Tutorial

Final and Construction Stage Analysis for a Cable-Stayed Bridge

Civil

CONTENTS

Summary

Cable-stayed bridges are structural systems effectively composing cables, main girders and towers. This bridge form has a beautiful appearance and easily fits 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 structures that require a high degree of technology for both design and construction, and hence demand sophisticated structural analysis and design techniques when compared with other types of conventional bridges.

In addition to static analysis for dead and live loads, a dynamic analysis must also be performed to determine eigenvalues. Also moving load, earthquake load and wind load analyses are essentially required for designing a cable-stayed bridge.

To determine the cable prestress forces that are introduced at the time of cable installation, the initial equilibrium state for dead load at the final stage must be determined first. Then, construction stage analysis according to the construction sequence is performed.

This tutorial explains techniques for modeling a cable-stayed bridge, calculating initial cable prestress forces, performing construction stage analysis and reviewing the output data. The model used in this tutorial is a three span continuous cable-stayed bridge composed of a 220 *m* center span and 100 *m* side spans. Fig. 1 below shows the bridge layout.

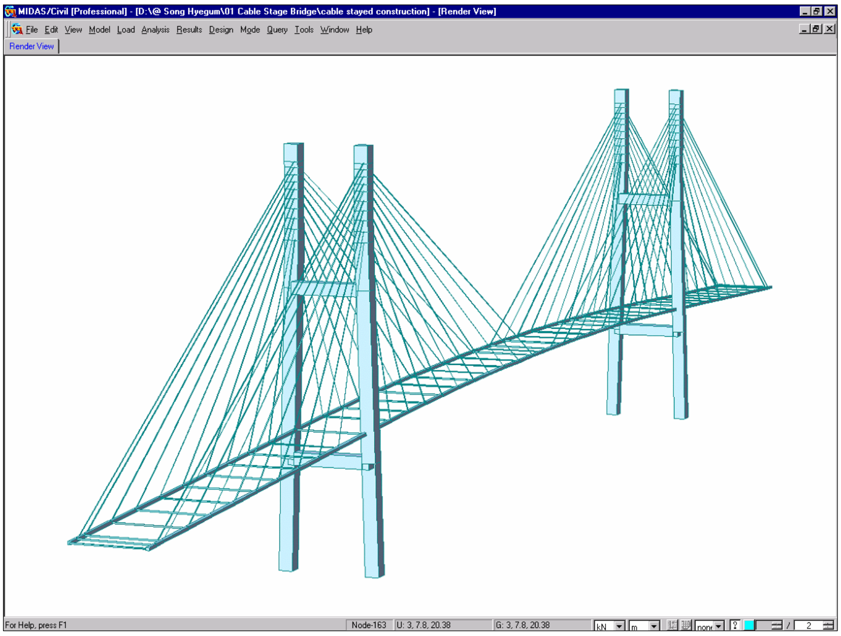


Fig. 1 Cable-stayed bridge analytical model

Bridge Dimensions

The bridge model used in this tutorial is simplified because its purpose is to explain the analytical sequences, and so its dimensions may differ from those of a real structure.

The dimensions and loadings for the three span continuous cable-stayed bridge are as follows:

Bridge type	Three span continuous cable-stayed bridge (self-anchored)
Bridge length	L = 100 *m*+220 *m*+100 *m* = 420 *m*
Bridge Width	B = 15.6 *m* (2 lanes)
Lanes	2 lane structure

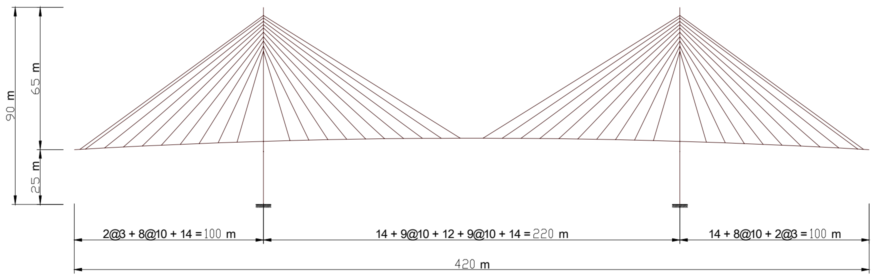


Fig. 2 General layout

Loading

- **Self-weight:** Automatically calculated within the program
- **Additional dead load:** pavement, railing and parapets
- **Initial cable prestress forces:** Cable prestress forces that satisfy initial equilibrium state at the final stage

We input initial cable prestress force values, which can be calculated by built-in optimization technique in MIDAS/Civil.

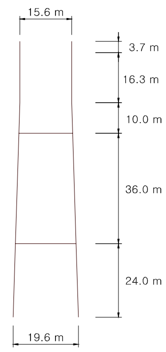


Fig. 3 Tower layout

Working Condition Setting

To perform the final stage analysis for the cable-stayed bridge, open a new file and save it as ‘**Cable Stayed Backward**’, and start modeling. Assign ‘**m**’ for length unit and ‘**kN**’ for force unit. This unit system can be changed any time during the modeling process for user’s convenience.

File / *New Project*
File / *Save* **(Cable Stayed Backward)**

Tools / ***Unit System***
Length>**m**; Force (Mass)>**kN (ton)** ↵

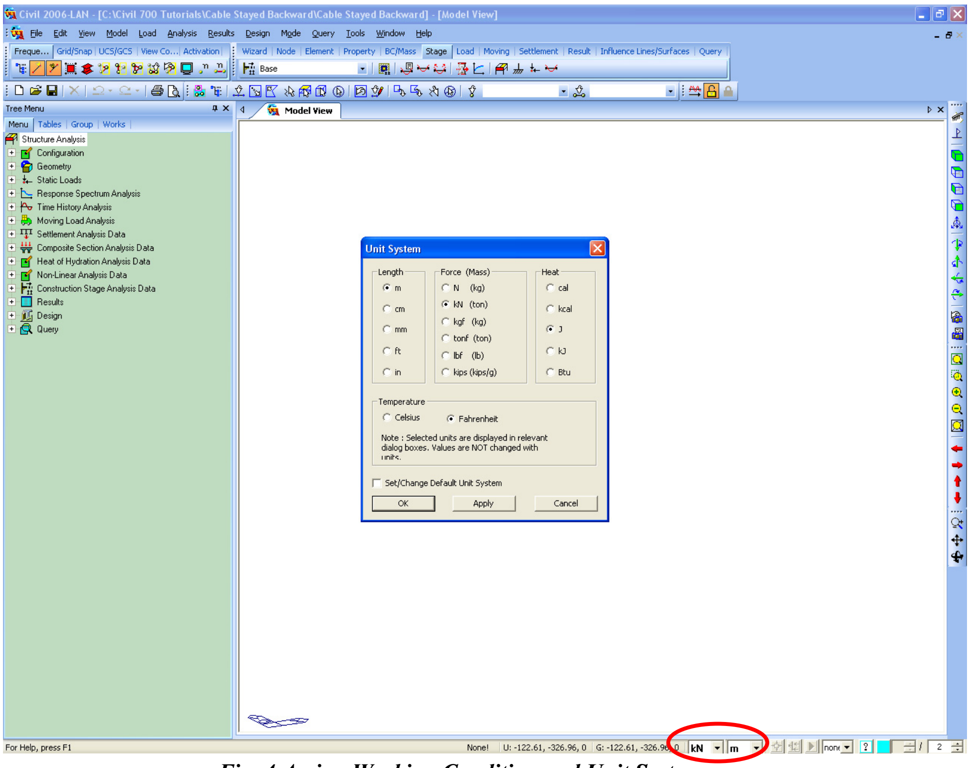


Fig. 4 Assign Working Condition and Unit System

Definition of Material and Section Properties

Input material properties for the cables, main girders, towers, cross beams between the main girders and tower cross beams. Click Add button under Material tab in Properties dialog box.

Model / Properties / ***Material***

Material ID (1); Name **(Cable);** Type of Design> **User Defined;**

User Defined>Standard > **None;** Type of Material> **Isotropic;**

Analysis Data>Modulus of Elasticity **(1.9613e8);** Poisson's Ratio **(0.3)**

Weight Density **(77.09)** ↵

Input material properties for the main girders, towers (pylons), cross beams between the main girders and tower cross beams similarly. The input values are shown in Table 1.

Table 1 Material Properties

Material ID	Name	Modulus of Elasticity (kN/m^2)	Poisson's Ratio	Weight Density (kN/m^3)
1	Cable	1.9613×10^8	0.3	77.09
2	Girder	1.9995×10^8	0.3	77.09
3	Pylon	2.78×10^7	0.2	23.56
4	CBeam_Girder	1.9613×10^8	0.3	77.09
5	CBeam_Pylon	2.78×10^7	0.2	23.56

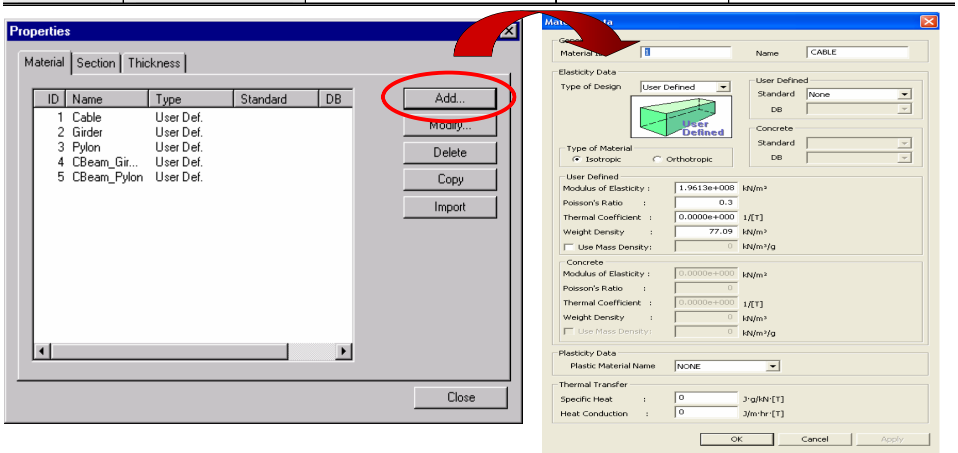


Fig. 5 Defined Material Properties

Input section properties for the cables, main girders, towers (pylons), cross beams between the main girders and tower cross beams. Click Add button under Section tab in Properties dialog box.

Model / Properties / *Section*

Value tab

Section ID **(1)**; Name **(Cable);** Built-Up Section (on); Consider Shear Deformation (on); Section Shape>**Solid Rectangle;** Section Properties>Area **(0.0052)** ↵

Input section properties for the main girders, towers (pylons), cross beams between the main girders and tower cross beams similarly. The values are shown in Table 2.

Table 2 Section Properties

Section ID	Name	Area (m^2)	Ixx (m^4)	Iyy (m^4)	Izz (m^4)
1	Cable	0.0052	0.0	0.0	0.0
2	Girder	0.3092	0.007	0.1577	4.7620
3	Pylon	9.2000	19.51	25.5670	8.1230
4	CBeam_Girder	0.0499	0.0031	0.0447	0.1331
5	CBeam_Pylon	7.2000	15.79	14.4720	7.9920

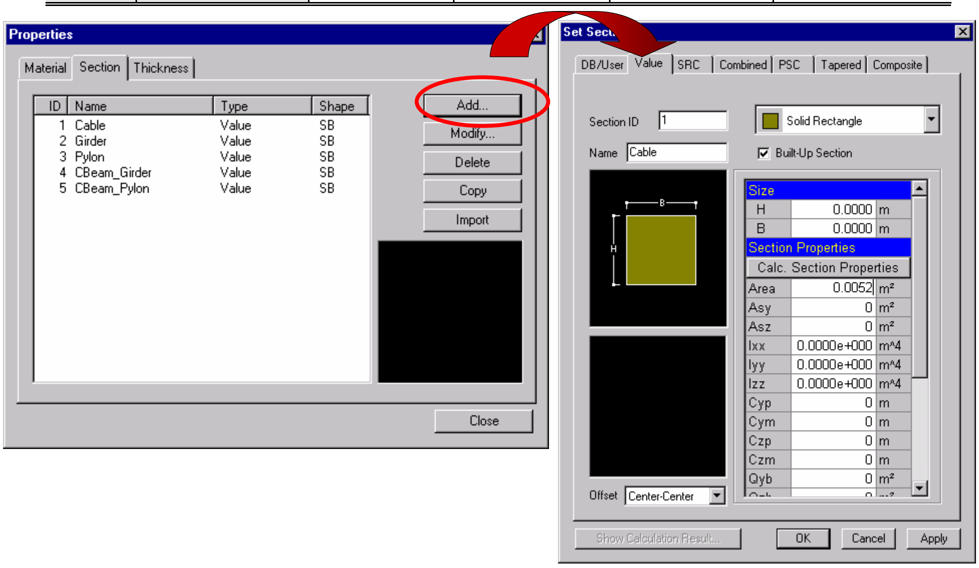


Fig. 6 Defined Section Properties

Final Stage Analysis

After completion of the final stage modeling for the cable-stayed bridge, we calculate the initial cable prestress forces for self-weights and additional dead loads. After that, we perform initial equilibrium state analysis with the calculated initial prestress forces.

To perform structural modeling of the cable-stayed bridge, we first generate a 2D model by Cable Stayed Bridge Wizard provided in **MIDAS/Civil**. We then copy the 2D model symmetrically to generate a 3D model. Initial cable forces introduced in the final stage can easily be calculated by the Unknown Load Factors function, which is based on an optimization technique. The final model of the cable-stayed bridge is shown in Fig. 7.

