact that various structural systems can be created by changing the tower shapes and cable arrangements.

Cable-stayed bridge is a bridge type where inclined cables transfer member forces induced in the girder. High compression is induced in the tower and main girder due to the structural system. Considering the above features, PC cable-stayed bridges using Prestressed Concrete material for the main girder, have following advantages:

- High buckling resistance compared to steel cable-stayed bridges because of the high stiffness of the tower and the main girder
- High wind and earthquake resistance compared to steel cable-stayed bridges because of the higher weight, stiffness, and damping ratio
- Concrete cable-stayed bridges are better than steel cable-stayed bridges in terms of serviceability as the stiffness of main girder is large, and thus the deflection due to live load is small (resulting in good control of noise/vibration).
- Low cost and easy maintenance compared to steel cable-stayed bridges
- Efficient constructability because it essentially consists of cantilevers, and can be built by constructing out from the towers.
- Economical because the minimized girder depth allows large space under the bridge and this type of bridge allows shorter approach length.

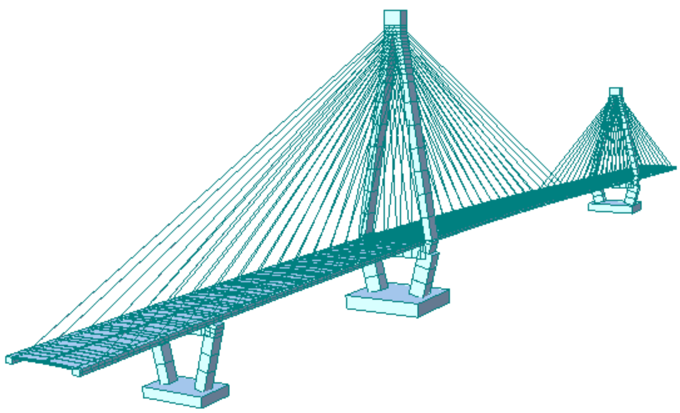


Figure 1. Cable-stayed Bridge

Initial cable pretension analysis considering construction stages

The dominant issue of the design and build of a cable-stayed bridge is to compute and achieve the initial equilibrium configuration at the completed state. The initial equilibrium configuration of cable-stayed bridges is the equilibrium position due to dead load and tension forces in the stay cables. It is called "initial cable pretension analysis" to optimize the cable pretension in order to improve section forces in the main girders and towers and support reactions in the bridge.

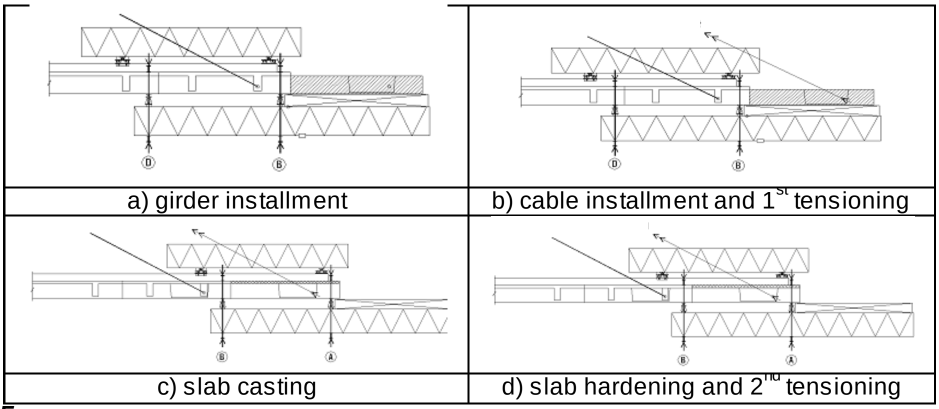
In order to guide the construction of each erection stage, backward analysis is commonly adopted, in which the bridge is disassembled stage by stage from the completed state until just before the first pairs of cables are jacked. The forward analysis starting from any construction stage will predict the states in the successive stages by simulating the actual construction procedures.

This tutorial uses an example of non-symmetrical, cable-stayed bridges. Ideally in backward stage analysis, at key segment closure, shear force and bending moment should be close to 0. However, if backward analysis is applied in this case, non-zero shear force and bending moment occur due to non-symmetry. Thus, it is impossible to apply backward stage analysis in this case. In addition, with backward stage analysis, concrete material effect cannot be considered. Errors due to concrete construction with time can be eliminated by forward iteration analysis. Sequential tensioning and erection sequence, as shown in Figure 2, cannot be represented by backward analysis.

On the other hand, forward stage analysis follows the real erection sequence. It takes more time for the designer as he/she has to conduct trial-and-error analysis to determine the limiting member forces due to cable tension up to a certain range.

In this tutorial, forward stage analysis is used. In the forward stage analysis, it is necessary to know the cable pretensions at each construction stage, which give the initial equilibrium configuration at the completed state due to dead load.

Figure 3 shows the sequence for initial cable pretension analysis with construction stages considered.



a) girder installment

b) cable installment and 1st tensioning

c) slab casting

d) slab hardening and 2nd tensioning

Figure 2. Construction Stage Cycle

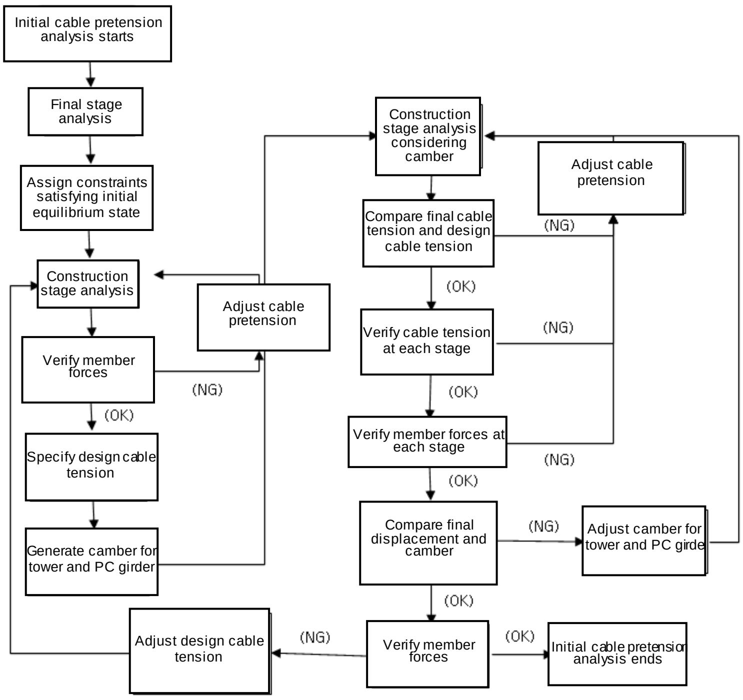


Figure 3. Flow chart for the initial cable pretension analysis considering construction stages

Bridge Dimensions.

