# Tutorial

**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 type 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 bridges are of 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 materials for the main girders have the following advantages:

- High buckling resistance compared to steel cable-stayed bridges due to high stiffness of the towers and main girders
- High wind and earthquake resistance compared to steel cable-stayed bridges due to heavier weights, higher stiffness and higher damping ratio
- Concrete cable-stayed bridges are better than steel cable-stayed bridges in terms of serviceability as the stiffness of main girders is large, and thus the deflection due to live loads is relatively 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 depths allow more space under the bridges and this type of bridges allows shorter approach length.

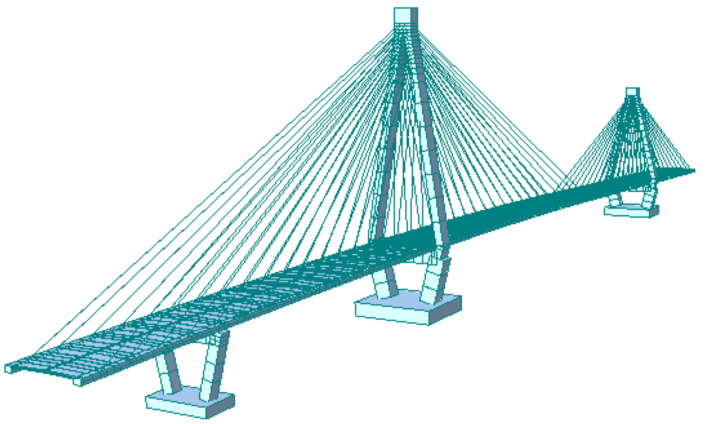


***Figure 1. Cable-stayed Bridge***

## Initial cable pretension analysis considering construction stages

The dominant issue of the design and construction of a cable-stayed bridge is to compute and achieve the initial equilibrium configuration at the completed state. The initial equilibrium configuration of a cable-stayed bridge 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 pretensions 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 a non-symmetrical cable-stayed bridge. 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 will occur due to non-symmetry. Thus, it is not desirable to apply backward stage analysis in this case. In addition, with backward stage analysis, time dependent material effects of concrete cannot be considered. Errors due to concrete construction with the time effect 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 produce the initial equilibrium configuration at the completed state due to dead load.

Figure 3 shows the sequence for initial cable pretension analysis while considering construction stages.

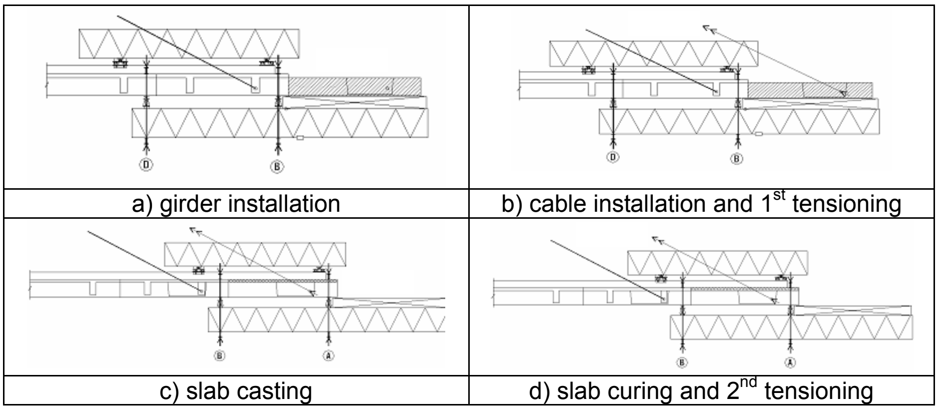



***Figure 2. Construction Stage Cycle***

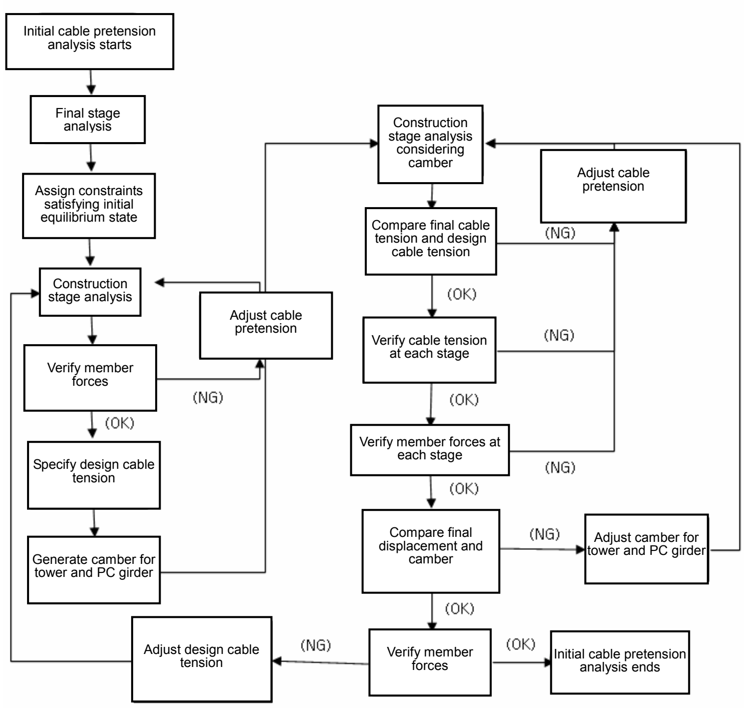



***Figure 3. Flow chart for the initial cable pretension analysis considering construction stages***

## 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 training purposes. We will learn how to calculate the initial forces in the cables 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 bridge
Bridge length: L = 46.5+113.5+260.0+100.0 = 520.0 m
2 sides of cables, diamond shape tower
Main girder: Beam and Slab type concrete sections
Tower: concrete sections
Number of cables: 52×2 sides = 104
Install 4 Key blocks in spans 1, 2, 3 & 4
Place 2 elastic bearings on PY1 & PY2