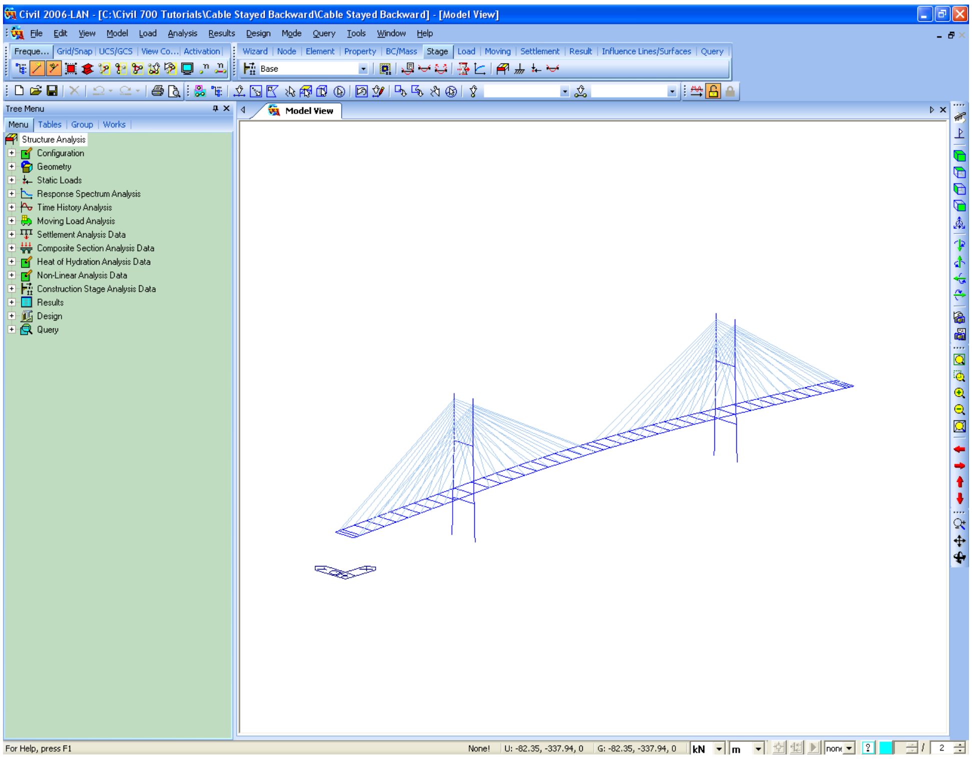


Fig. 7 Final Model for Cable-Stayed Bridge

Bridge Modeling

In this tutorial, the analytical model for the final stage analysis will be completed first and subsequently analyzed. The final stage model will then be saved under a different name, and then using this model the construction stage model will be developed.

Modeling process for the final stage analysis of the cable-stayed bridge is as follows:

1. 2D Model Generation by Cable-Stayed Bridge Wizard
2. Tower Modeling
3. Expand into a 3D Model
4. Main Girder Cross Beam Generation
5. Tower Bearing Generation
6. End Bearing Generation
7. Boundary Condition Input
8. Initial Cable Prestress Force Calculation by Unknown Load Factors
9. Loading Condition and Loading Input
10. Perform Structural Analysis
11. Unknown Load Factors Calculation

2D Model Generation

MIDAS/Civil provides a Cable-Stayed Bridge Wizard function that can automatically generate a 2D cable-stayed bridge model based on basic structural dimensions of the bridge. Input basic structural dimensions of the cable-stayed bridge in the Cable-Stayed Bridge Wizard as follows.

Using the Cable Stayed Bridge Wizard function, a 2D model can be generated automatically based on material and section properties of the cables, main girders and towers.

Front View ***Point Grid*** (off) ***Point Grid Snap*** (off)
Line Grid Snap (off) ***Node Snap*** (on) ***Elements Snap*** (on)

Model / Structure Wizard / ***Cable Stayed Bridge***

Type>**Symmetric Bridge**
A>X (m) **(0)** ; Z (m) **(25)** ; B>X (m) **(100)** ; Z (m) **(90)**
Height>H1 (m) **(90)**
Material>Cable>**1:Cable** ; Deck>**2:Girder** ; Tower>**3:Pylon**
Section>Cable>**1:Cable** ; Deck>**2:Girder** ; Tower>**3:Pylon**
Select Cable & Hanger Element Type>**Truss**
Shape of Deck (on)>Left Slope (%) **(5)** ; Arc Length (m) **(220)**
Cable Distances & Heights
Left>Distance (m) **(3, 8@10, 14)** ; Height (m) **(1.2, 3@1.5, 3@2, 2@2.3, 45)**
Center>Distance (m) **(14, 9@10, 12, 9@10, 14)** ↵

If Truss is selected as the element type for cables, truss elements are generated; and if Cable is selected, it will automatically generate equivalent truss elements for linear analysis and elastic catenary cable elements for nonlinear analysis.

Input vertical slopes as 5% for both side spans, and use a circular curve for the center span, which is continuous from each side span.

If Drawing in View option is selected, the 2D model shape, which will be generated based on the input dimensions, can be viewed in the wizard window.

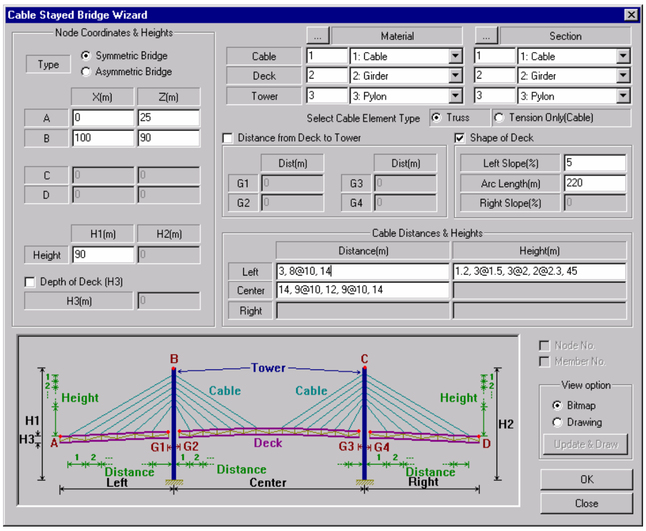


Fig. 8 Cable-Stayed Bridge Wizard Dialog Box

Girder Modeling

Duplicated nodes will be generated at the tower locations since the Cable-Stayed Bridge Wizard will generate the main girders as a simple beam type for the side and center spans. This tutorial example is a continuous self-anchored cable-stayed bridge. We will use the Merge Node function to make the girders continuous at the tower locations.

Node Number (on) ***Front View***

Model / Nodes / ***Merge Nodes***

Merge>**All**

Tolerance **(0.001)**

Remove Merged Nodes (on) ↵

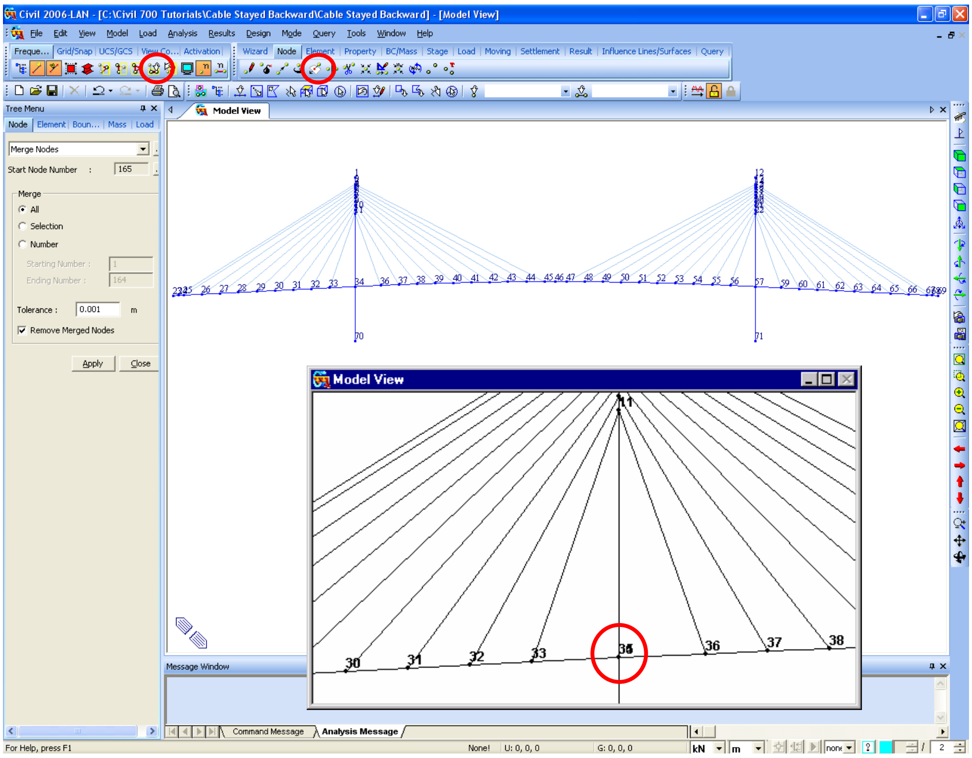


Fig. 9 Generated 2D Model of the Cable-Stayed Bridge

Tower Modeling

The upper and lower widths of the towers are 15.600 m and 19.600 m respectively. To model the inclined towers, the lower parts of the towers will be moved 2m in the –Y direction using the Translate Node function.

Left View ***Auto Fitting*** ***Node Number*** (off)

Model / Nodes / ***Translate Nodes***

Select Window (Nodes: A in Fig. 10)

Mode>**Move**; Translation>**Equal Distance**; dx, dy, dz **(0, -2, 0)** ↵

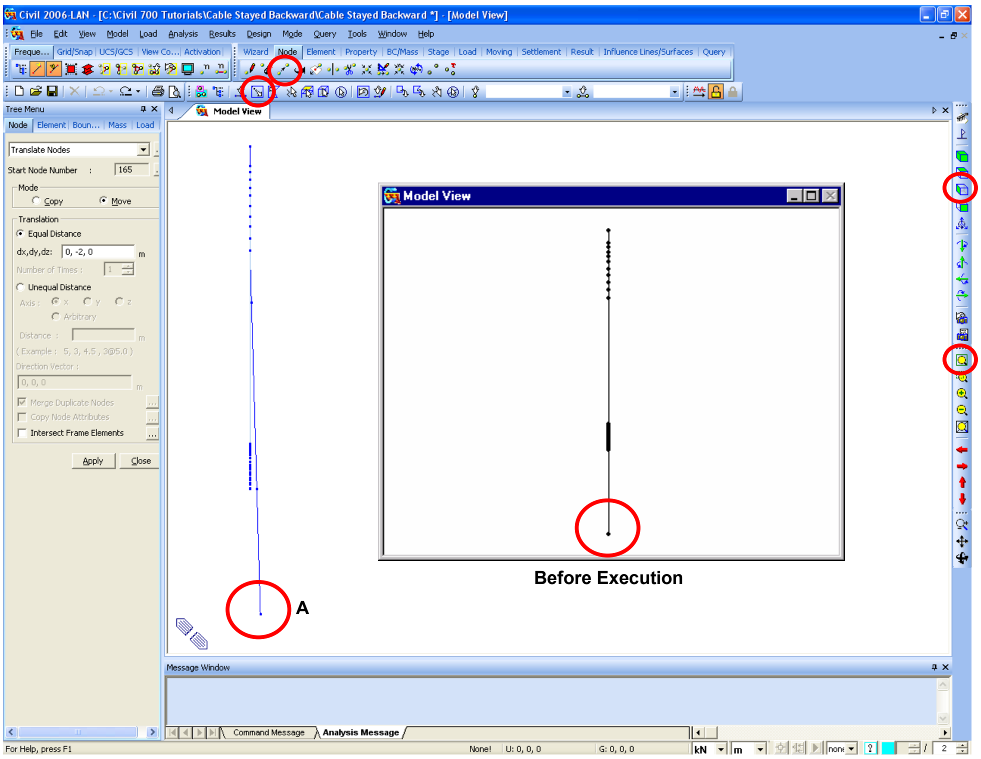


Fig. 10 Arrangement of Inclined Towers

Detailed explanation for Beta Angle can be found in "Tutorial for 3D Simple 2-Bay Frame" or "Truss Element" parts in "Types of Elements and Important Considerations" in "Analysis for Civil Structures".

Note that the local coordinate system of the inclined tower elements is changed with the movement of the nodes. The y & z-axes become rotated by 90° when the element is inclined - this is a built-in feature of the program. To revert y & z axes to their original positions, the Beta Angle is changed to -90°. By changing the Beta Angle of the tower elements to -90°, we also make the local element coordinate systems of the upper and lower tower elements coincide for the ease of reviewing analysis results.