This tutorial has been based on a real project of a PC cable-stayed bridge, and has been simplified since it will still suffice for educational purpose. We will learn how to calculate the initial force in the cable from this tutorial. Before performing initial cable pretension analysis with Construction Stages, initial cable forces due to the dead load at the final stage should be first calculated.

The figures and loadings for the bridge are as follows:

Bridge type: PC cable-stayed bridges

Bridge length: L = 46.5+113.5+260.0+100.0 = 520.0 m

2 pair of cables, diamond shape tower

Main girder: Beam and Slab type concrete section

Tower: concrete section

Number of cables: 52×2 pair = 104

Install 4 Key blocks in 1,2,3,4 spans

Put 2 elastic bearings on PY1, PY2

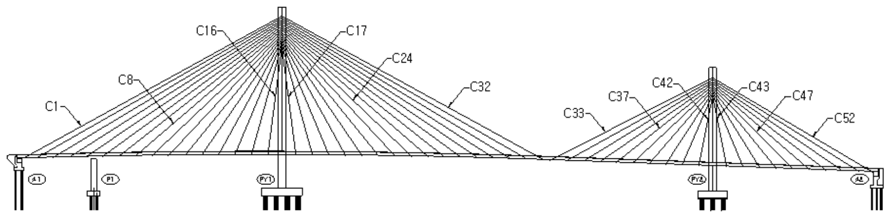


Figure 4. General Layout of Bridge Structure

Loading

Self-Weight.

Automatically calculated by the program.

▶ Superimposed dead load.

	Unit weight (kN/m)	Remark
Pavement	35.75	2.3 x 0.08 x 19.43
Railing	7.28	-
Parapet	14.76	-
Sum	57.75	

Self-weight of cross beams.

Enter the weight of cross beams, which were excluded in the modeling, using Nodal Loads.

Work Environment Setting

To perform the analysis of a PC cable-stayed bridge open a new file (***New Project***) and save it (***Save***) under the name **'PC.mcb'**.

Assign '**kN**' for Force (Mass) unit and '**m**' for Length unit. This unit system can be changed any time during the modeling process as per the convenience of the user.

The Status Bar is located on the bottom of the screen and the units can be changed by clicking on it (kN , m).

File / ***New Project***
File / ***Save* (PC)**

Tools / ***Unit System***

Length > **m** ; Force > **kN** ↵

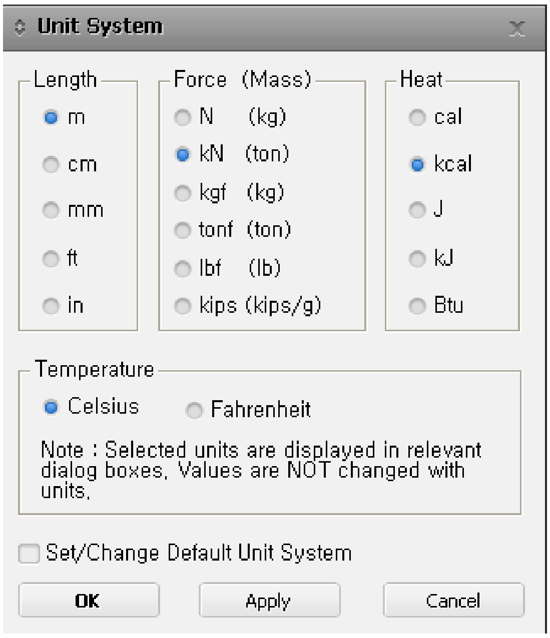


Figure 5. Assign Unit System

Definition of Properties (Attributes)

Definition of Material Properties

Input material properties of cables and bridge deck in the Material Data dialog box.

[Unit : kN, m]

ID	Name	Type of Design	Standard	Modulus of Elasticity	Poisson's Ratio	Thermal Coefficient	Weight Density
1	Main	Concrete	None	2.7389e7	0.167	1.0e-5	24.52
2	Sub	Concrete	None	2.6063e7	0.167	1.0e-5	24.52
3	Cable	Steel	None	2.0594e8	0.3	1.2e-5	76.98

Properties / ***Material Properties***

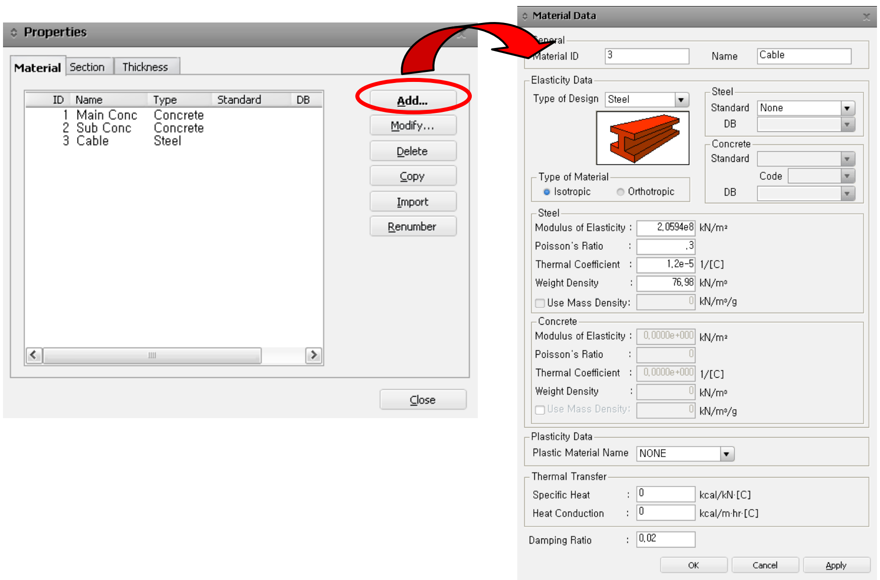


Figure 6. Material Property Input Dialog Box

Definition of Section Properties

With Section Property Calculator (SPC), section properties for irregular shape can be easily obtained and even the shape can be depicted in MIDAS/Civil. Import the *.sec file drawn in SPC to define main girder sections (101, 102 and 103).