---

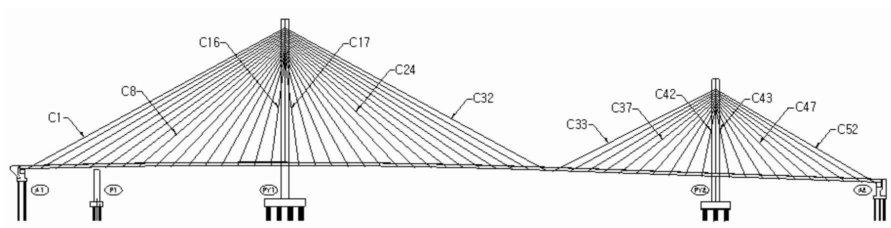



***Figure 4. General Layout of Bridge Structure***

# Loading

**Self Weight**

Automatically calculated by the program

▶ Superimposed dead load.

| | Unit weight (kN/m) | Remarks |
|---|---|---|
| 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 at any time during the modeling process as desired by the user.

**The Status Bar is located at 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** ↵

---

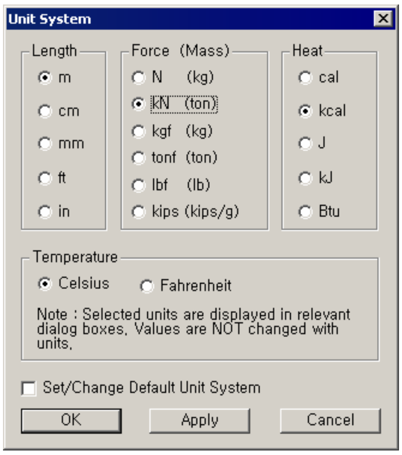



***Figure 5. Assign Unit System***

# 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 |

Model / Properties / ***Material***

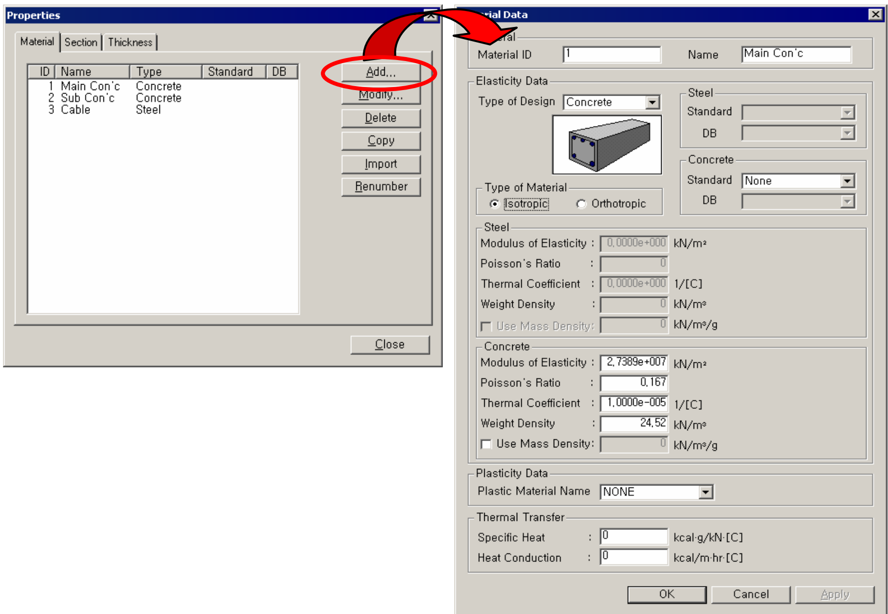


***Figure 6. Material Property Input Dialog Box***

## Definition of Section Properties

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

---

Model / Properties / ***Section***

**PSC tab > General Section**

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

Click on Import from SPC... 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.