Display

Element > **Local Axis** (on) ↵

Model / Elements / ***Change Element Parameters***

Select Intersect (Elements: A in Fig. 11)

Parameter Type > **Element Local Axis** (on) > **Beta Angle**

Beta Angle (Deg) **(-90)** ↵

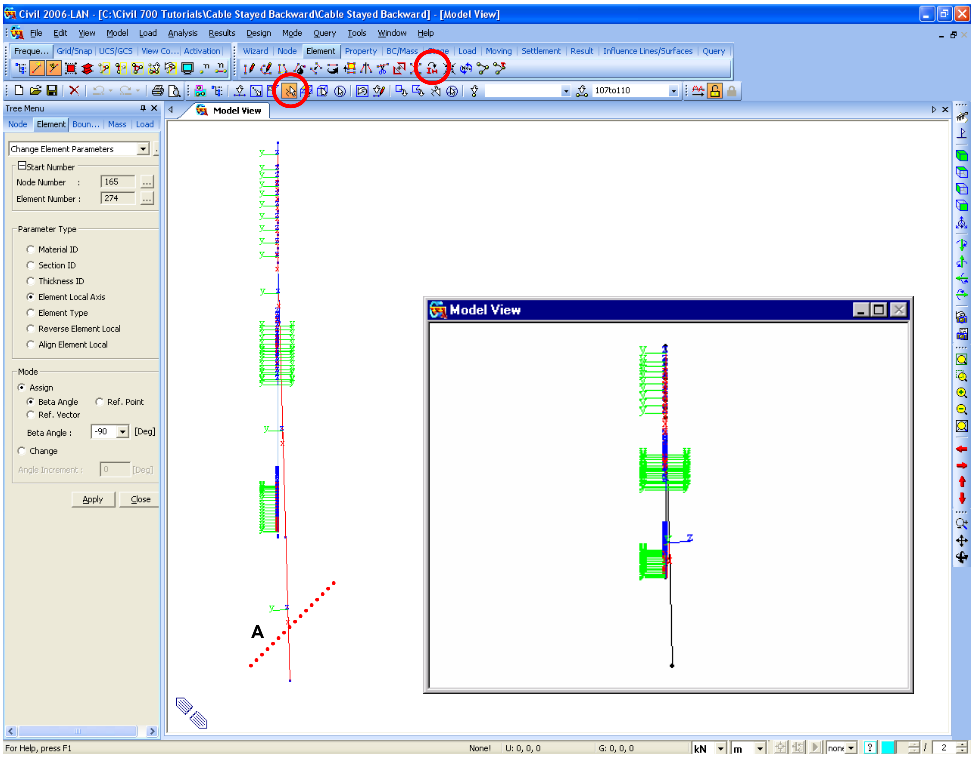


Fig. 11 Local Element Axis Transformation for Tower Elements

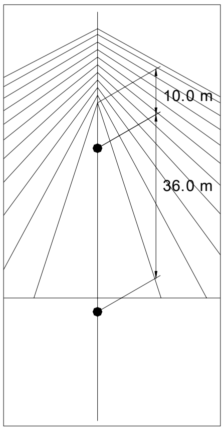
To generate the tower cross beams, divide the tower elements in the Z-axis direction by *Divide Elements*.

Model / Elements / ***Divide Elements***

Select Previous

Divide > Element type > **Frame**; **Unequal Distance**

x (m) **(10, 36)** ↵



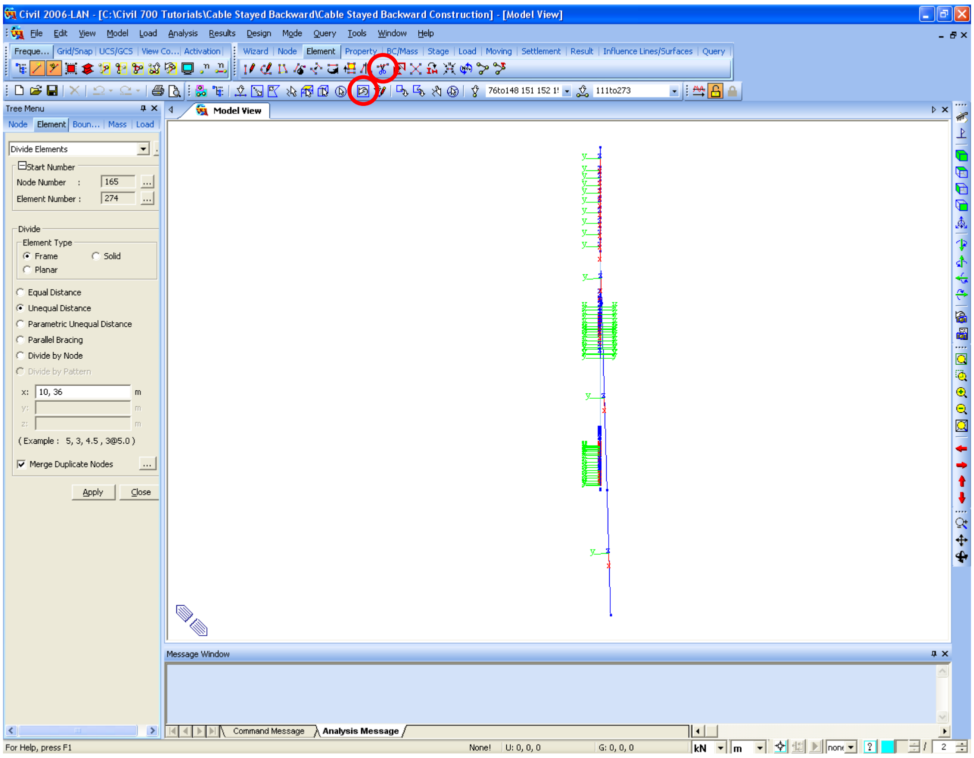


Fig. 12 Division of Tower Elements

3D Model Generation

To generate the 3D model, we move the 2D model –7.800*m* in the Y direction, as the bridge width is 15.600 m.

Model / Nodes / ***Translate Nodes***

Select All

Mode>**Move**; Translation>**Equal Distance**; dx, dy, dz **(0, -7.8, 0)** ↵

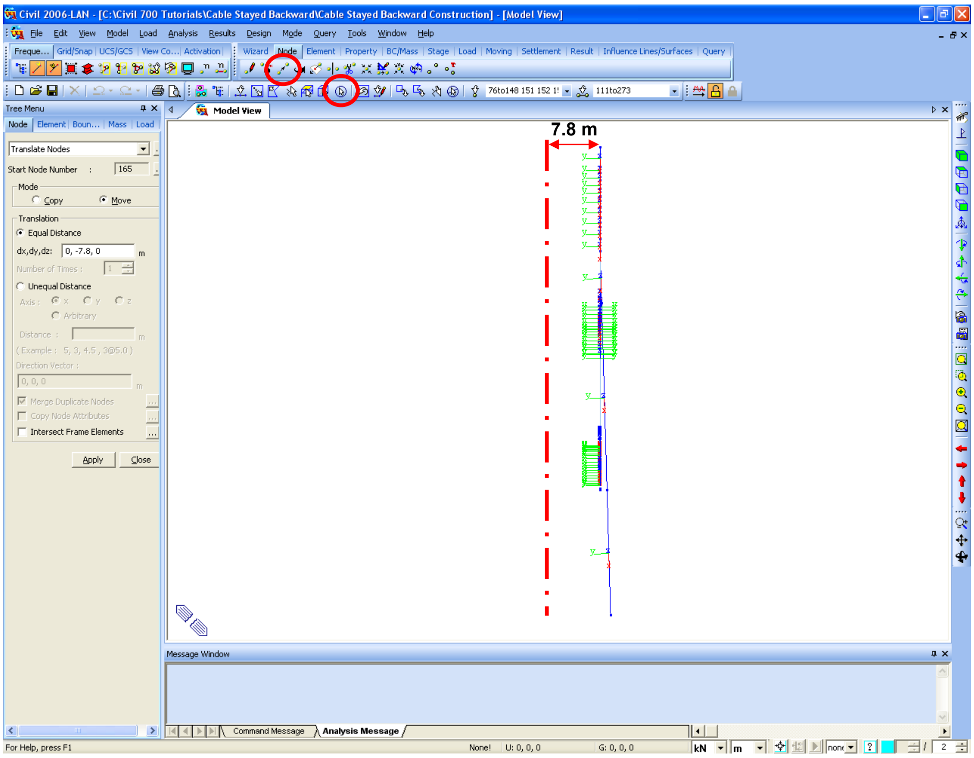


Fig. 13 Moving 2D Model –7.8 m in the Y direction

We now copy the cables, main girders and towers symmetrically with respect to the centerline of the bridge. At this time, we will check on Mirror Element (Beta) Angle to match the local coordinates of the copied towers to those of the origin towers.

Model / Elements / *Mirror Elements*

Select All

Mode>**Copy**

Reflection>**z-x plane (m) (0)**

Copy Element Attributes (on) ; **Mirror Beta Angle** (on) ↵

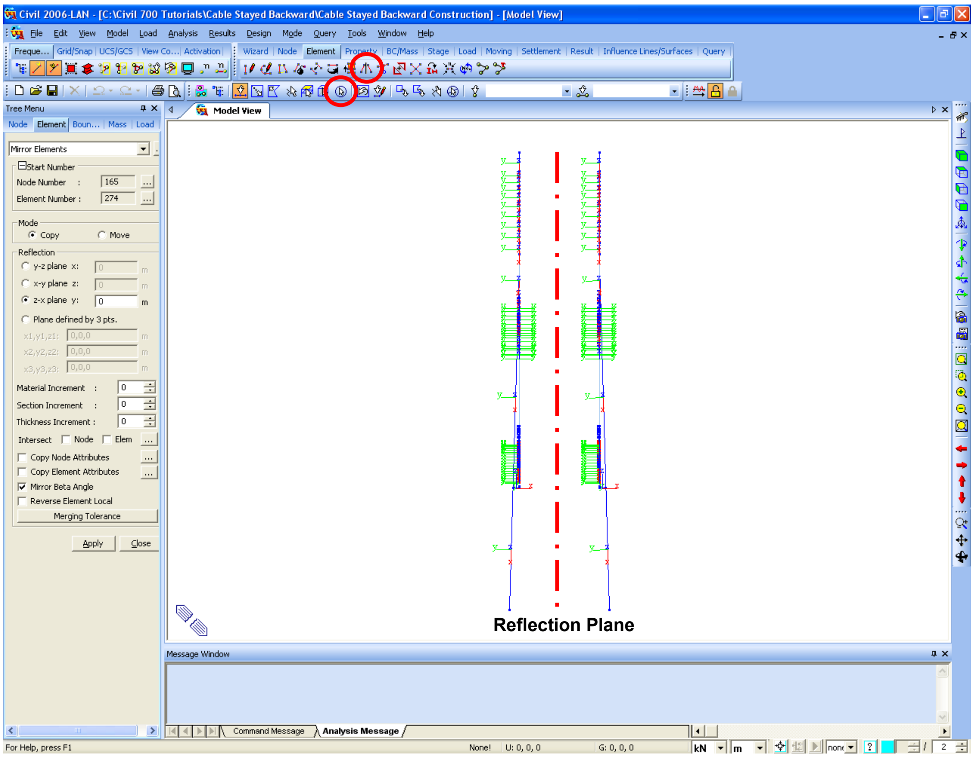


Fig. 14 Generating 3D Model

Main Girder Cross Beam Generation

Clear Display for the element coordinate axes and then generate the crossbeams between the main girders by the **Extrude Element** function, which creates line elements from nodes.

Top View
Display
Element> **Local Axis** (off) ↵

Model / Elements / ***Extrude Elements***
Select Identity - Nodes
Select Type>Material>**2: Girder** ; **Nodes** (on), **Elements** (on) ↵
Unselect window (Nodes: A in Fig. 15)
Extrude Type>**Node → Line Element**
Element Attribute>Element Type>**Beam**
Material>**4: CBeam_Girder**
Section>**4: CBeam_Girder**
Generation Type>**Translate**
Translation>**Equal Distance**; dx, dy, dz **(0, -15.6, 0)**
Number of Times **(1)** ↵

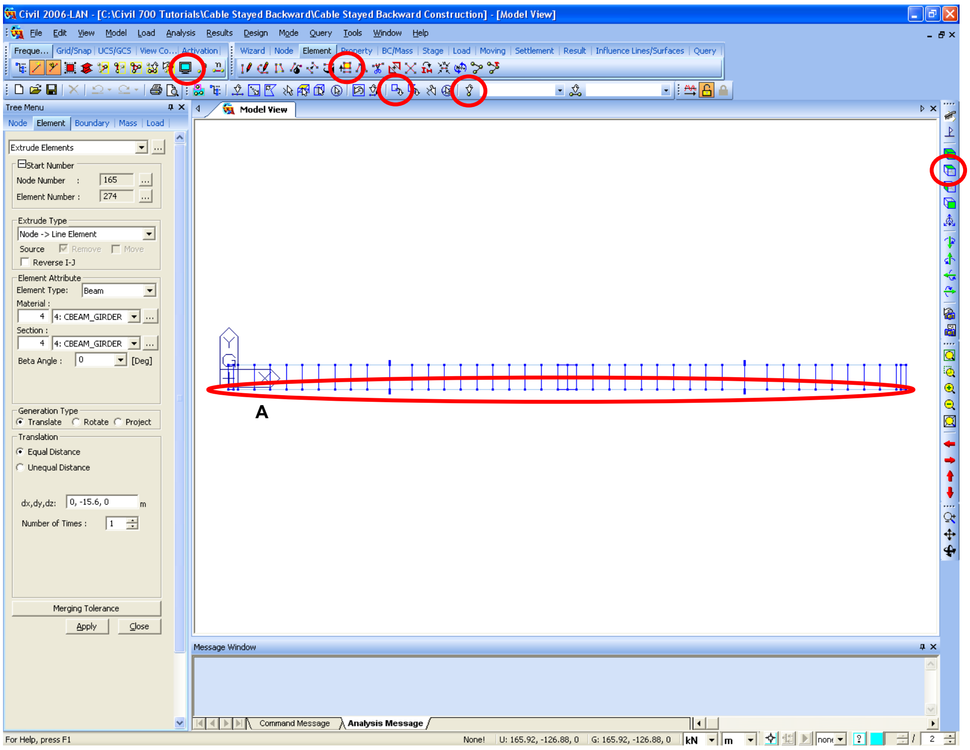


Fig. 15 Main Girder Cross Beam Generation

Tower Cross Beam Generation

Before generating the tower cross beams, we activate only the tower elements for effective modeling.