Properties / ***Section Properties / Add***

PSC tab > PSC-Value (drop-down menu)

Section ID **(101)** ; Name **(D_center)**

Click on Import from SPC... (under ***Section data***) button and invoke "D_center.sec".

Referring to the guide diagram, enter the design parameters in the "Param. for Design Input" cell. These parameters are used for section capacity check, but not used for analysis. For sections 102 and 103, enter the same parameters.

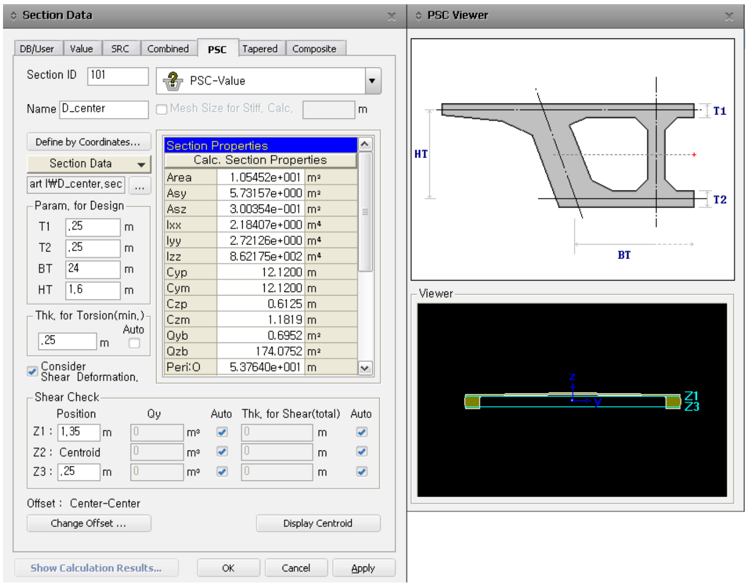


Figure 7. Input dimensions for PC box section

• PC Section

ID	Type	Sub Type	Name	*.sec File Name	Remarks
101	PSC	PSC-Value	D_center	D_center.sec	Center part
102	PSC	PSC-Value	D_spt	D_suppot.sec	Support part
103	PSC	PSC-Value	D_py	D_py.sec	Tower part

• Tower Section

ID	Type	Name	ID	Type	Name
201	Value	PY1_head	211	Value	PY2_head
202	Value	PY1_top	212	Value	PY2_top
203	Value	PY1_down	213	Value	PY2_down
204	DB/User	PY1_cross	214	DB/User	PY2_cross
205	DB/User	PY1_footing	215	DB/User	PY2_footing

• Pier Section

Input 301~304 in Section ID in DB/User Type for modeling the pier section.

• Cable Section

401~409 sections to be used for cables are defined by Value Type. Import *.sec file to define the main girder section. To define other sections, copy the data on the section tab of *Struc.xls* file and paste it into the section table. Classify Section Types into each tab as the type of data and the number of data are different from Section Type to Section Type.

Properties / Property Tables / Section Table

DB/User tab; Value tab (using the provided excel file (struc), copy data accordingly)

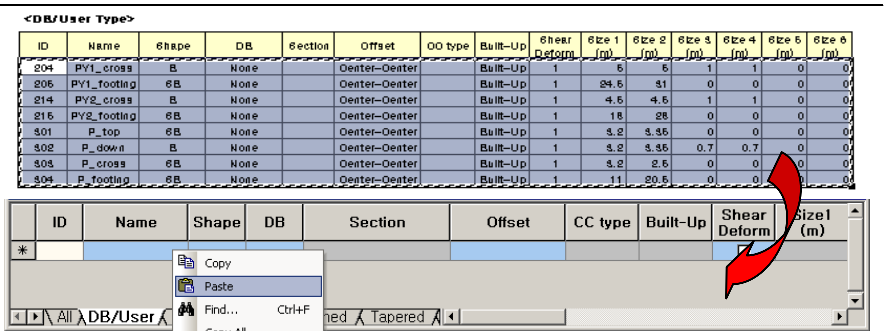


Figure 8. Section Table Input

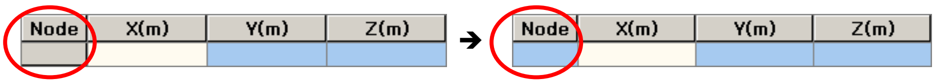
Modeling of Structure

Input Nodes

Input node data in *Struc.xls* file and copy the node information from the file into the Node Tables.

Node/Element > Node ***Tables***

To copy and paste Node Data into Node Table, activate the Node Column as shown below. Right-click over the Node column and select “Enable Edit”. Now the Node column becomes enabled.



Copy the Node Data from the MS-Excel (Struc) file and input it in the Table.

Node No.	X(m)	Y(m)	Z(m)
101	0.200005	0	-0.120999
102	0.950005	0	-0.080931
103	3.49908	0	-0.02553
104	5.49796	0	0.037519
105	7.39972	0	0.045447
106	9.60077	0	0.087961
107	11.2969	0	0.187265
108	14.5967	0	0.262128
109	16.2967	0	0.328129
110	17.9967	0	0.394131
111	21.2967	0	0.460133
112	24.5967	0	0.521044
113	26.2967	0	0.581956
114	27.9967	0	0.642868

➔

Node	X(m)	Y(m)	Z(m)
101	0.200005	0.000000	-0.120999
102	0.950005	0.000000	-0.080931
103	3.499080	0.000000	-0.025530
104	5.497960	0.000000	0.037519
105	7.399720	0.000000	0.045447
106	9.600770	0.000000	0.087961
107	11.296900	0.000000	0.187265
108	14.596700	0.000000	0.262128
109	16.296700	0.000000	0.328129
110	17.996700	0.000000	0.394131
111	21.296700	0.000000	0.460133

Figure 9. Node Information and Input Table

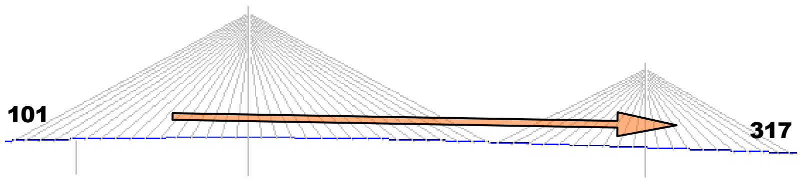
Input Elements

Likewise, enable the Element No. Column for pasting the data into the table. Copy the Element Data from Excel File and paste it into the table.