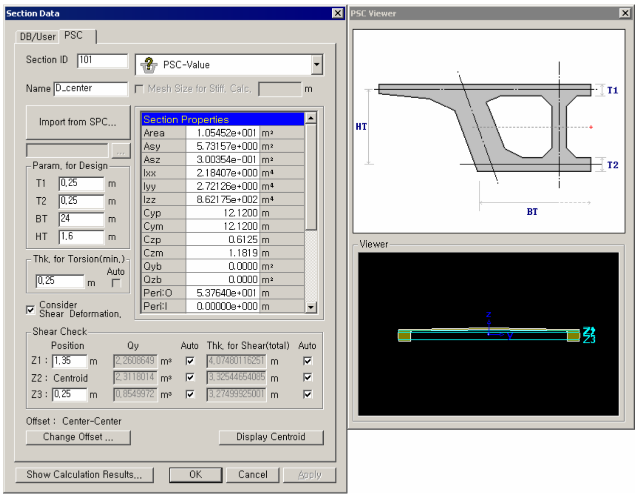


***Figure 7. Input dimensions for PC box section***

- **PC Sections**

| 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 Sections**

| 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 |

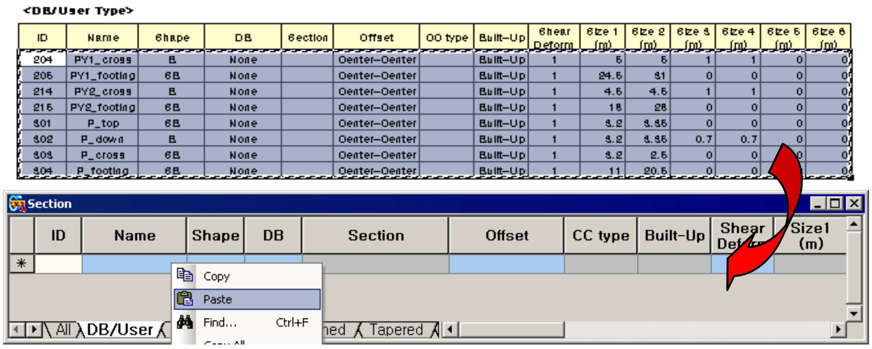
- **Pier Sections**

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

- **Cable Sections**

401~409 sections to be used for cables are defined by Value Type. Import *.sec file to define the main girder sections. 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.

Model / Properties / Section Table
**DB/User tab ; Value tab**


<DB/User Type>

| ID | Name | Shape | DB | Section | Offset | CC type | Built-Up | Shear Deform | Size 1 (m) | Size 2 (m) | Size 3 (m) | Size 4 (m) | Size 5 (m) | Size 6 (m) |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 204 | PY1_cross | B | None | | Center-Center | | Built-Up | 1 | 5 | 5 | 1 | 1 | 0 | 0 |
| 205 | PY1_footing | SB | None | | Center-Center | | Built-Up | 1 | 24.5 | 31 | 0 | 0 | 0 | 0 |
| 214 | PY2_cross | B | None | | Center-Center | | Built-Up | 1 | 4.5 | 4.5 | 1 | 1 | 0 | 0 |
| 215 | PY2_footing | SB | None | | Center-Center | | Built-Up | 1 | 18 | 28 | 0 | 0 | 0 | 0 |
| 301 | P_top | SB | None | | Center-Center | | Built-Up | 1 | 3.2 | 3.35 | 0 | 0 | 0 | 0 |
| 302 | P_down | B | None | | Center-Center | | Built-Up | 1 | 3.2 | 3.35 | 0.7 | 0.7 | 0 | 0 |
| 303 | P_cross | SB | None | | Center-Center | | Built-Up | 1 | 3.2 | 2.5 | 0 | 0 | 0 | 0 |
| 304 | P_footing | SB | None | | Center-Center | | Built-Up | 1 | 11 | 20.5 | 0 | 0 | [illegible] | 0 |



***Figure 8. Section Table Input***

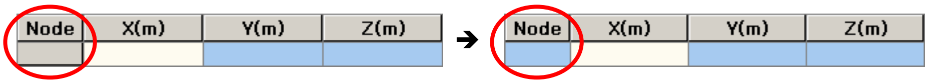
# Modeling of Structure

## Input Nodes

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

Model / Nodes / ***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 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***

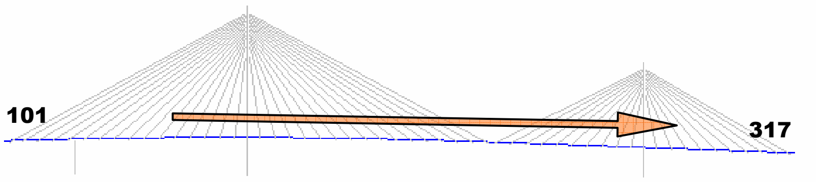
## Input Elements

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.

Model / Nodes / ***Element Tables***

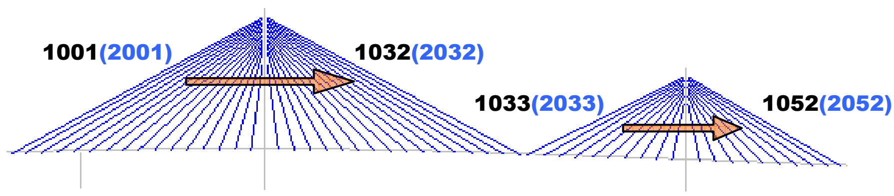
**• Main girders**

Main girder numbers are 101 ~ 317 from the left.

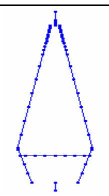
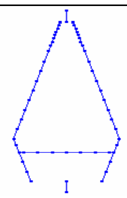
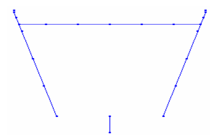



**• Cable**

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.