Front View
Select Single (A in Fig. 16)
Activate

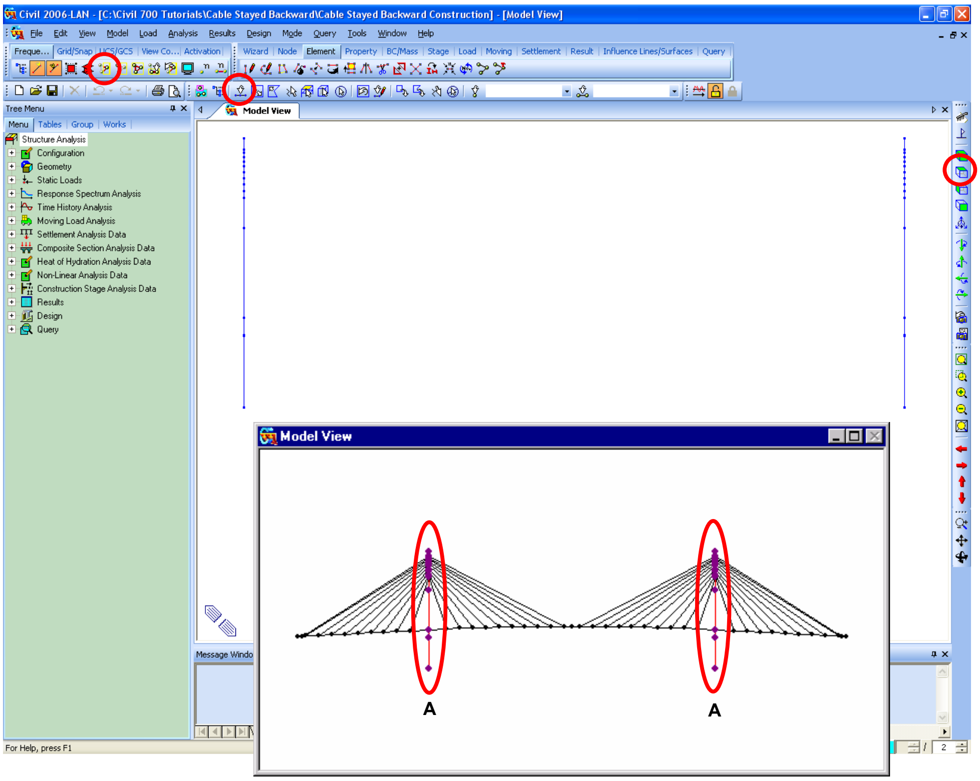


Fig. 16 Selecting Tower Elements

Generate the tower cross beams by the ***Create Element*** function.

Iso View ***Node Number*** **(on)** ***Element Snap*** **(off)**

Model / Elements / ***Create Elements***

Element type>**General Beam/Tapered Beam**

Material>**5: CBeam_Pylon**

Section>**5: CBeam_Pylon**

Nodal Connectivity **(142, 72) (145, 73) (144, 74) (147, 75)**

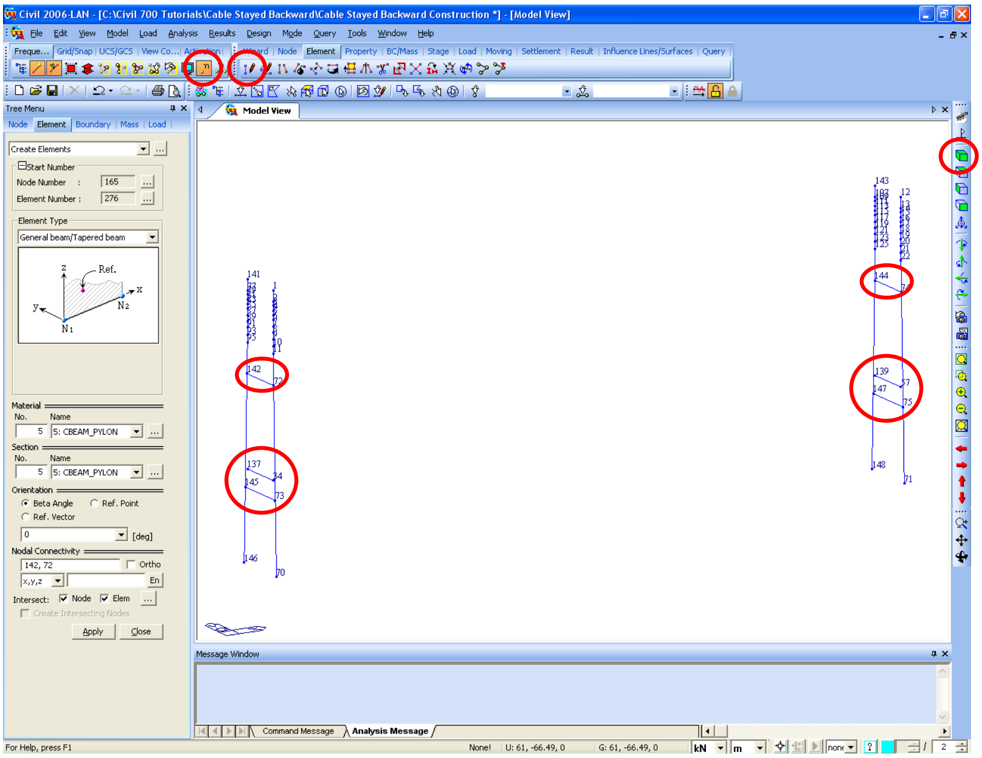


Fig. 17 Tower Cross Beam Generation

Tower Bearing Generation

Create new nodes at the tower bearing locations by the ***Project Nodes*** function.

Model / Nodes / ***Project Nodes***

Mode > **Copy**; Projection Type > **Project nodes on a plane**

Select Single (Nodes: **34, 137, 57, 139**)

Base Plane Definition > P1 **(145)** ;P2 **(73)** ; P3 **(75)** **;** Direction > **Normal**

Merge Duplicate Nodes (on); **Intersect Frame Elem.** (on) ↵

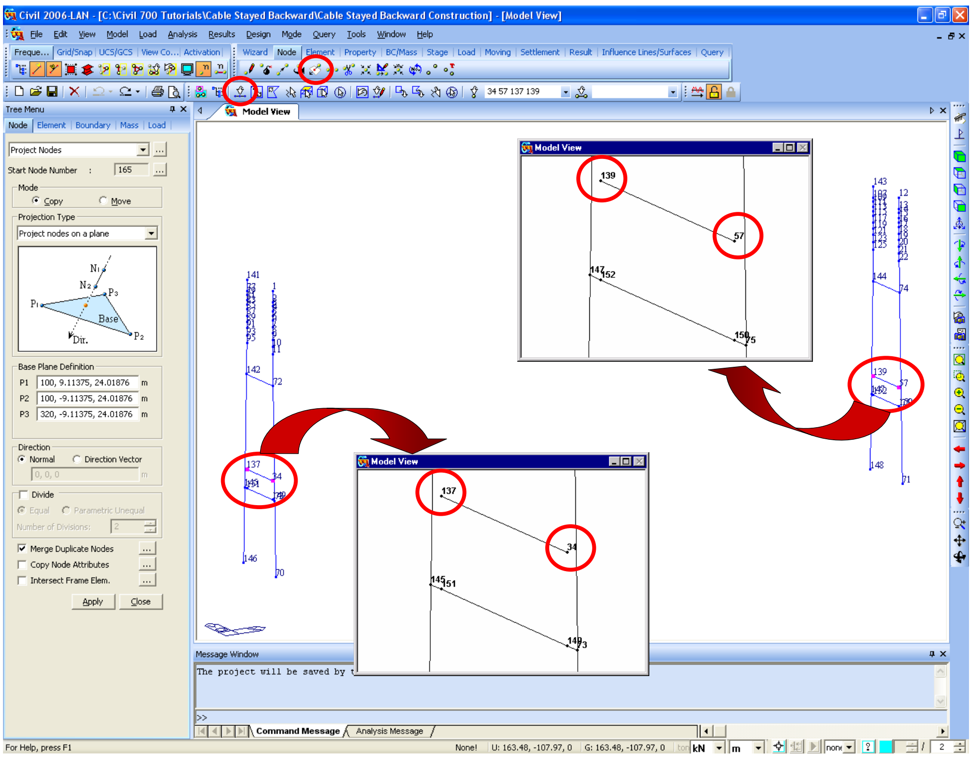


Fig. 18 Tower Bearing Generation

Generate nodes at the tower bearing locations using the ***Translate Nodes*** function to reflect the bearing heights.

Model / Nodes / ***Translate Nodes***

Select Single (Nodes: **149 to 152**)

Mode>**Copy**; Translation>**Equal Distance**

dx, dy, dz **(0, 0, 0.27)** ↵

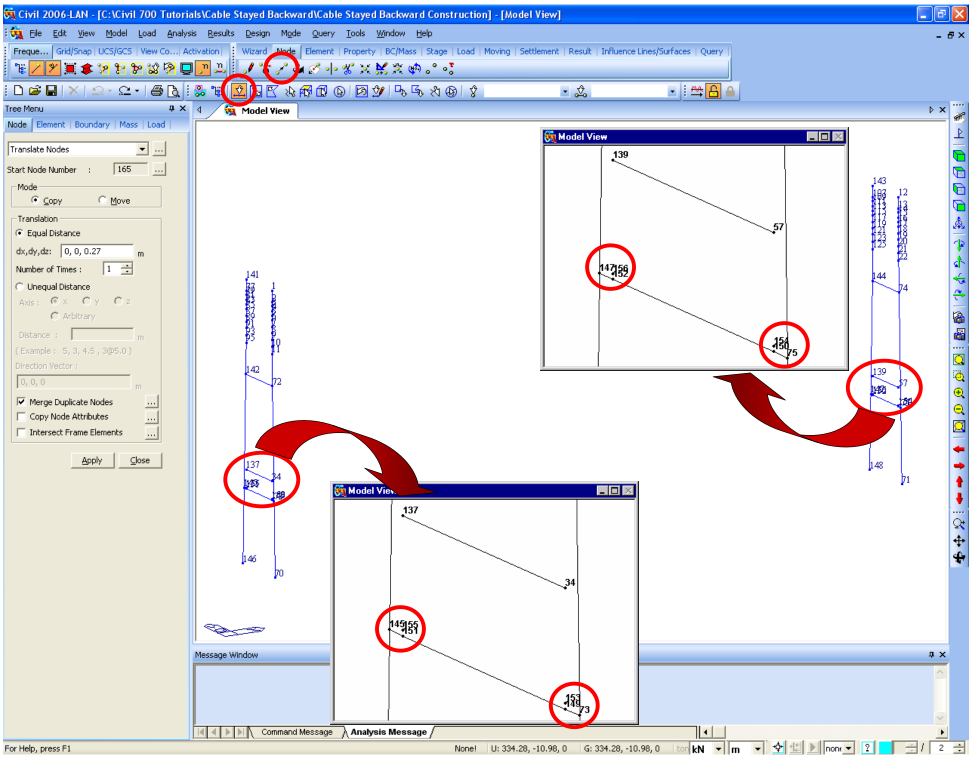


Fig. 19 Tower Bearing Location Generation

Model the tower bearings using the element link elements.

Bearing properties are as follows:
SDx: 199,736,032 kN/m
SDy: 73,373 kN/m
SDz: 73,373 kN/m

Model / Boundaries / ***Elastic Link***
Zoom Window (A in Fig. 20)
Options>**Add**; Link Type>**General Type**
SDx (kN/m) **(199736032)**; SDy (kN/m) **(73373)**; SDz (kN/m) **(73373)**
Copy Elastic Link (on)>Axis>**x;** Distances (m) **(220)**
2 Nodes **(151,155)**
2 Nodes **(149,153)**

Simultaneously input elastic link elements for both towers by entering tower spacing of 220 m.

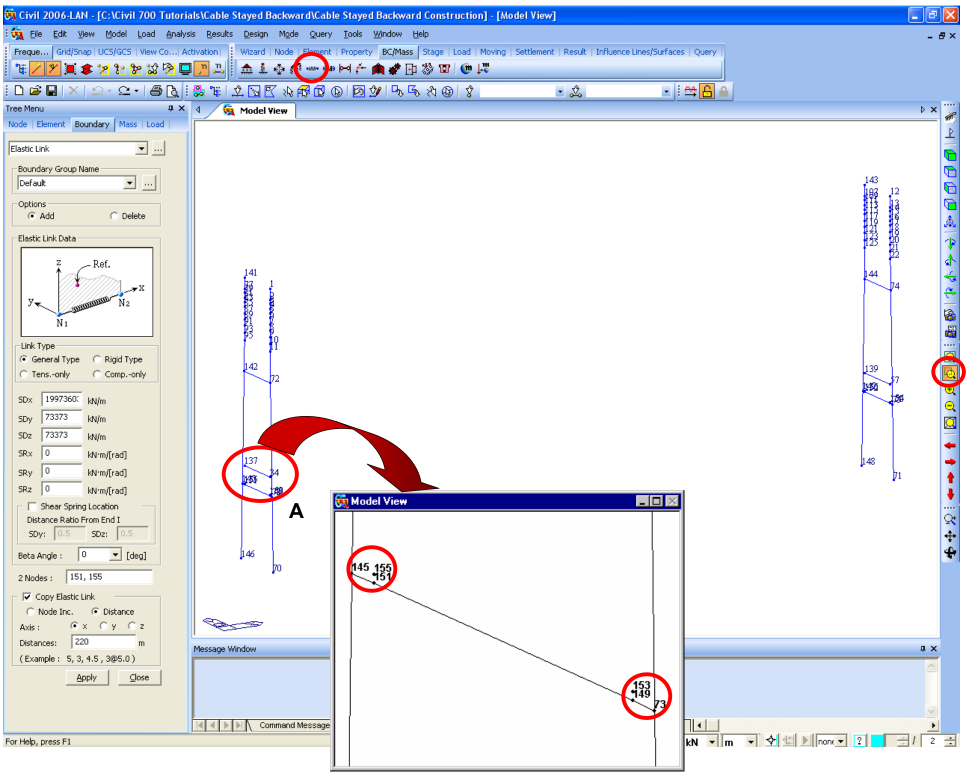


Fig. 20 Tower Bearing Generation

End Bearing Generation

Generate nodes at the end bearing locations using the ***Translate Nodes*** function.