Node/Element > Element ***Tables***

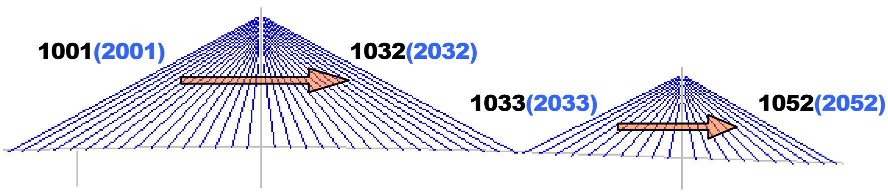
- **Main girder**

Main girder numbers are 101 ~ 317 from the left.

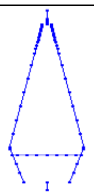
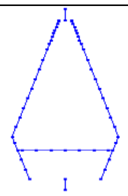
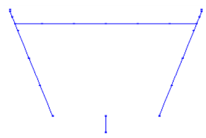


- **Cable**

Cable numbers are 1001 ~ 1032, 2001 ~ 2052 from the left. Numbers in parenthesis indicate the rear cables.



- **Tower and Pier.**

Main tower	Small tower	Pier
		
501to561	601to656	701to719

Input Boundary Conditions

Input Supports

Input the supports as shown in the figure below.

Boundary / Define ***Supports***

Select Single (Node: **389, 390, 397, 398, 2311, 2780, 3106**)
Support Type>**Dx** (on), **Dy** (on), **Dz**(on), **Rx** (on), **Ry** (on), **Rz**(on) ↵

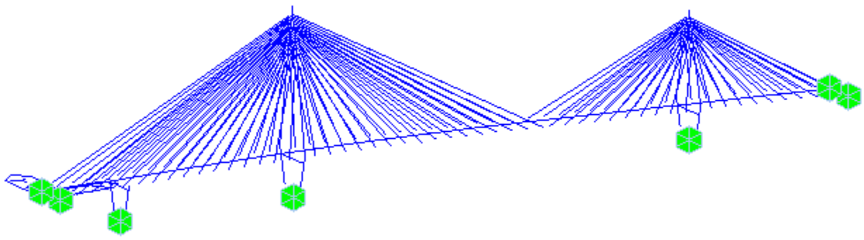


Figure 10. Input Supports

Input Beam End Offsets

Input the width of Beam End Offset at the pier step.

Boundary / ***Beam End Offset***

[Unit: m]

Elem	Type	RGDXi	RGDYi	RGDZi	RGDXj	RGDYj	RGDZj
710	Global	0.0	1.72	0.0	0.0	0.0	0.0
715	Global	0	0	0	0	-1.72	0

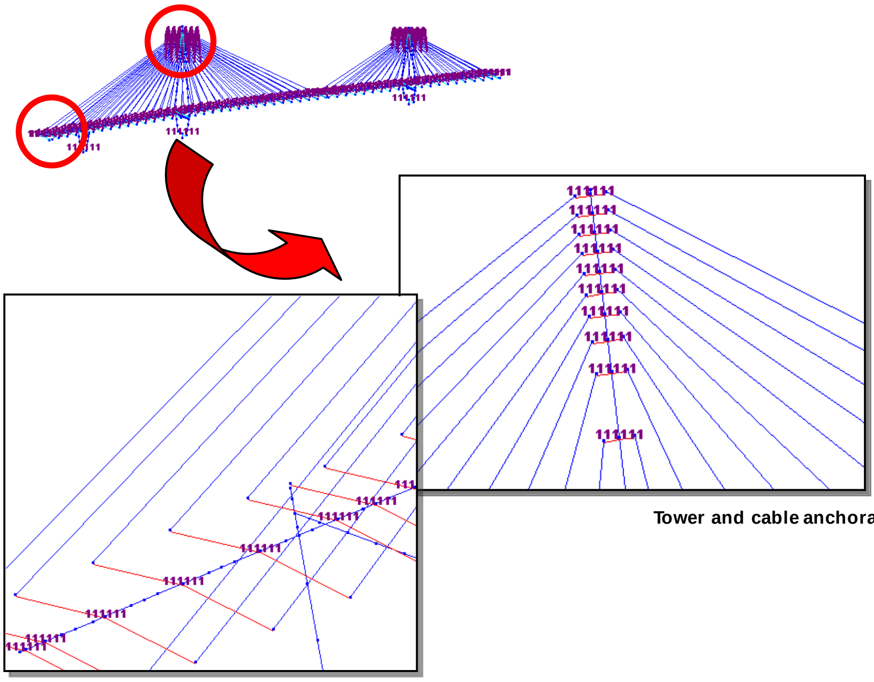
Rigid body connection

Enter rigid body connection between the main girder and cable anchorages, and between the tower and cable anchorages. Copy the data on Rigid Link tab of *Struc.xls* and paste it into Rigid Link Table.

Boundary / Boundary Tables / ***Rigid Link Table***

Later input the rigid body connection at the same location.

• **PC girder, tower and cable anchorages.**



Tower and cable anchorage

PC girder and cable anchorage

Modeling Bridge Supports

Input Elastic Links at bridge supports connecting the bridge superstructure to the substructure.