Model / Boundary / ***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) ↵

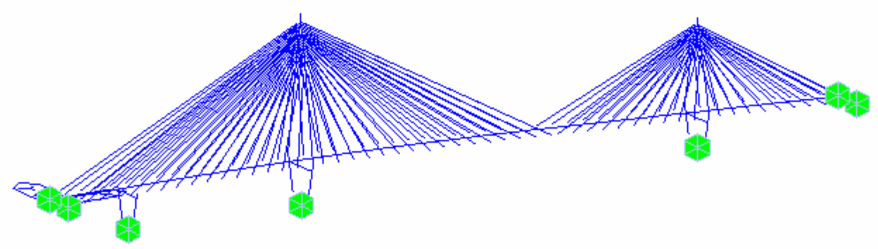


***Figure 10. Input Supports***

## Input Beam End Offsets

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

Model / 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 |

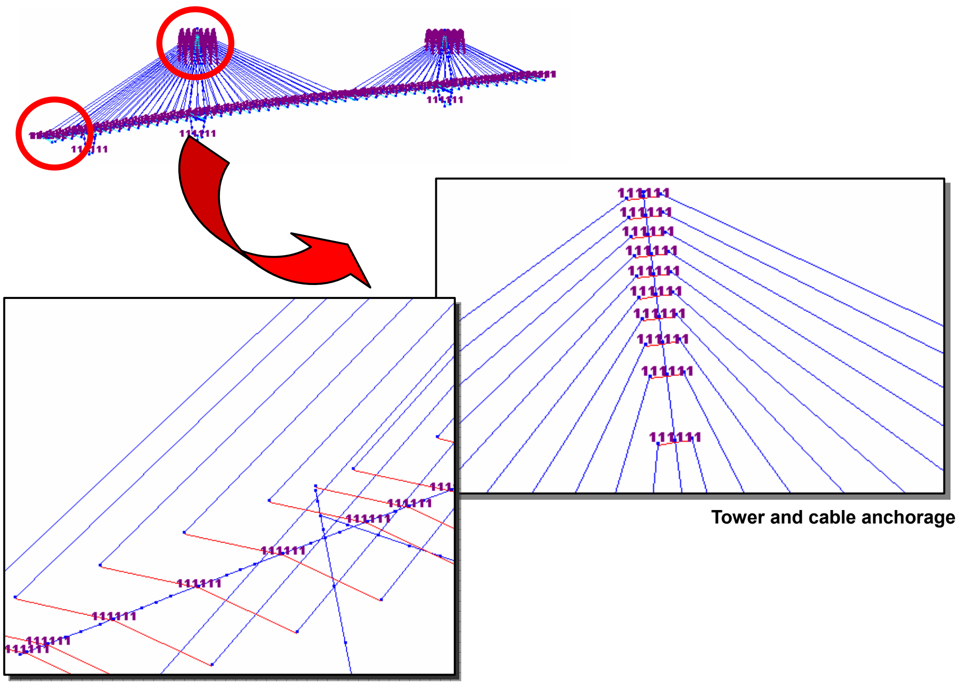
## Rigid body connection

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

Model / Boundary / ***Rigid Link Table***

Input rigid body connections at the following locations.

**• PC girders, towers and cable anchorages**



Tower and cable anchorage sockets

PC girders and cable anchorage sockets

## Modeling Bridge Supports

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

Model / Boundary / ***Elastic Link***

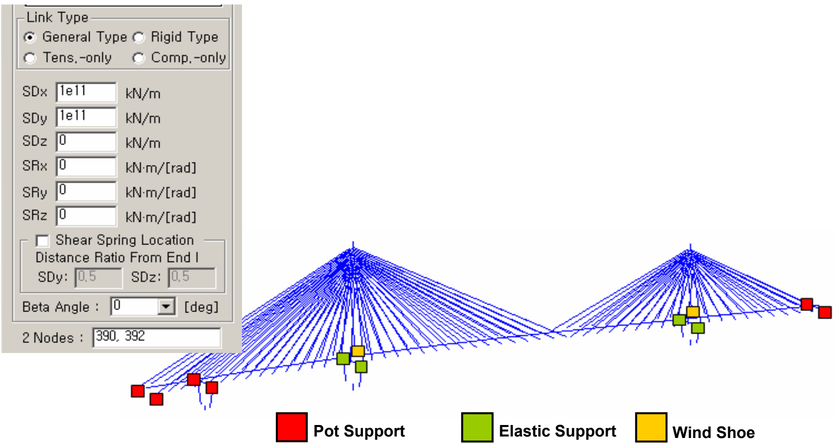



***Figure 11. Locations for Installing Bridge Supports***

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

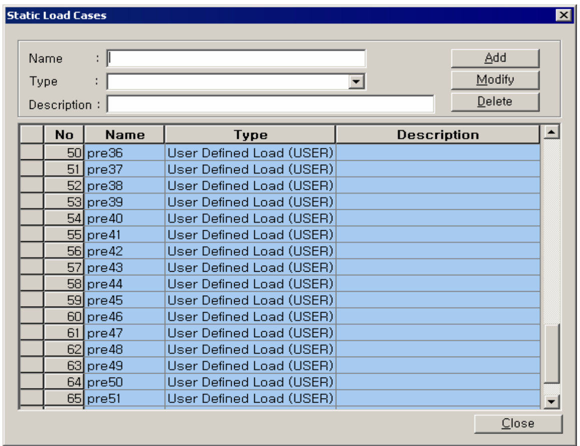
## Define Loading Conditions

The loading conditions used in the analysis are defined.