Activate All

Model / Nodes / ***Translate Nodes***

Select Single (Nodes: **76, 24, 135, 68**)

Mode>**Copy**; Translation>**Unequal Distance**

Axis>**z;** Distance (m) **(-4.5, -0.27)** ↵

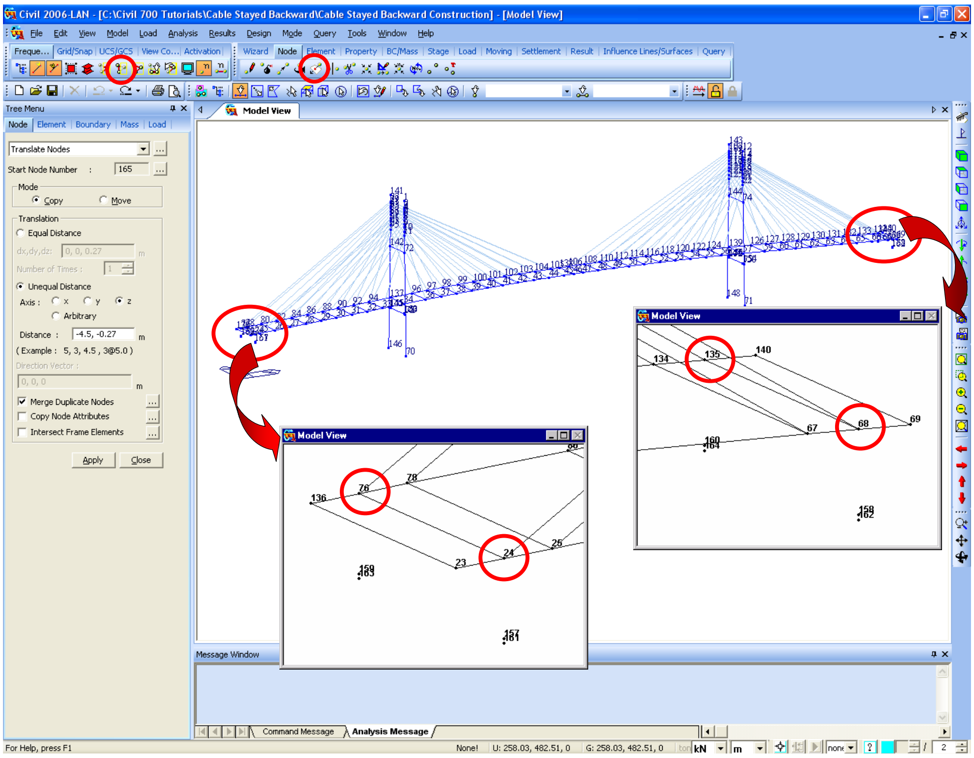


Fig. 21 Generating Nodes at the End Bearing Locations

Model the end bearings using the element link elements.

Bearing properties are as follows:
SDx: 199,736,032 kN/m
SDy: 73,373 kN/m
SDz: 73,373 kN/m

Model / Boundaries / ***Elastic Link***
Zoom Window (A in Fig. 22)
Options>**Add**; Link Type>**General Type**
SDx (kN/m) **(199736032)**; SDy (kN/m) **(73373)**; SDz (kN/m) **(73373)**
Copy Elastic Link (on) > Axis>**x**; Distances (m) **(414)**
2 Nodes **(159,163)**
2 Nodes **(157,161)**

Generate the elastic links simultaneously for the right end. The distance between the ends is 420-3*2= 414 m.

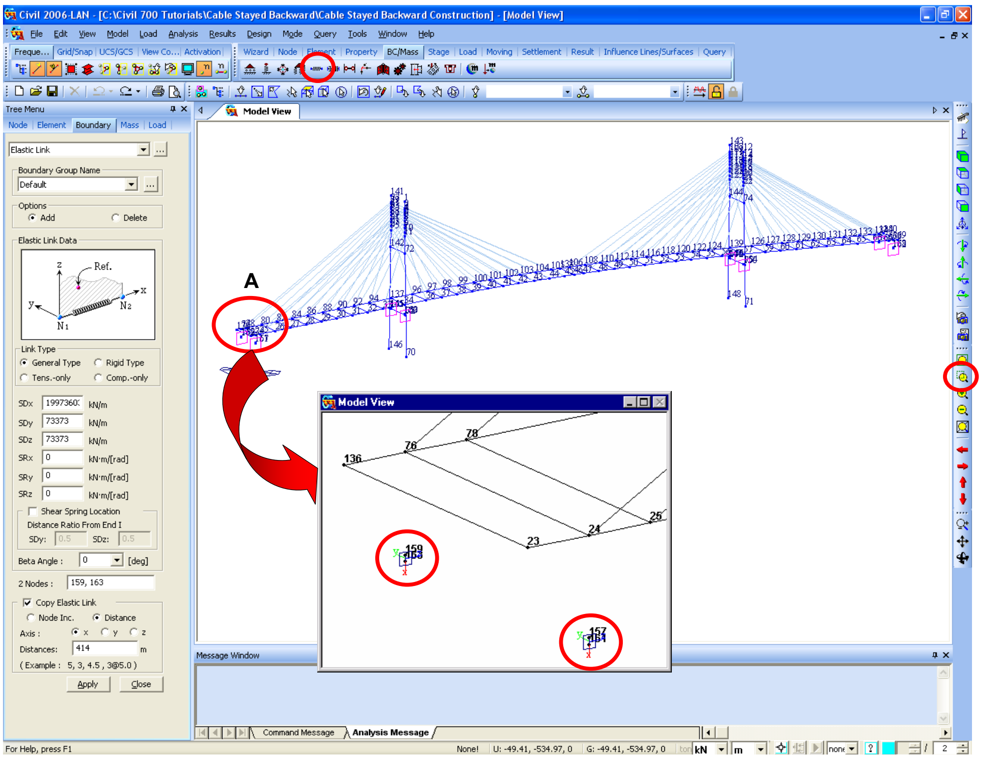


Fig. 22 Generating End Pier Bearings

Boundary Condition Input

Boundary conditions for the analytical model are as follows:

- Tower base, Pier base: Fixed condition (Dx, Dy, Dz, Rx, Ry, Rz)
- Connections between Main Girders and Bearings: Rigid Link (Dx, Dy, Dz, Rx, Ry, Rz)

Input boundary conditions for the tower and pier bases.

Front View

Model / Boundary / ***Supports***

Select Window (Nodes: A, B, C, D in Fig. 23)

Boundary Group Name>**Default**

Options>**Add**; Support Type>**D-ALL, R-ALL** (on) ↵

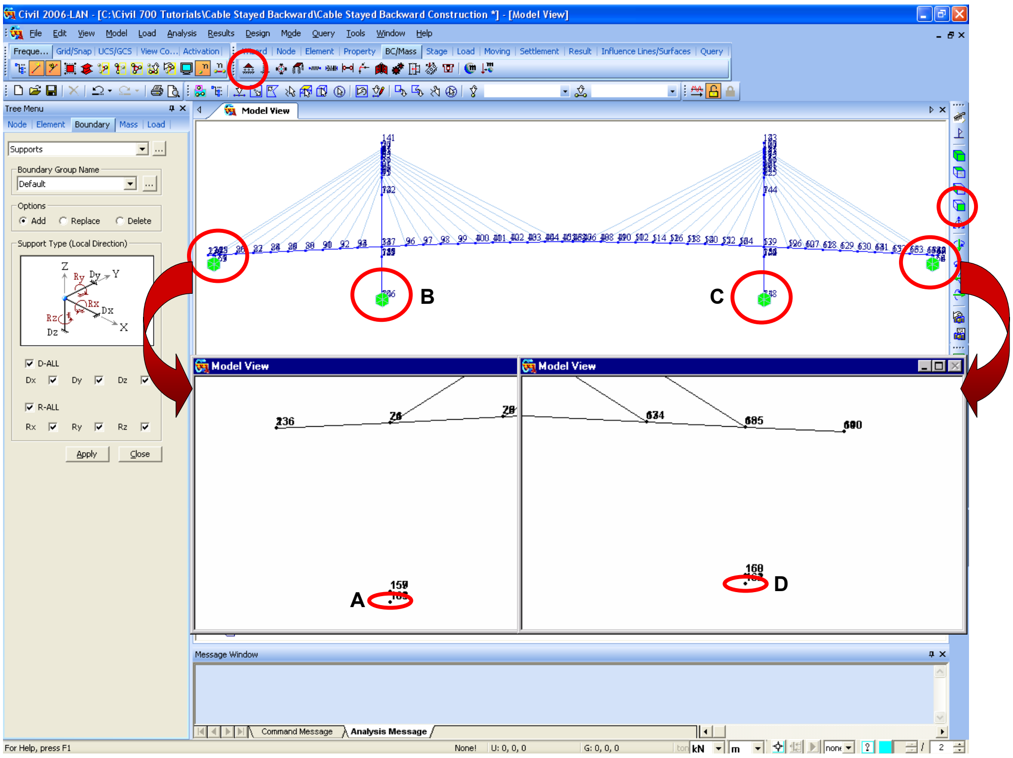


Fig. 23 Specifying Fixed Boundary Conditions for Tower and Pier Bases

Connect the centroids of the main girders to the tower bearings using *Rigid Link*.

Iso View

Model / Boundary / ***Rigid Link***
Zoom Window (A in Fig. 24)
Boundary Group Name>**Default**; Options>**Add**
Copy Rigid Link (on); Axis>**x;** Distances (m) **(220)**
Typical Type> Rigid Body **(**DOF of Rigid Link>**DX, DY, DZ, RX, RY, RZ)**
Master Node number **(155);** ***Select Single*** (Node: **137**) ↵
Master Node number **(153);** ***Select Single*** (Node: **34**) ↵

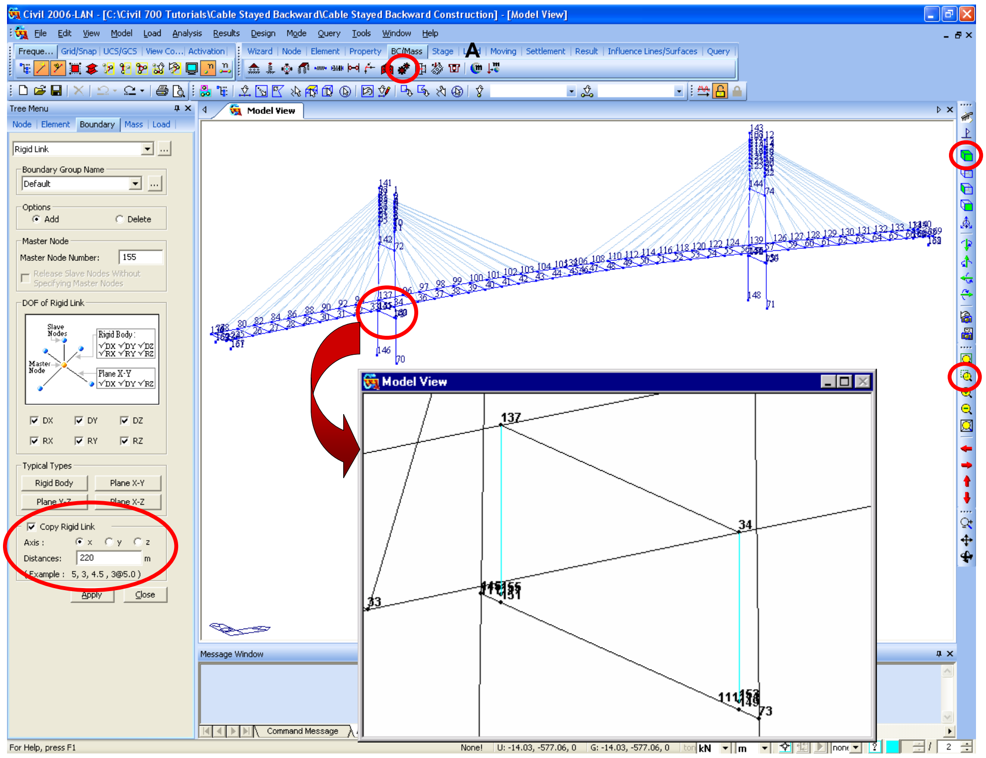


Fig. 24 Connecting Main Girders and Tower Bearings using Rigid Link

Connect the centroids of the main girders to the pier bearings using *Rigid Link*.