Boundary / ***Elastic Link***

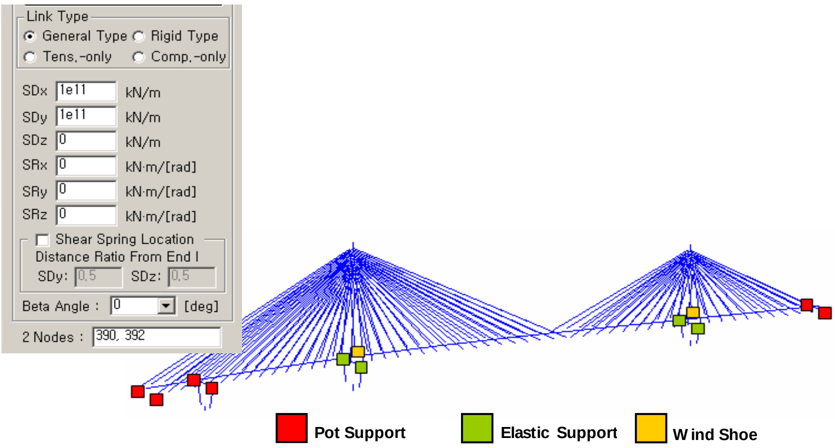


Figure 11. Locations for Installing Bridge Supports

Input the data for elastic links at the bridge supports as shown in the table below:

[Unit: kN, m]

No.	Node1	Node2	Type	SDx	SDy	SDz	Remarks
1	390	392	GEN	1E+11	1E+11	0	Pot support
2	389	391	GEN	1E+11	1E+11	0	Pot support
3	567	394	GEN	25230100	20670	20670	Elastic support
4	561	393	GEN	25230100	20670	20670	Elastic support
5	667	396	GEN	23870000	19810	19810	Elastic support
6	661	395	GEN	23870000	19810	19810	Elastic support
7	398	400	GEN	1E+11	1E+11	0	Pot support
8	397	399	GEN	1E+11	1E+11	0	Pot support
9	1009	168	GEN	0	7808220	0	Wind Shoe
10	1010	275	GEN	0	7808220	0	Wind Shoe
11	3013	3015	GEN	1E+11	1E+11	0	Pot support
12	3012	3014	GEN	1E+11	1E+11	0	Pot support

Input Loads

Define Loading Conditions

The loading conditions used in the analysis are defined.

Redraw

Load / Static Load / ***Static Load Cases***

Name **(Self)** ; Type>**Dead Load (D)** ↵
Name **(2nd Dead)** ; Type>**Dead Load (D)** ↵
Name **(Cross W't)** ; Type>**Dead Load (D)** ↵
Name **(Pre01)** ; Type>**Prestress (PS)** ↵
Name **(Pre02)** ; Type>**Prestress (PS)** ↵
Name **(Pre03)** ; Type>**Prestress (PS)** ↵

…

Name **(Pre52)** ; Type>**Prestress (PS)** ↵

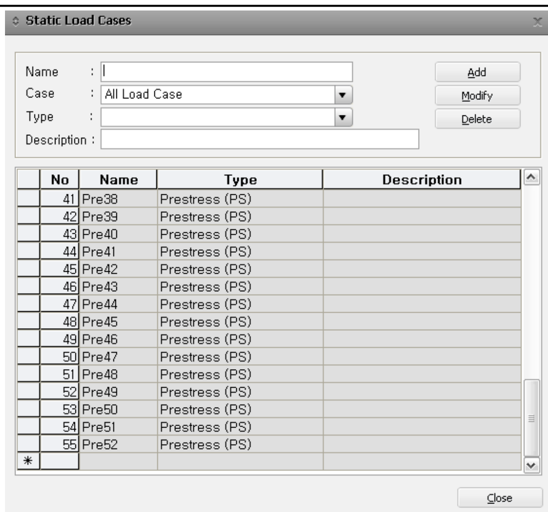


Figure 12. Define Load Case Dialog Box

Input Self Weight

Input the self weight as follows.

Load / *Self Weight*
Load Case Name>**Self**
Load Group Name>**Self**
Self Weight Factor>**Z (-1)** Add ↵

Superimposed Dead Load

Then apply the 2nd dead load by inputting it as Element Beam Load.

Load / Static Loads/Beam Load / *Element*

101to317 **101to317** ↵
Load Case Name>**2nd Dead**
Load Group Name>**2nd Dead**
Load Type>**Uniform Loads**
Value
Relative ; x1(**0**) ; x2 (**1**) ; w (**-56.633**) Apply

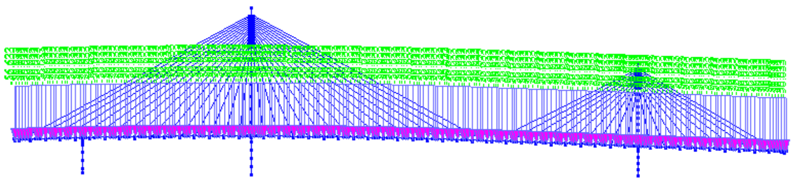


Figure 13. Apply 2nd Dead Load

Self weight of Cross Beams

Enter the weight of cross beams, which was excluded from the modeling, using nodal loads. Copy the loading information from the Load tab in *Struc.xls* and paste it into Nodal Load Table.

Load / Load Tables / Static Load / Nodal Loads

Input Pretension Loads

For the case of a symmetric cable-stayed bridge, as is the case in this tutorial, identical initial pretension in the cables will be introduced to each of the corresponding cable symmetric to the bridge center. Therefore, we will input identical loading conditions to the cable pairs that form the symmetry.

Load / Temp./Prestress / Pretension Loads

Select Intersect (Elements: **1001, 2001**)
Load Case Name > **Pre01**; Load Group Name > **Default**
Options > **Add**; Pretension Load **(1)** ↵

Select Intersect (Elements: **1002, 2002**)
Load Case Name > **Pre02**; Load Group Name > **Default**
Options > **Add**; Pretension Load **(1)** ↵

…

Select Intersect (Elements: **1052, 2052**)
Load Case Name > **Pre 52**; Load Group Name > **Default**
Options > **Add; Pretension** Load **(1)** ↵

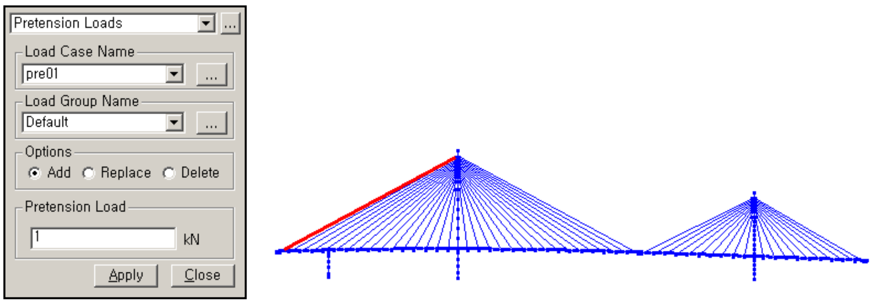


Figure 14. Input Pretension Loads

Perform Structural Analysis

After completing all the processes for modeling and load input, structural analysis is performed.