***Redraw***

Load / ***Static Load 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)** ↵ |




| No | Name | Type | Description |
|---|---|---|---|
| 50 | pre36 | User Defined Load (USER) | |
| 51 | pre37 | User Defined Load (USER) | |
| 52 | pre38 | User Defined Load (USER) | |
| 53 | pre39 | User Defined Load (USER) | |
| 54 | pre40 | User Defined Load (USER) | |
| 55 | pre41 | User Defined Load (USER) | |
| 56 | pre42 | User Defined Load (USER) | |
| 57 | pre43 | User Defined Load (USER) | |
| 58 | pre44 | User Defined Load (USER) | |
| 59 | pre45 | User Defined Load (USER) | |
| 60 | pre46 | User Defined Load (USER) | |
| 61 | pre47 | User Defined Load (USER) | |
| 62 | pre48 | User Defined Load (USER) | |
| 63 | pre49 | User Defined Load (USER) | |
| 64 | pre50 | User Defined Load (USER) | |
| 65 | pre51 | User Defined Load (USER) | |

***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 / ***Element Beam Loads***

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

---

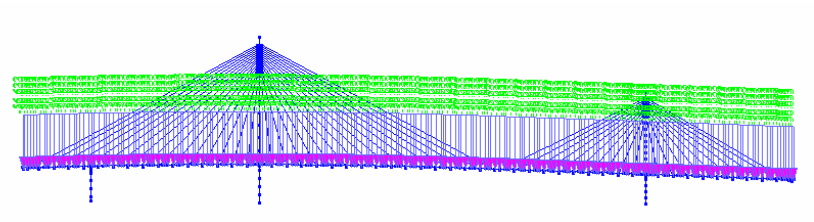


***Figure 13. Apply 2nd Dead Load***

## 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 / Nodal Loads

---

## Input Pretension Loads

Since this cable-stayed bridge has two sides, which are transversely symmetrical, identical initial pretensions in the cables on both sides will be introduced symmetrically to the bridge center. Therefore, we will input identical loading conditions to the cable pairs that form the transverse symmetry.

---

Load / Prestress Loads / 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 )** ↵

---

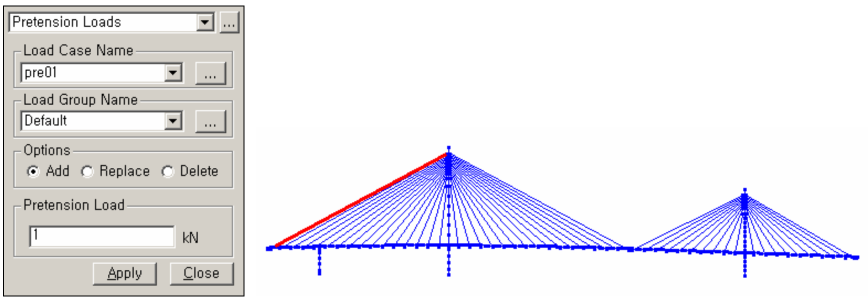



***Figure 14. Input Pretension Loads***

## Perform Structural Analysis

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

---

Analysis / Perform Analysis

---

# Calculate Initial Pretensions

## Create Load Combinations

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

---

Results / ***Combinations***

Load Combination List > 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)** ↵

---

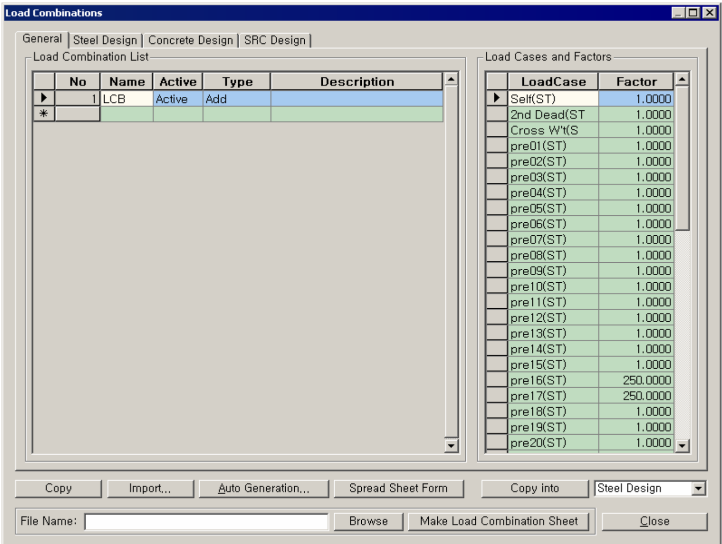



***Figure 15. Input Load Combination***

## 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 deflections 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 / ***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)

---

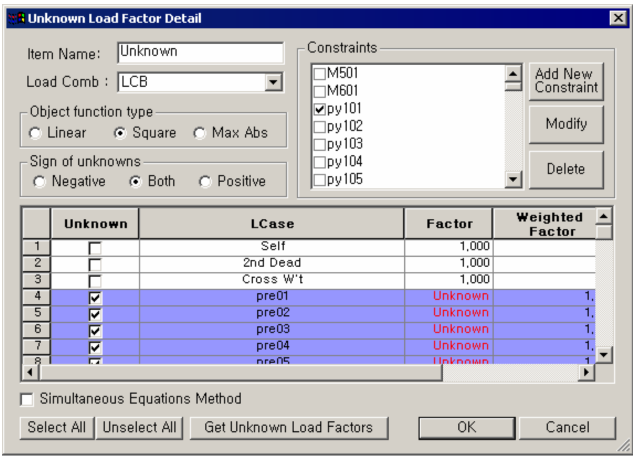



***Figure 16. Unknown Load Factors***

Specify the constraint conditions, which restrict the displacements 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.**