Model / Boundary / ***Rigid Link***
Zoom Window (A in Fig. 25)
Boundary Group Name>**Default**; Options>**Add/Replace**
Copy Rigid Link (on); Axis>**x;** Distances (m) **(414)**
Typical Type> Rigid Body **(**DOF of Rigid Link>**DX, DY, DZ, RX, RY, RZ)**
Master Node number **(159)**; ***Select Single*** (Node: **76**) ↵
Master Node number **(157)**; ***Select Single*** (Node: **24**) ↵

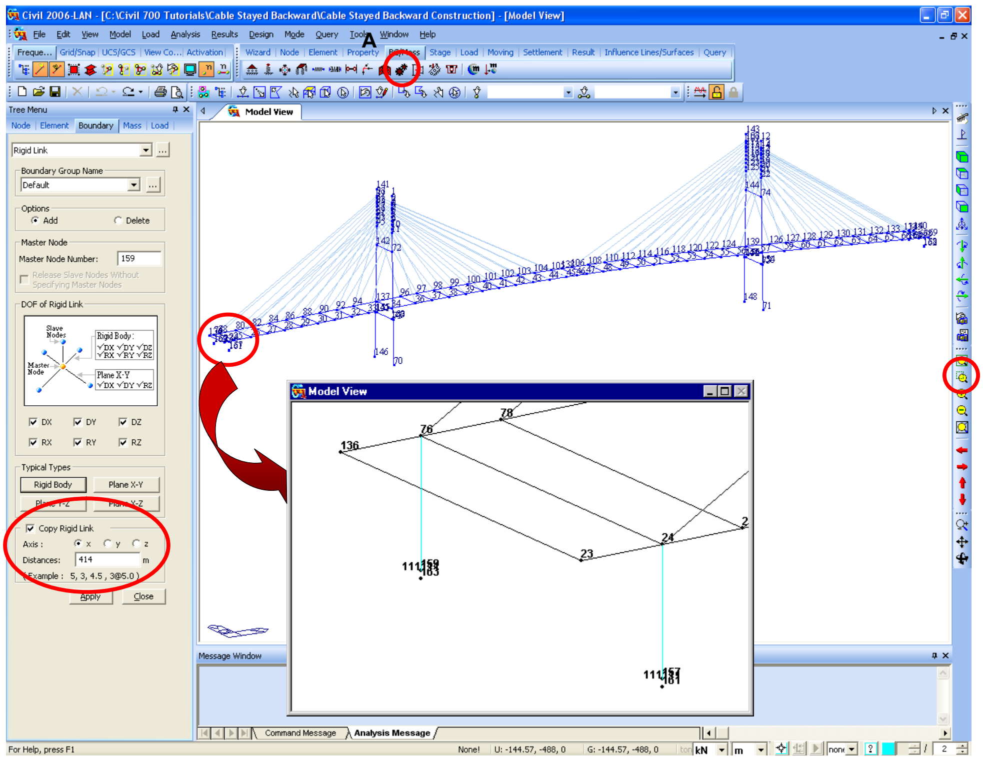


Fig. 25 Connecting Main Girders and Pier Bearings using Rigid Link

Initial Cable Prestress Calculation

The initial cable prestress, which is balanced with dead loads, is introduced to improve section forces in the main girders and towers, and cable tensions and support reactions in the bridge. It requires many iterative calculations to obtain initial cable prestress forces because a cable-stayed bridge is a highly indeterminate structure. And there are no unique solutions for calculating cable prestresses directly. Each designer may select different initial prestresses for an identical cable-stayed bridge.

The **Unknown Load Factor** function in **MIDAS/Civil** is based on an optimization technique, and it is used to calculate optimum load factors that satisfy specific boundary conditions for a structure. It can be used effectively for the calculation of initial cable prestresses.

The procedure of calculating initial prestresses for cable-stayed bridges by Unknown Load Factor is outlined in Table 3.

Step 1	Cable-Stayed Bridge Modeling
Step 2	Generate Load Conditions for Dead Loads for Main Girders and Unit Pretension Loads for Cables
Step 3	Input Dead Loads and Unit Loads
Step 4	Load Combinations for Dead Loads and Unit Loads
Step 5	Calculate unknown load factors using the Unknown Load Factor function
Step 6	Review Analysis Results and Calculate Initial Prestresses

Table 3. Flowchart for Initial Cable Prestress Calculation

Loading Condition Input

Input loading conditions for self-weight, superimposed dead load and unit loads for cables to calculate initial prestresses for the dead load condition. The number of required unknown initial cable prestress values will be set at 20, as the bridge is a symmetric cable-stayed bridge, which has 20 cables on each side of each tower. Input loading conditions for each of the 20 cables.

> It may be more convenient to use the MCT Command Shell for the input of loading conditions *STLDCASE> INSERT DATA>RUN

Load / ***Static Load Cases***

Name **(SelfWeight)**; Type>**Dead Load**

Description **(Self Weight)** ↵

Name **(Additional Load)**; Type>**Dead Load**

Description **(Additional Load)** ↵

Name **(Tension 1)**; Type>**User Defined Load**

Description **(Cable1- UNIT PRETENSION)** ↵

….

Name **(Tension 20)**; Type>**User Defined Load**

Description **(Cable20- UNIT PRETENSION)** ↵

Input the loading conditions repeatedly from Name (Tension 1) to Name (Tension 20).

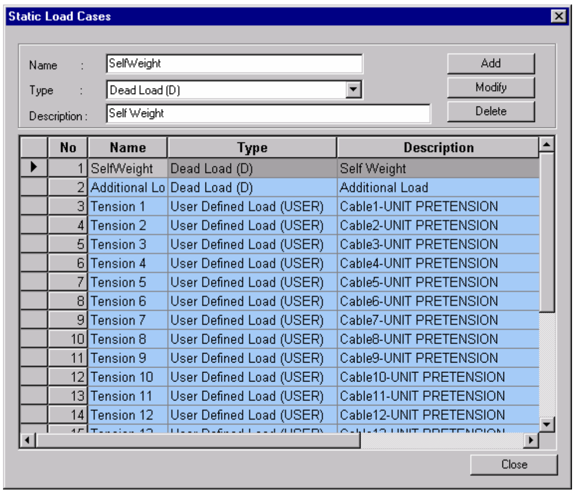


Fig. 26 Generation of Loading Conditions for Dead Loads and Unit Loads

Loading Input

Input the self-weight, superimposed dead load for the main girders and unit loads for the cables. After entering the self-weight, input the superimposed dead load that includes the effects of barriers, parapets and pavement. Input unit pretension loads for the cable elements for which initial cable prestresses will be calculated. First, input the self-weight.

Node Number (off)

Load / ***Self Weight***

Load Case Name>**SelfWeight**

Load Group Name>**Default**

Self Weight Factor>Z **(-1)** ↵

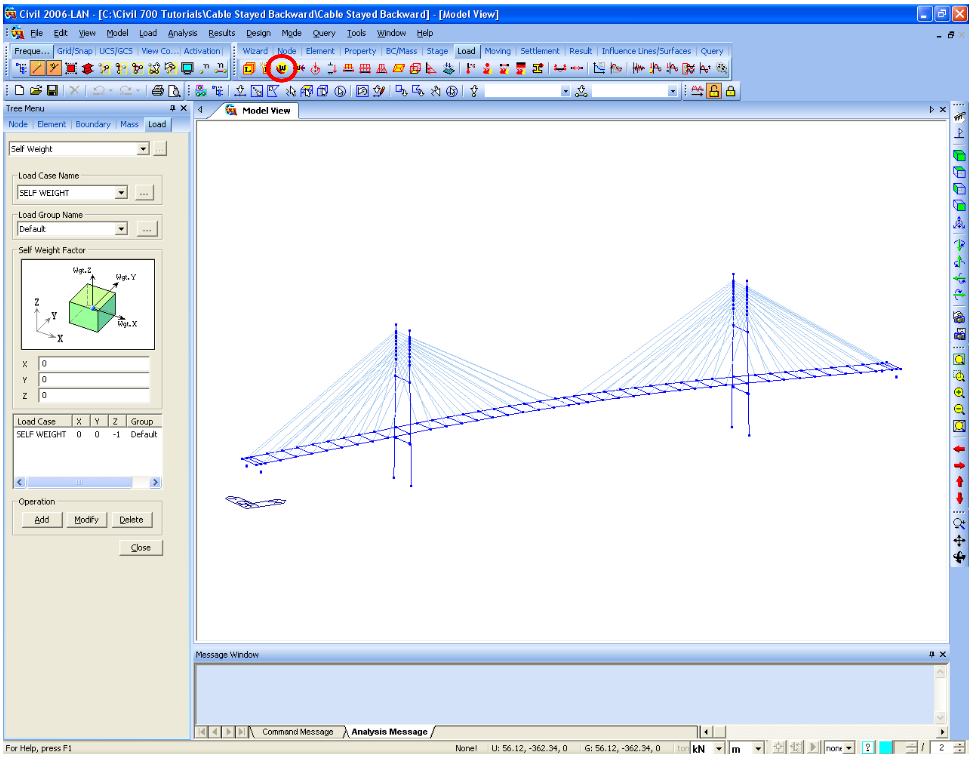


Fig. 27 Entering Self-Weight

Specify superimposed dead loads for the main girders. Divide and load the superimposed dead loads for the two main girders.

Input the superimposed dead load –18.289 kN/m, which is due to barriers, pavement, etc by the ***Element Beam Loads*** function.

Load / ***Element Beam loads***

Select identity - Elements

Select Type>Material>**Girder** ↵

Load Case Name>**Additional Load**; Options>**Add**

Load Type>**Uniform Loads**; Direction>**Global Z**

Projection>**Yes**

Value>**Relative**; x1 **(0)**, x2 **(1)**, w **(-18.289)** ↵

If the superimposed dead loads are applied to inclined elements, true loads will be applied reflecting the actual element lengths.

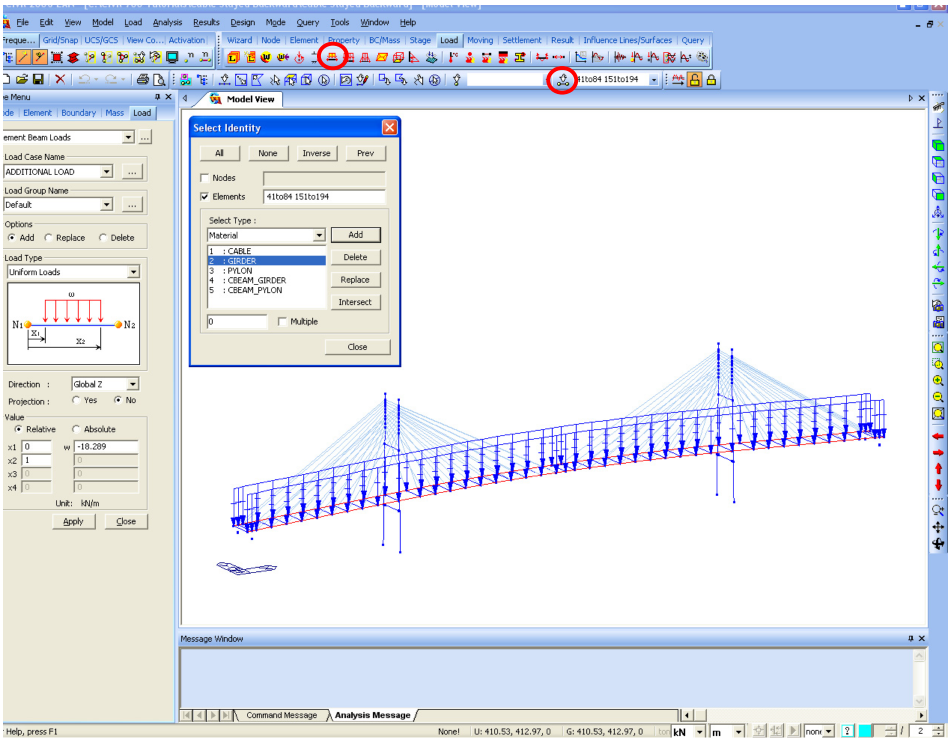


Fig. 28 Entering Superimposed Dead Loads to Main Girders

Input a unit pretension load to each cable. For the case of a symmetric cable-stayed bridge, identical initial cable prestresses will be introduced to each of the corresponding cables symmetrically to the bridge center. As such, we will input identical loading conditions to the cable pairs that form the symmetry.

Front View

Load / Prestress Loads / ***Pretension Loads***

Select Intersect (Elements: A in Fig. 29)

Select Intersect (Elements: B in Fig. 29)

Load Case Name>**Tension 1**; Load Group Name>**Default**

Options>**Add**; Pretension Load (**1**) ↵

…

Load Case Name>**Tension 20**; Load Group Name>**Default**

Options>**Add;** Pretension Load (**1**) ↵

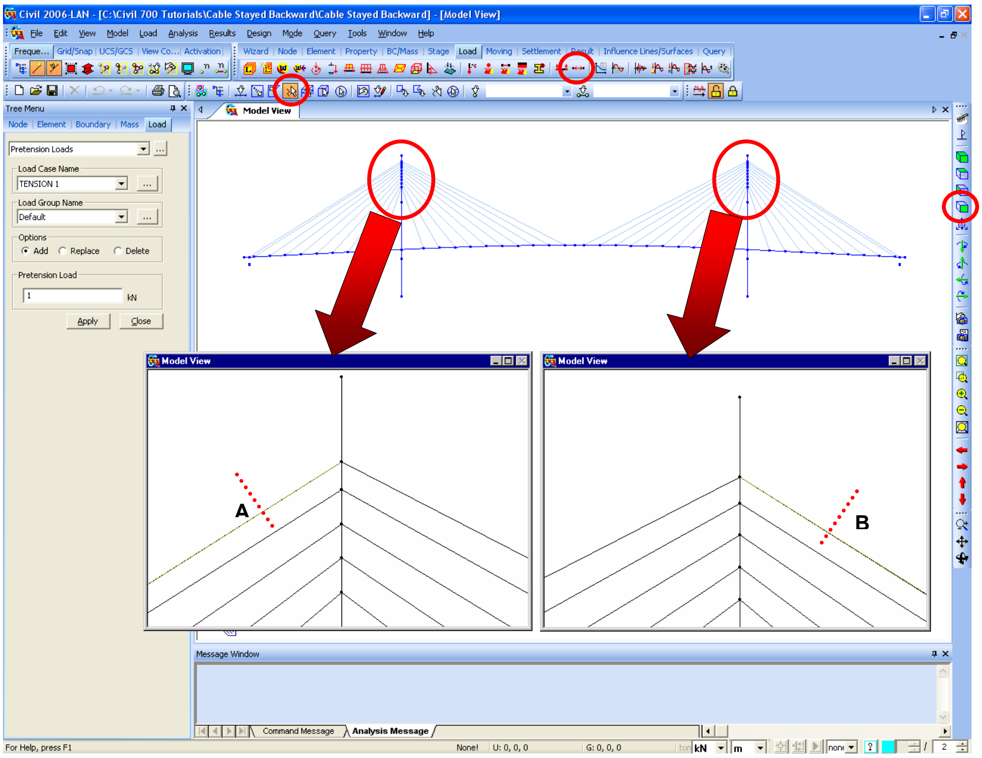


Fig. 29 Entering Unit Pretension Load to Cables

Input the unit pretension loads for all the cables repeatedly from Tension 2 to Tension 20 according to Table 4.