Analysis / Perform Analysis

Calculate Initial Pretension

Create Load Combinations

Create a load combination from the 52 unit pretension load cases introduced to each cable, self weight load case, and superimposed dead load case and cross beam self weight load case.

Results / ***Load Combinations***

Load Combination LisLCBt > Name > **LCB1**

LoadCase > **Self(ST)** ; Factor **(1.0)**

LoadCase > **2nd Dead(ST)** ; Factor **(1.0)**

LoadCase > **Cross W't(ST)** ; Factor **(1.0)**

LoadCase > **Pre01(ST)** ; Factor **(1.0)**

…

LoadCase > **Pre16(ST)** ; Factor **(250)**

LoadCase > **Pre17(ST)** ; Factor **(250)**

…

LoadCase > **Pre52(ST)** ; Factor **(1.0)** ↵

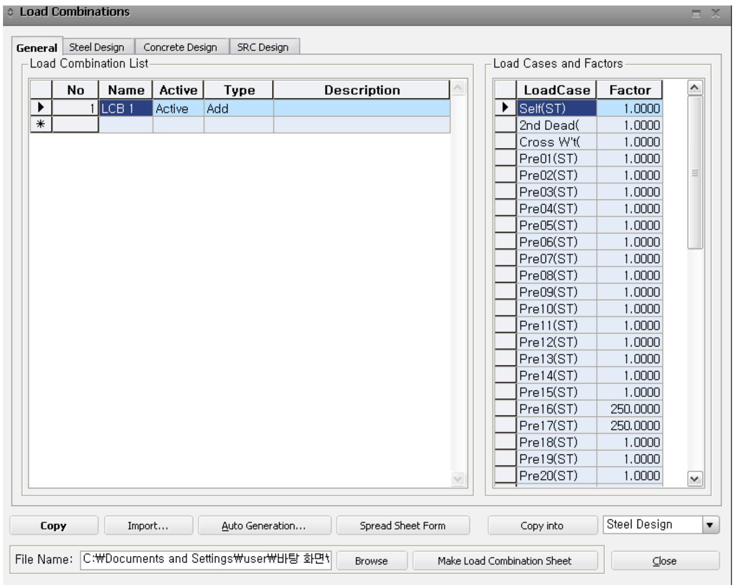


Figure 15. Input Load Combinations

Calculate Unknown Load Factors

Calculate unknown load factors that satisfy the boundary conditions by the *Unknown Load Factor* function for LCB, which was generated through load combination. The constraints are specified to limit the deflection of the tower and the main girders.

Specify the load condition, constraints and method of forming the object function in *Unknown Load Factor*. First, we define the cable unit loading conditions as unknown loads.

Results / Bridge / ***Unknown Load Factor***

Unknown Load Factor Group > Add New

Item Name **(Unknown)** ; Load Comb > **LCB**

Object function type > **Square** ; Sign of unknowns > **Both**

LCase > **Pre01** (on)

…

LCase > **Pre15** (on)

LCase > **Pre18** (on)

…

LCase > **Pre52** (on)

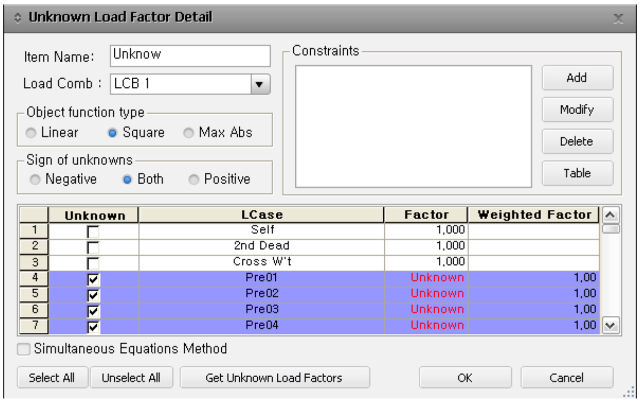


Figure 16. Unknown Load Factors

Specify the constraining conditions, which restrict the displacement of the tower and the main girders by using the ***Constraints*** function.

The boundary conditions for the Unknown Load Factors can also be applied through the MCT Command Shell.

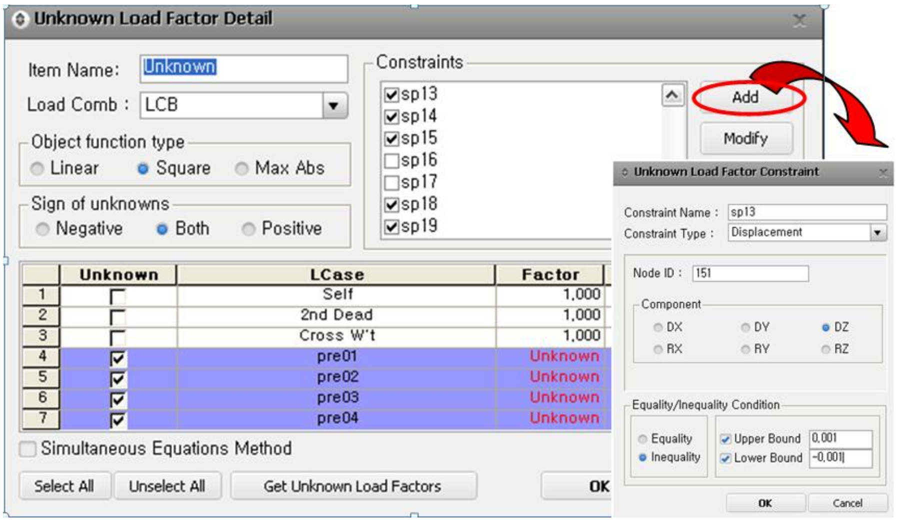


Figure 17. Input Constraint Conditions

Refer to the table below for inputting the constraint conditions for calculating the unknown load factors.