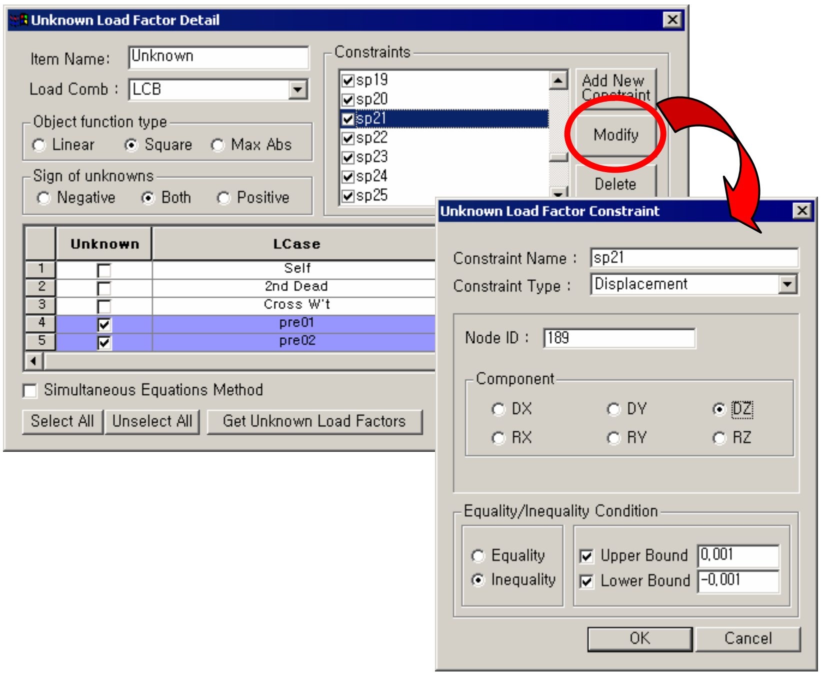



***Figure 17. Input Constraint Conditions***

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

[Unit: m]

| No | Node1 | Node2 | Type | SDx | SDy | SDz |
|---|---|---|---|---|---|---|
| py101 | DISP | 554 | RY | Inequality | 1.00E-06 | -1.00E-06 |
| py201 | DISP | 645 | RY | Inequality | 1.00E-06 | -1.00E-06 |
| sp01 | DISP | 103 | DZ | Inequality | -0.001 | 0.001 |
| sp02 | DISP | 107 | DZ | Inequality | 0.001 | -0.001 |
| sp03 | DISP | 111 | DZ | Inequality | 0.001 | -0.001 |
| sp04 | DISP | 115 | DZ | Inequality | 0.001 | -0.001 |
| sp05 | DISP | 119 | DZ | Inequality | 0.001 | -0.001 |
| sp06 | DISP | 123 | DZ | Inequality | 0.001 | -0.001 |
| sp07 | DISP | 127 | DZ | Inequality | 0.001 | -0.001 |
| sp08 | DISP | 131 | DZ | Inequality | 0.001 | -0.001 |
| sp09 | DISP | 135 | DZ | Inequality | 0.001 | -0.001 |
| sp10 | DISP | 139 | DZ | Inequality | 0.001 | -0.001 |
| sp11 | DISP | 143 | DZ | Inequality | 0.001 | -0.001 |
| sp12 | DISP | 147 | DZ | Inequality | 0.001 | -0.001 |
| sp13 | DISP | 151 | DZ | Inequality | 0.001 | -0.001 |
| sp14 | DISP | 155 | DZ | Inequality | 0 | -0.001 |
| sp15 | DISP | 159 | DZ | Inequality | 0 | -0.001 |
| sp16 | DISP | 163 | DZ | Inequality | 0.001 | -0.001 |
| sp17 | DISP | 173 | DZ | Inequality | 0.001 | -0.001 |
| sp18 | DISP | 177 | DZ | Inequality | 0 | -0.001 |
| sp19 | DISP | 181 | DZ | Inequality | 0 | -0.001 |
| sp20 | DISP | 185 | DZ | Inequality | 0.001 | -0.001 |
| sp21 | DISP | 189 | DZ | Inequality | 0.001 | -0.001 |
| sp22 | DISP | 193 | DZ | Inequality | 0.001 | -0.001 |
| sp23 | DISP | 197 | DZ | Inequality | 0.001 | -0.001 |
| sp24 | DISP | 201 | DZ | Inequality | 0.001 | -0.001 |
| sp25 | DISP | 205 | DZ | Inequality | 0.001 | -0.001 |
| sp26 | DISP | 209 | DZ | Inequality | 0.001 | -0.001 |
| sp27 | DISP | 213 | DZ | Inequality | 0.001 | -0.001 |
| sp28 | DISP | 217 | DZ | Inequality | 0.001 | -0.001 |
| sp29 | DISP | 221 | DZ | Inequality | 0.001 | -0.001 |
| sp30 | DISP | 225 | DZ | Inequality | 0.001 | -0.001 |
| sp31 | DISP | 229 | DZ | Inequality | 0.001 | -0.001 |
| sp32 | DISP | 233 | DZ | Inequality | 0.001 | -0.001 |
| sp33 | DISP | 234 | DZ | Inequality | 0.001 | -0.001 |
| sp34 | DISP | 238 | DZ | Inequality | 0.001 | -0.001 |
| sp35 | DISP | 242 | DZ | Inequality | 0.001 | -0.001 |
| sp36 | DISP | 246 | DZ | Inequality | 0.001 | -0.001 |
| sp37 | DISP | 250 | DZ | Inequality | 0.001 | -0.001 |