Table 4. Loading Conditions and Element Numbers

Load Case	Element No.	Load Case	Element No.
Tension 1	1, 40, 111, 150	Tension 11	20, 21, 130, 131
Tension 2	2, 39, 112, 149	Tension 12	19, 22, 129, 132
Tension 3	3, 38, 113, 148	Tension 13	18, 23, 128, 133
Tension 4	4, 37, 114, 147	Tension 14	17, 24, 127, 134
Tension 5	5, 36, 115, 146	Tension 15	16, 25, 126, 135
Tension 6	6, 35, 116, 145	Tension 16	15, 26, 125, 136
Tension 7	7, 34, 117, 144	Tension 17	14, 27, 124, 137
Tension 8	8, 33, 118, 143	Tension 18	13, 28, 123, 138
Tension 9	9, 32, 119, 142	Tension 19	12, 29, 122, 139
Tension 10	10, 31, 120, 141	Tension 20	11, 30, 121, 140

Check the unit pretension loads entered for the cables using ***Display***.

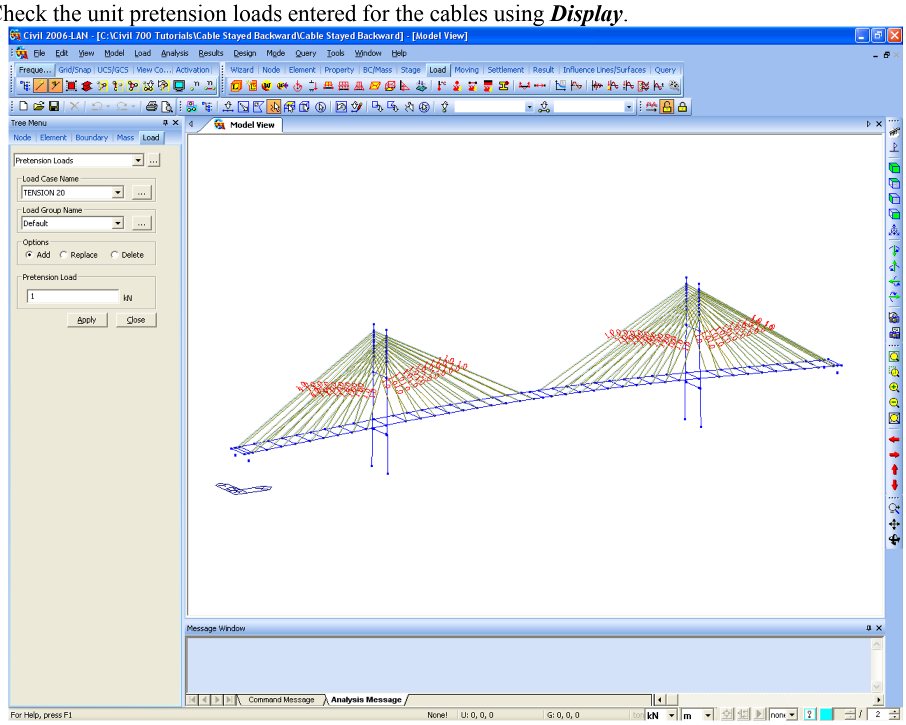


Fig. 30 Unit Pretension Loads entered for Cables

Perform Structural Analysis

Perform static analysis for self-weight, superimposed dead loads and unit pretension loads for the cables.

Analysis / ***Perform Analysis*** ↵

Final Stage Analysis Results Review

Load Combination Generation

Create load combinations using the 20 loading conditions for cable unit pretension loading, self-weights and superimposed dead loads.

Results / ***Combinations***

General Tab

Load Combination List>Name>**(LCB 1)**; Active>**Active**; Type>**Add**

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

LoadCase>**Additional Load (ST)**; Factor **(1.0)**

LoadCase>**Tension 1(ST)**; Factor **(1.0)**

…

LoadCase>**Tension 20(ST)**; Factor **(1.0)** ↵

Repeat input for cable loading conditions from Tension 1(ST) to Tension 20 (ST).

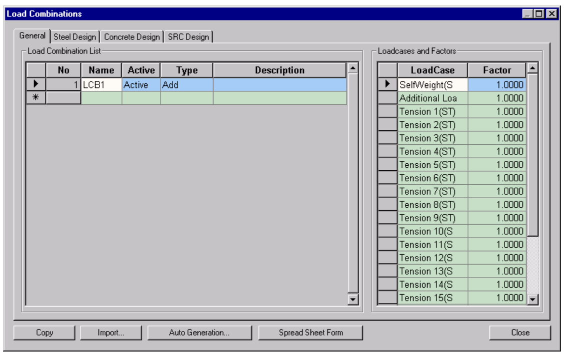


Fig. 31 Creating Load Combinations

Unknown Load Factors Calculation

Calculate unknown load factors that satisfy the boundary conditions by the *Unknown Load Factor* function for LCB1, which was generated through load combination. The constraints are specified to limit the vertical deflection (Dz) of the 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 1**

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

Select All

LCase>**SelfWeight** (off)

LCase>**Additional Load** (off)

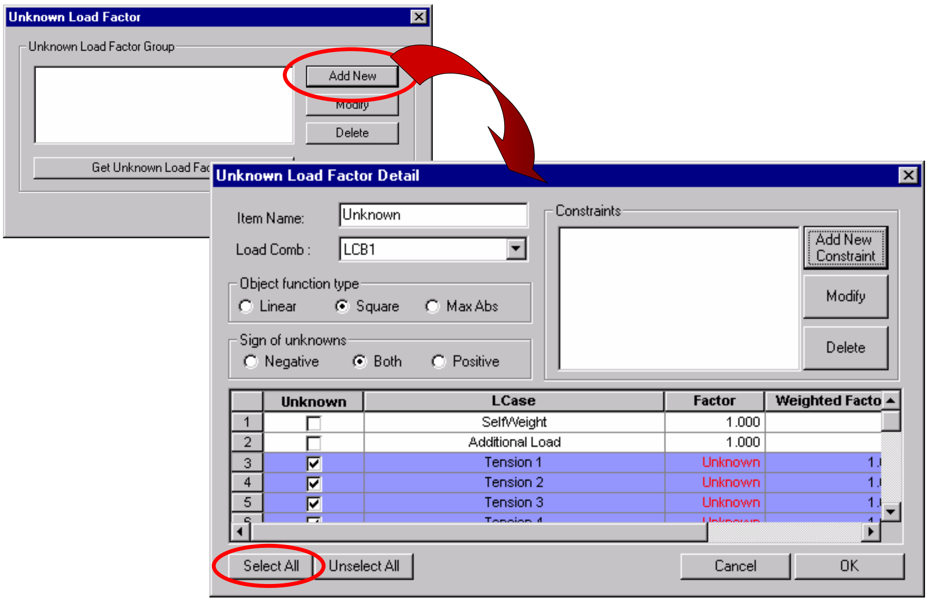


Fig. 32 Unknown Load Factor Dialog Box

Specify the constraining conditions, which restrict the vertical displacement (Dz) of the main girders by the ***Constraints*** function.

Constraints>Add New Constraint

Constraint Name **(Node 23)**

Constraint Type>**Displacement**

Node ID **(23)**

Component>**Dz**

Equality/Inequality Condition>**Inequality**; Upper Bound **(0.01)**; Lower Bound **(-0.01)**

↵ ↵

Constraints>Add New Constraint

Constraints Name **(Node 24)**

Constraints Type>**Displacement**

Node ID **(24)**

Component>**Dz**

Equality/Inequality Condition>**Inequality**; Upper Bound **(0.01)**; Lower Bound **(-0.01)**

↵ ↵

In this tutorial, we will apply constraints to restrict the vertical displacement of the main girders. Because the analytical model is symmetric, we define only half of the main girders with constraints. Use Node 23 to Node 45 on the left half of the bridge as constraints.

Repeatedly input the remaining constraints from Node 25 to Node 45 of the main girder. Node 35 is excluded because it was deleted by ***Merge Nodes***.

The constraints for calculating Unknown Load Factors can be easily entered by MCT Command Shell *UNKCONS > INSERT DATA >RUN

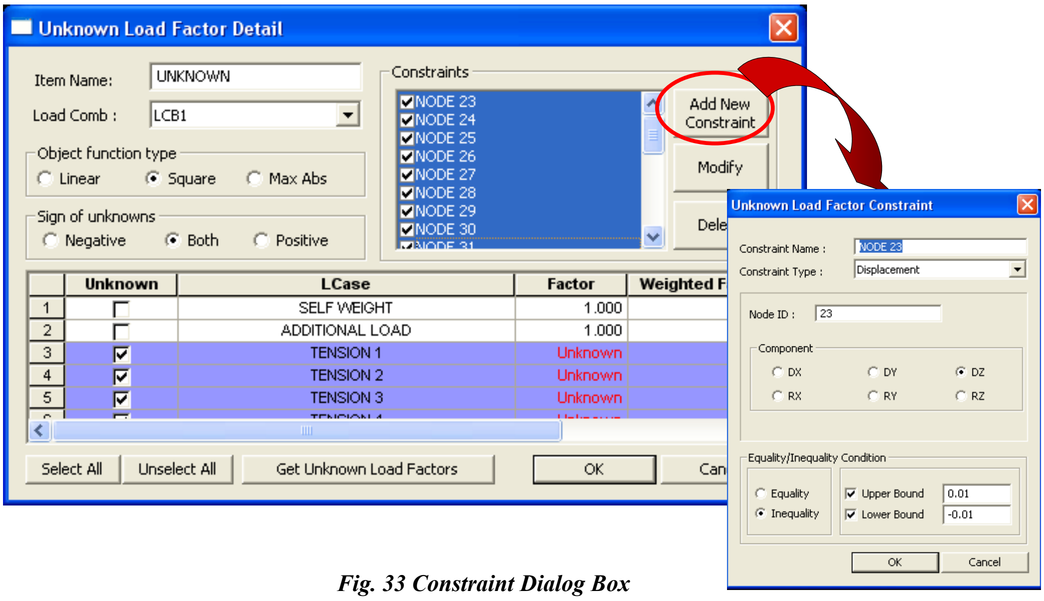


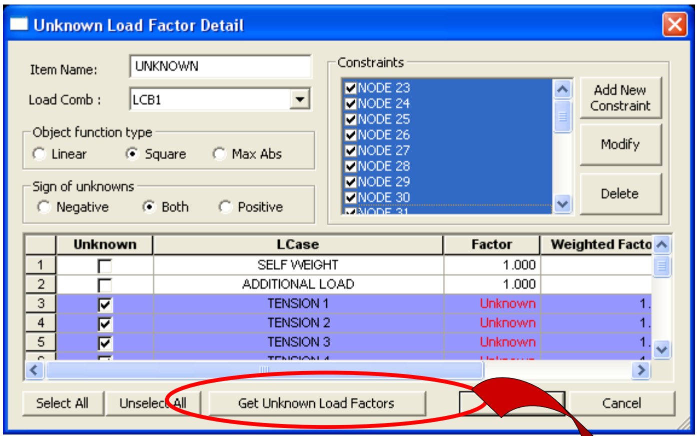
Fig. 33 Constraint Dialog Box

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

The explanations for the calculation of unknown load factors can be found in "Solution for Unknown Loads using Optimization Technique" in Analysis for Civil Structures.

Unknown Load Factor Group> Get Unknown Load Factors !

Fig. 34 shows the analysis results for unknown load factors calculated by ***Unknown Load Factor***.



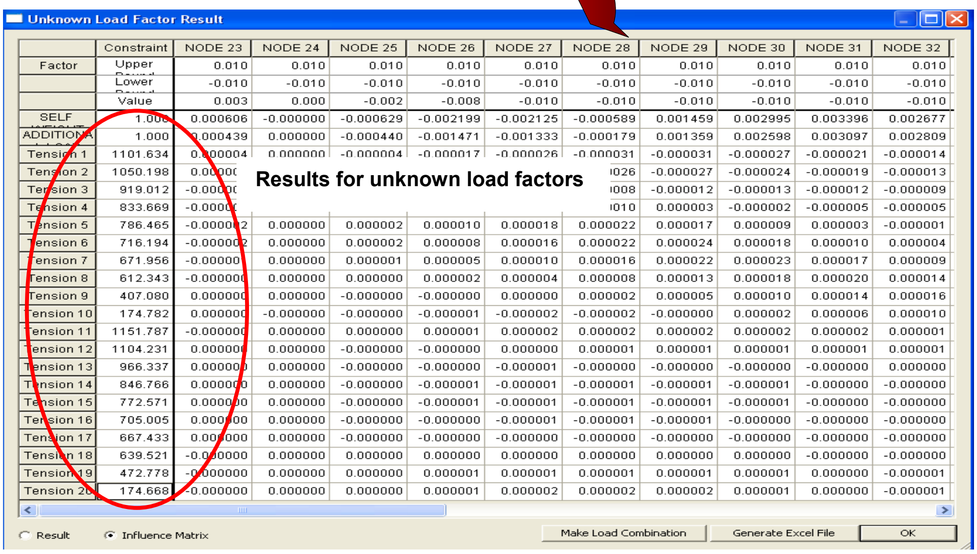


Fig. 34 Analysis Results for Unknown Load Factors

We now check to see if the calculation results satisfy the constraints by generating a new loading combination using the unknown load factors.

Influence Matrix (on)

Make Load Combination>Name>**(LCB 2)** ↵ ↵

Results>Combination ↵

Load Combinations are shown in Fig. 35.