Name	Node	Node ID	Type	Upper	Lower
py101	554	RY	Inequality	1.00E-06	-1.00E-06
py201	645	RY	Inequality	1.00E-06	-1.00E-06
sp02	107	DZ	Inequality	0.001	-0.001
sp03	111	DZ	Inequality	0.001	-0.001
sp04	115	DZ	Inequality	0.001	-0.001
sp05	119	DZ	Inequality	0.001	-0.001
sp06	123	DZ	Inequality	0.001	-0.001
sp07	127	DZ	Inequality	0.001	-0.001
sp08	131	DZ	Inequality	0.001	-0.001
sp09	135	DZ	Inequality	0.001	-0.001
sp10	139	DZ	Inequality	0.001	-0.001
sp11	143	DZ	Inequality	0.001	-0.001
sp12	147	DZ	Inequality	0.001	-0.001

sp13	151	DZ	Inequality	0.001	−0.001
sp14	155	DZ	Inequality	0	−0.001
sp15	159	DZ	Inequality	0	−0.001
sp16	163	DZ	Inequality	0.001	−0.001
sp17	173	DZ	Inequality	0.001	−0.001
sp18	177	DZ	Inequality	0	−0.001
sp19	181	DZ	Inequality	0	−0.001
sp20	185	DZ	Inequality	0.001	−0.001
sp21	189	DZ	Inequality	0.001	−0.001
sp22	193	DZ	Inequality	0.001	−0.001
sp23	197	DZ	Inequality	0.001	−0.001
sp24	201	DZ	Inequality	0.001	−0.001
sp25	205	DZ	Inequality	0.001	−0.001
sp26	209	DZ	Inequality	0.001	−0.001
sp27	213	DZ	Inequality	0.001	−0.001
sp28	217	DZ	Inequality	0.001	−0.001
sp29	221	DZ	Inequality	0.001	−0.001
sp30	225	DZ	Inequality	0.001	−0.001
sp31	229	DZ	Inequality	0.001	−0.001
sp32	233	DZ	Inequality	0.001	−0.001
sp33	234	DZ	Inequality	0.001	−0.001
sp34	238	DZ	Inequality	0.001	−0.001
sp35	242	DZ	Inequality	0.001	−0.001
sp36	246	DZ	Inequality	0.001	−0.001
sp37	250	DZ	Inequality	0.001	−0.001
sp38	254	DZ	Inequality	0.001	−0.001
sp39	258	DZ	Inequality	0	−0.001
sp40	262	DZ	Inequality	0	−0.001
sp41	266	DZ	Inequality	0	−0.001
sp42	270	DZ	Inequality	0.001	−0.001
sp43	280	DZ	Inequality	0.001	−0.001
sp44	284	DZ	Inequality	0	−0.001

sp45	288	DZ	Inequality	0	−0.001
sp46	292	DZ	Inequality	0	−0.001
sp47	296	DZ	Inequality	0.001	−0.001
sp48	300	DZ	Inequality	0.001	−0.001
sp49	304	DZ	Inequality	0.001	−0.001
sp50	308	DZ	Inequality	0.001	−0.001
sp51	312	DZ	Inequality	0.001	−0.001

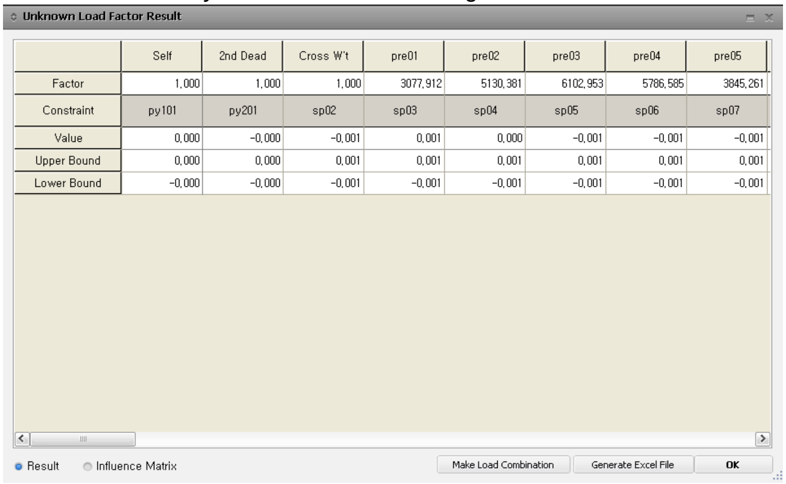
We now check the constraints used to calculate the cable initial pretension and unknown load factors in *Unknown Load Factor Result.*

Unknown Load Factor Group > Get Unknown Load Factors !

Figure 18 shows the analysis results for calculating the Unknown Load Factors.

Make Load Combination uses the Unknown Load Factors which are automatically created for the Load Combination.

Generate the Influence Matrix as a MS-Excel File from the calculation results of Unknown Load Factors.



	Self	2nd Dead	Cross W't	pre01	pre02	pre03	pre04	pre05
Factor	1,000	1,000	1,000	3077,912	5130,381	6102,953	5786,585	3845,261
Constraint	py101	py201	sp02	sp03	sp04	sp05	sp06	sp07
Value	0,000	-0,000	-0,001	0,001	0,000	-0,001	-0,001	-0,001
Upper Bound	0,000	0,000	0,001	0,001	0,001	0,001	0,001	0,001
Lower Bound	-0,000	-0,000	-0,001	-0,001	-0,001	-0,001	-0,001	-0,001

Figure 18. Unknown Load Factor Calculation Results

We will now check whether the calculation results satisfy the constraints by auto-generating a new load combination using the unknown load factors in the ***Make Load Combination*** function.