| | | | | | | |
|---|---|---|---|---|---|---|
| sp38 | DISP | 254 | DZ | Inequality | 0.001 | -0.001 |
| sp39 | DISP | 258 | DZ | Inequality | 0 | -0.001 |
| sp40 | DISP | 262 | DZ | Inequality | 0 | -0.001 |
| sp41 | DISP | 266 | DZ | Inequality | 0 | -0.001 |
| sp42 | DISP | 270 | DZ | Inequality | 0.001 | -0.001 |
| sp43 | DISP | 280 | DZ | Inequality | 0.001 | -0.001 |
| sp44 | DISP | 284 | DZ | Inequality | 0 | -0.001 |
| sp45 | DISP | 288 | DZ | Inequality | 0 | -0.001 |
| sp46 | DISP | 292 | DZ | Inequality | 0 | -0.001 |
| sp47 | DISP | 296 | DZ | Inequality | 0.001 | -0.001 |
| sp48 | DISP | 300 | DZ | Inequality | 0.001 | -0.001 |
| sp49 | DISP | 304 | DZ | Inequality | 0.001 | -0.001 |
| sp50 | DISP | 308 | DZ | Inequality | 0.001 | -0.001 |
| Sp51 | DISP | 312 | DZ | Inequality | 0.001 | -0.001 |

**Refer to the portion on the optimization technique of unknown loads for an explanation on calculation of Unknown Load Factors given in "Analysis for Civil Structures".**

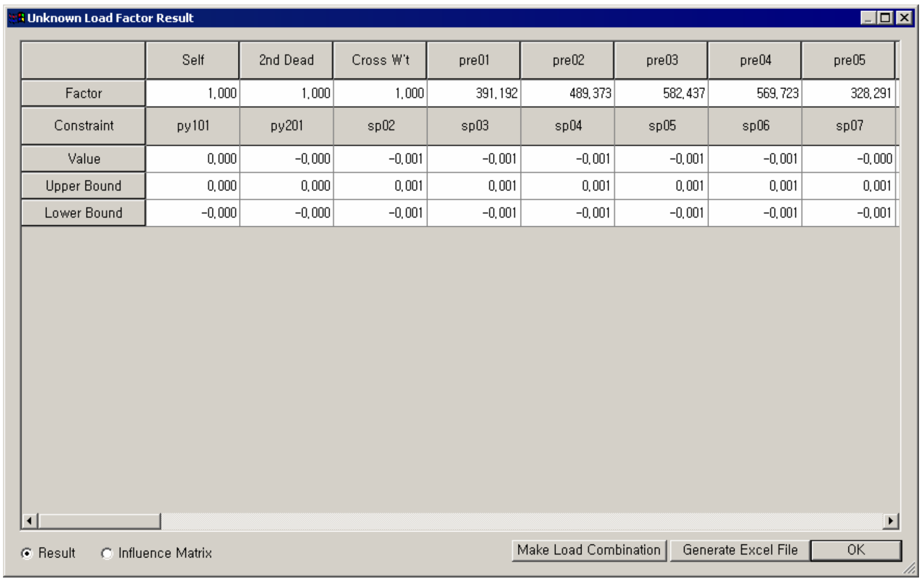
We now check the constraints used to calculate the cable initial pretensions and unknown load factors in *Unknown Load Factor Result*.

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 | 391.192 | 489.373 | 582.437 | 569.723 | 328.291 |
| Constraint | py101 | py201 | sp02 | sp03 | sp04 | sp05 | sp06 | sp07 |
| Value | 0.000 | -0.000 | -0.001 | -0.001 | -0.001 | -0.001 | -0.001 | -0.000 |
| 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.

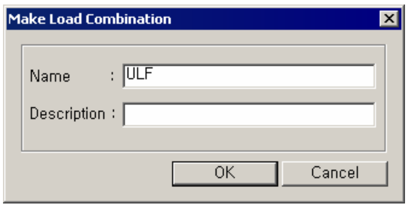



***Figure 19. Automatic generation of "ULF" load combination using the unknown load factors***

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

Results / ***Combinations***

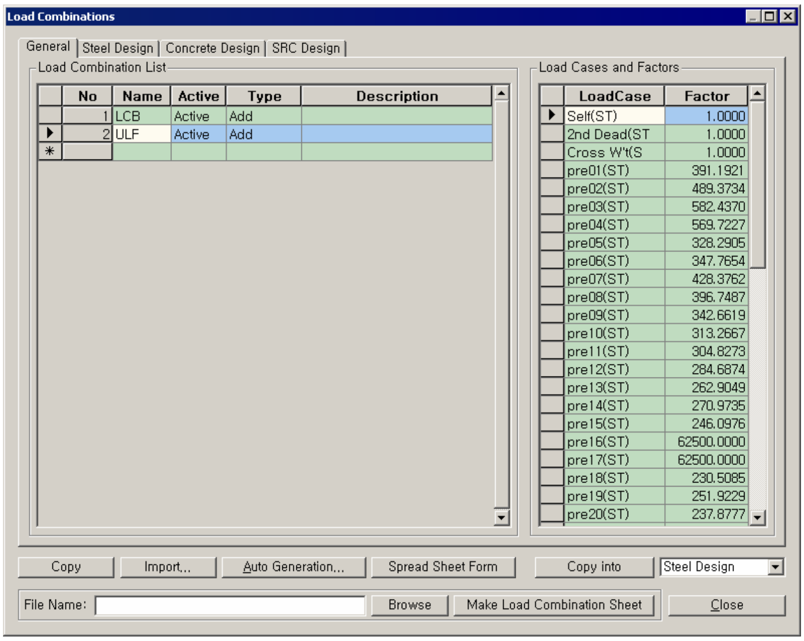


***Figure 20. Automatic Generation of "ULF" Load Combination that uses the Unknown Load Factors***

# Review Analysis Results

## Review deformed shape.

Review the deformed shape for the “ULF” load combination calculated from the initial pre-tensions using the Unknown Load Factors.

---

Result / Deformations /  ***Deformed Shape***

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

Components > **DZ**

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

---

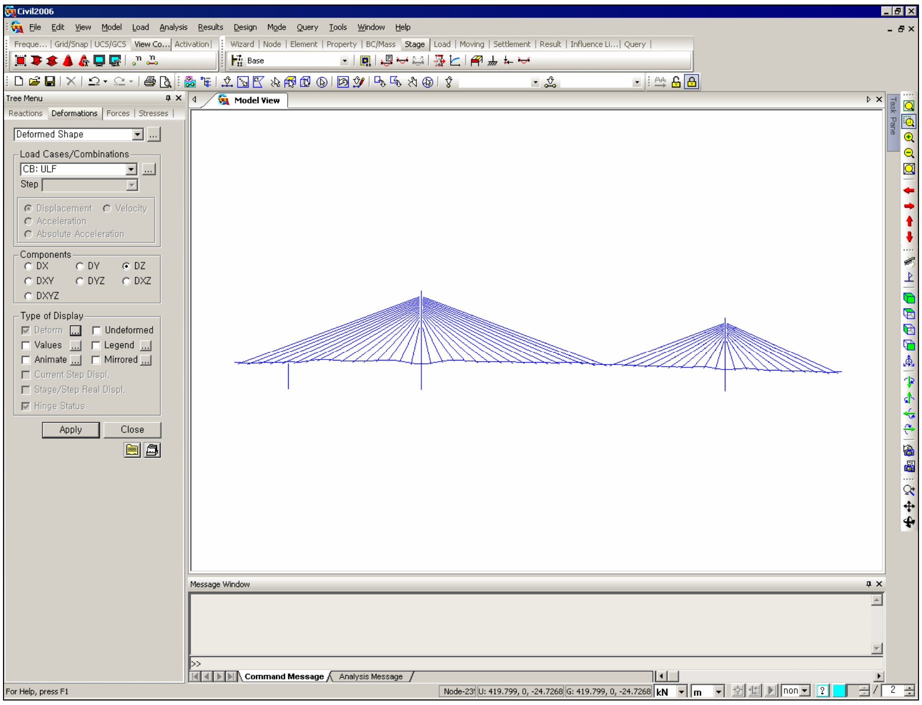



***Figure 21. Deformed Shape Results***

## Review Member Forces

Review the member forces for the "ULF" load combination.

Result / Force / Beam Diagram
  Load Cases / Combinations > **CB:ULF**
  Components > **DXYZ**
  Type of Display > **Undeformed** (on) ; **Legend** (on)

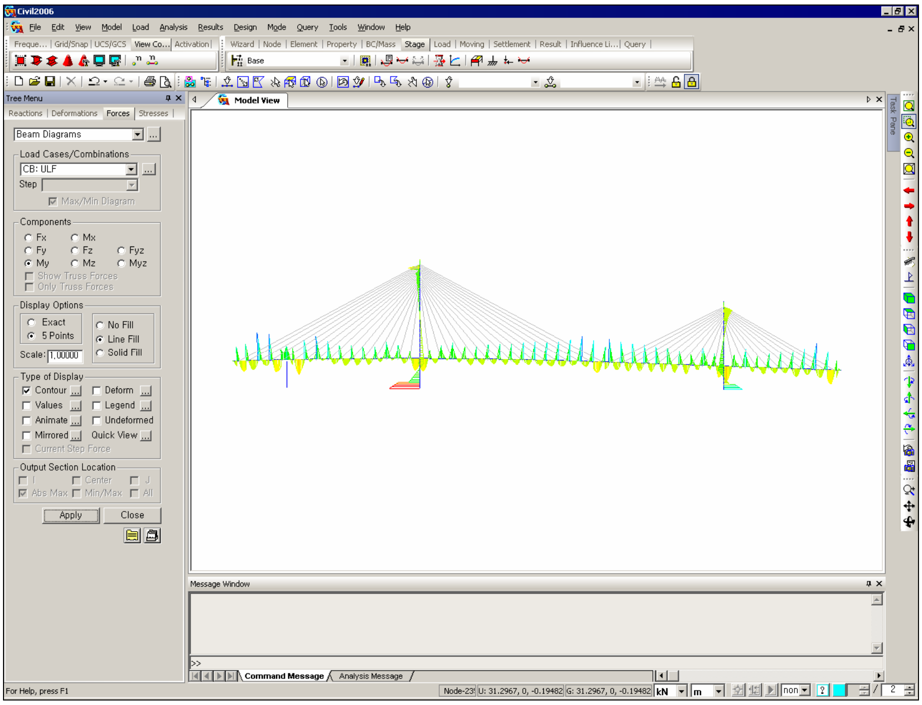


***Figure 22. Review Member Forces***