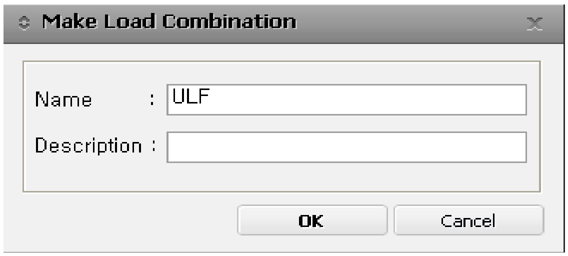


Figure 19. Automatic generation of "ULF" load combination using the unknown load factors

Confirm the results of the load combination that is automatically generated using the unknown load factors.

Results / ***Combinations***

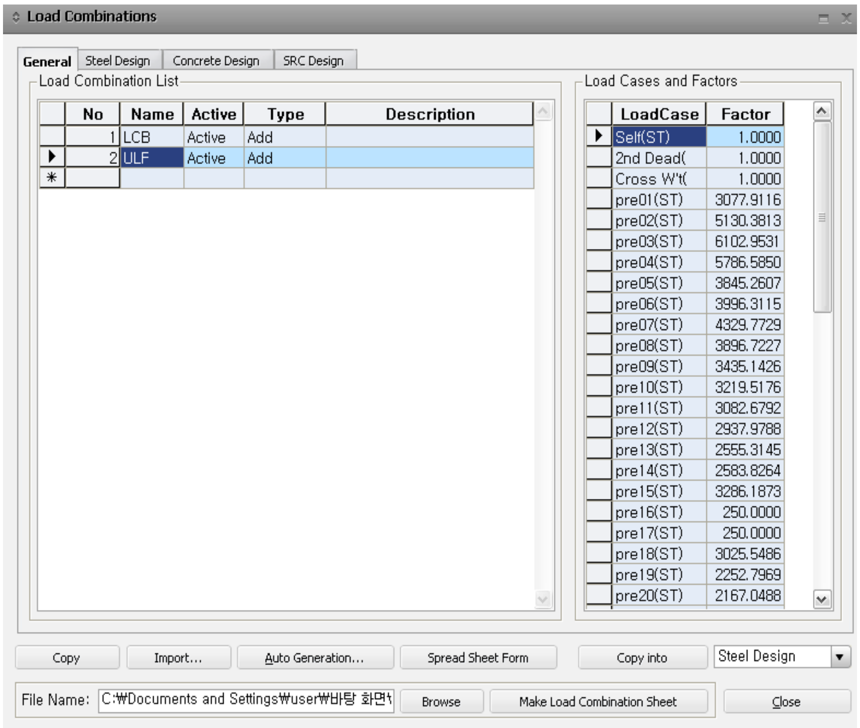


Figure 20. Automatic Generation of "ULF" Load Combination that uses the Unknown Load Factor

Review Analysis Results

Review deformed shape.

Review the deformed shape for the "ULF" load combination, which includes the Unknown Load Factors calculated for the initial tension.

Results / Deformations / ***Deformed Shape***

Load Cases / Combinations > **CB:ULF**

Components > **DZ**

Type of Display > **Undeformed** (on) ; **Legend** (on)

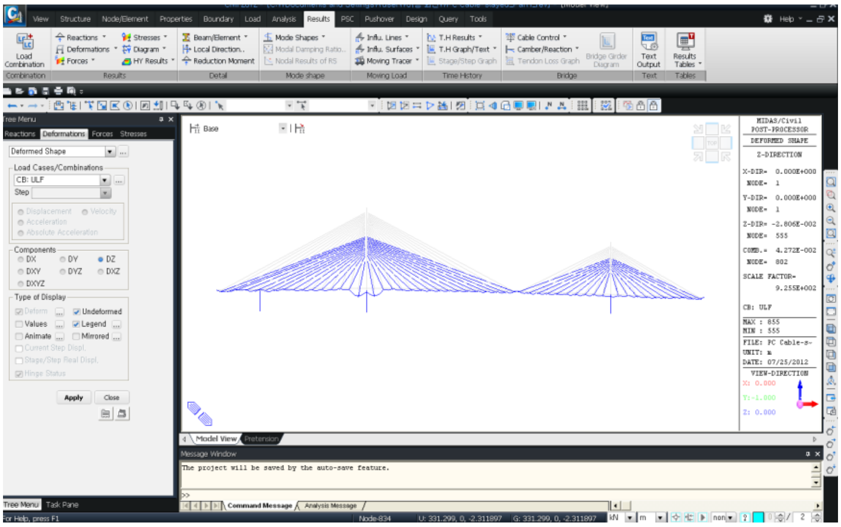


Figure 21. Deformed Shape Results

Review Member Forces

Review the member forces for the “ULF” load combination.

Results / Forces / Beam Diagrams

Load Cases / Combinations > **CB: ULF**

Components > **My**

Type of Display > **Undeformed** (on); Legend (on)

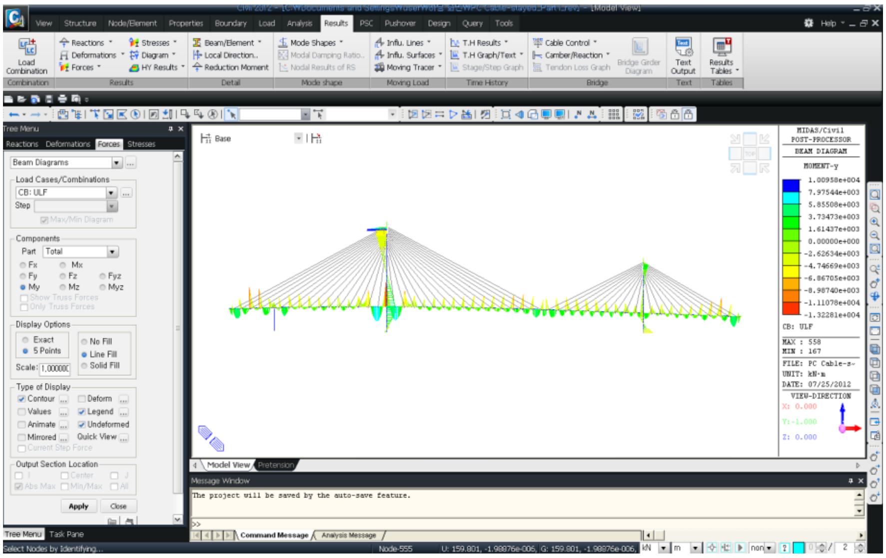


Figure 22. Review Member Forces