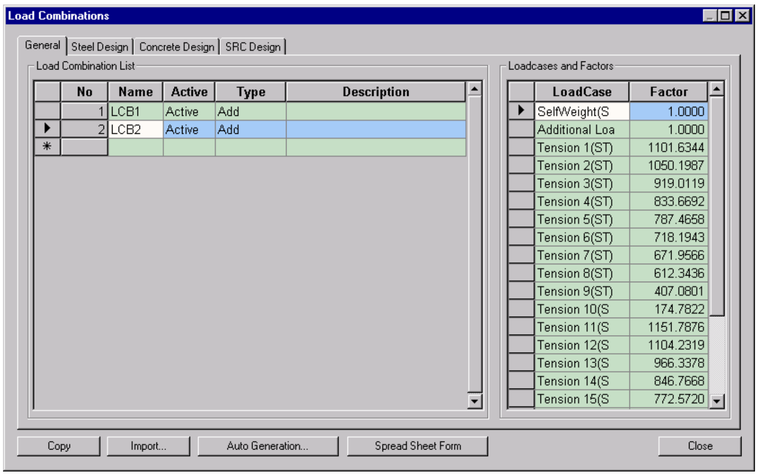


Fig. 35 New Load Combination using Unknown Load Factors

Deformed Shape Review

We now confirm deflections at the final stage to which initial cable prestresses, self-weights and superimposed dead loads are applied.

Tools / ***Unit System***

Length>**mm** ↵

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

Load Cases/Combinations>**CB:LCB 2**

Components>**DXYZ**

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

Deform ...

Deformation Scale Factor (**0.3**) ↵

Zoom Window (A, B in Fig. 36)

If the default Deformation Scale Factor is too large, we can adjust the factor.

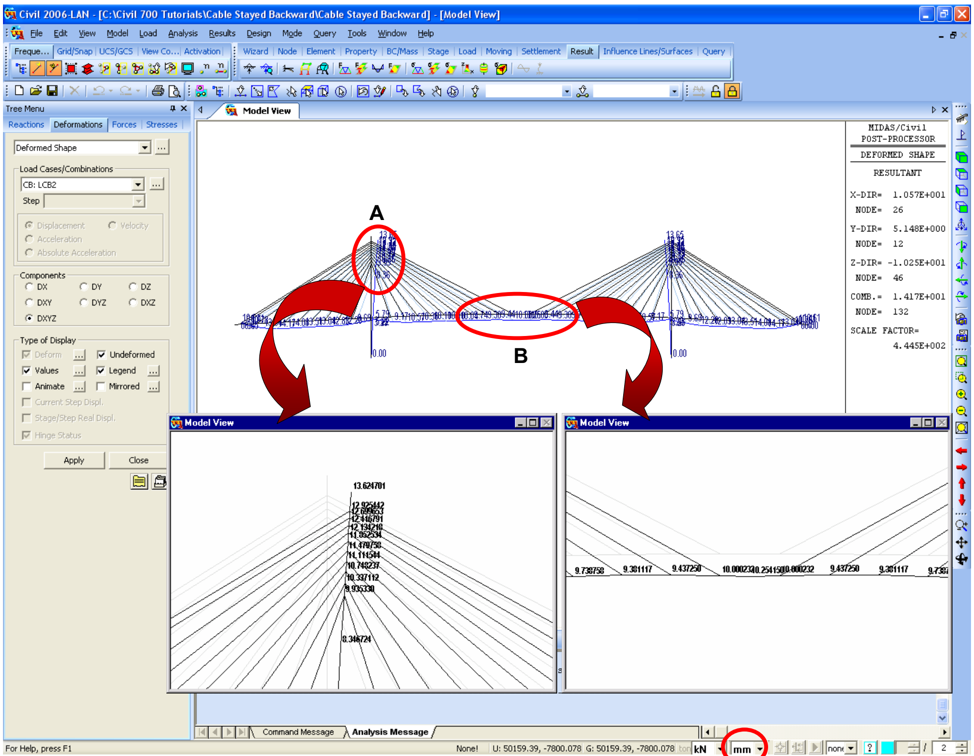


Fig. 36 Checking Deformed Shape

Construction Stage Analysis

To design a cable-stayed bridge, its construction stages should be defined to check the stability during construction. The structural system could change significantly based on the erection method. And the change of system during construction can result in more critical condition for the structure compared to the state of the final stage. As such, an accurate construction stage analysis should be performed for designing a cable-stayed bridge to check the stability and to review stresses for the structure.

The cable prestresses, which are introduced during the construction of a cable-stayed bridge, could be calculated by backward analysis from the final stage. To perform a construction stage analysis, construction stages should be defined to consider the effects of the activation and deactivation of main girders, cables, cable anchorage, boundary conditions, loads, etc. Each stage must be defined to represent a meaningful structural system, which changes during construction.

Construction Stage Category

In construction stage analysis, we need to consider constantly changing structures, boundary conditions and loading conditions, which are different in every stage. Using the final stage model, we can then generate the structural systems for each construction stage. In this tutorial, we will consider the stages from the construction stage, which represents completion of the towers and the main girders of the side spans, to the construction stage, which applies loading for superimposed dead loads.

The construction basics for the cable-stayed bridge in this tutorial are as follows:

- Towers
  - Large Block construction method
- Main Girders
  - Side Spans : Temporary Bents + Large Block method
  - Center Span: Small Block method by Traveler Crane
- Cables
  - Direct Lifting by Truck Crane

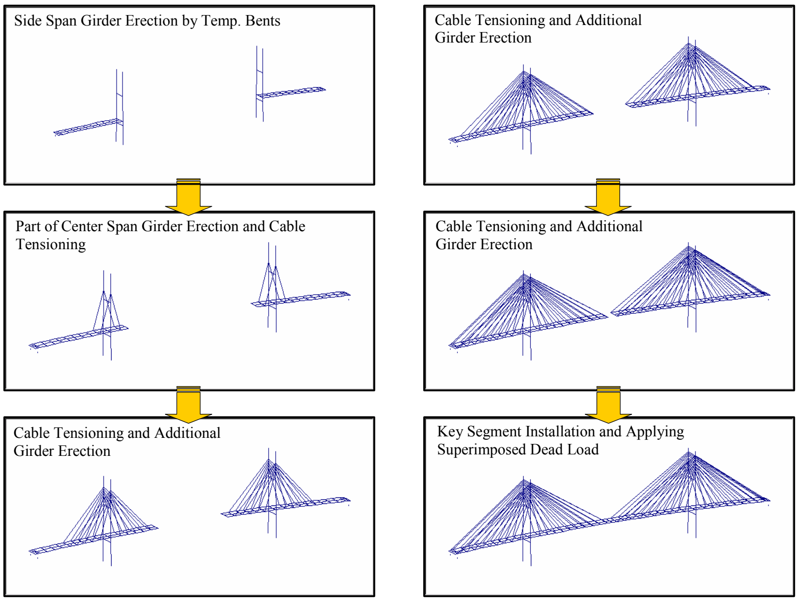


Fig. 37 Construction Sequence for Analytical Model

Cannibalization Stage Category

In this tutorial, 33 cannibalization stages are generated to simulate the changes of loading and boundary conditions.

The cannibalization stages applied in this tutorial are outlined in Table 5.

Table 5 Cannibalization Stage Category

Stage	Content	Stage	Content
CS 0	Final Stage (Dead Load+Superimposed Dead Load+Initial Prestress)	CS 17	Main Girder (6) removal
CS 1	Superimposed Dead Load removal	CS 18	Cable (15, 26) removal
CS 2	Apply Temporary Bents & Key Segment removal (Main Girder No. 11)	CS 19	Cable (6, 35) removal
CS 3	Cable (20, 21) removal	CS 20	Main Girder (5) removal
CS 4	Cable (1,40) removal	CS 21	Cable (14, 27) removal
CS 5	Main Girder (10) removal	CS 22	Cable (7, 34) removal
CS 6	Cable (19, 22) removal	CS 23	Main Girder (4) removal
CS 7	Cable (2, 39) removal	CS 24	Cable (13, 28) removal
CS 8	Main Girder (9) removal	CS 25	Cable (8, 33) removal
CS 9	Cable (18, 23) removal	CS 26	Main Girder (3) removal
CS 10	Cable (3, 38) removal	CS 27	Cable (12, 29) removal
CS 11	Main Girder (8) removal	CS 28	Cable (9, 32) removal
CS 12	Cable (17, 24) removal	CS 29	Main Girder (2) removal
CS 13	Cable (4, 37) removal	CS 30	Cable (11, 30) removal
CS 14	Main Girder (7) removal	CS 31	Cable (10, 31) removal
CS 15	Cable (16, 25) removal	CS 32	Main Girder (1) removal
CS 16	Cable (5, 36) removal		

* Cable (1) is outer cable and Cable (10) is inner cable in the left span.
* Cable (11, 30) are inner cables and Cable (20, 21) are outer cables in the center span.
* Cable (31) is inner cable and Cable (40) is outer cable in the right span.
* Elements representing the main girders in the center span are divided according to the cable spacing, and the main girder (11) is a closure key segment.

Backward Construction Stage Analysis

Construction stage analysis for a cable-stayed bridge can be classified into forward analysis and backward analysis, based on the analysis sequence. Forward analysis reflects the real construction sequence. Whereas backward analysis is performed from the state of the finally completed structure for which an initial equilibrium state is determined, and the elements and loads are eliminated in reverse sequence to the real construction sequence.

In this tutorial, we will examine the structural behavior of the analytical model and the changes of cable tensions, displacements and moments.

The analytical sequence of backward construction stage analysis is as shown in Fig. 38.

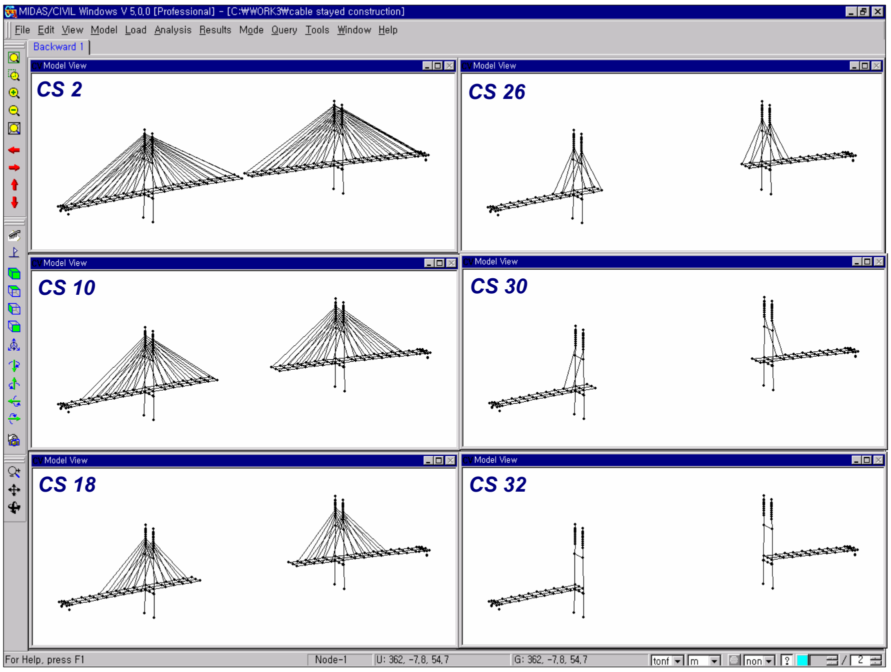


Fig. 38 Analysis Sequence by Backward Construction Stage Analysis

We will generate a construction stage analytical model using the model used in the final stage analysis by saving the file under a different name.

File / ***Save As*** **(Cable Stayed Backward Construction)**

The following steps are carried out to generate the construction stage analysis model:

1. **Input initial cable tension forces**
   Change the truss element used in the final stage analysis to cable element.
   Input the unknown load factors calculated by the Unknown Load Factor function as the initial cable prestress.

2. **Define Construction Stage names**
   Define each construction stage and the name.

3. **Define Structural Group**
   Define the elements by group, which are added/deleted in each stage.

4. **Define Boundary Group**
   Define the boundary conditions by group, which are added/deleted in each stage.

5. **Define Load Group**
   Define the loading conditions by group, which are added/deleted in each stage.

6. **Define Construction Stages**
   Define the elements, boundary conditions and loadings pertaining to each stage.

Input Initial Cable Prestress

In order to create the construction stage analysis model from the final stage model, delete the load combinations LCB 1 & 2 and unit pretension loading conditions, Tension 1 to Tension 20. To input the unknown load factors calculated by optimization technique as Pretension Loads, define a new loading case for initial prestress.

Results / ***Combinations***

Load Combination List>Name>**LCB 1, LCB 2** Delete

Load / ***Static Load Cases***

Name **(Tension 1)** ~ Name **(Tension 20)** Delete

Name **(Pretension)**; Type > **User Defined Load** ↵

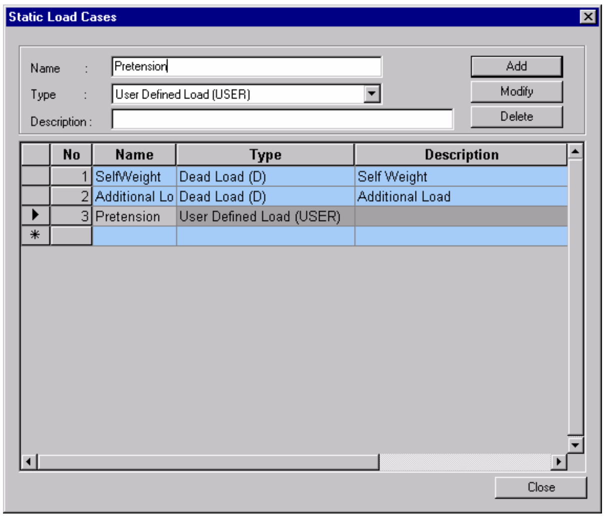


Fig. 39 Entering Initial Prestress Loading Condition

In construction stage analysis for cable-stayed bridges, geometrical nonlinear analysis for cable element should be performed. To consider the sag effect of cable element in cable-stayed bridges, the truss elements used in the final stage analysis should be transformed to cable elements. In a cable-stayed bridge, an equivalent truss element is used for the cable element. This element considers the stiffness due to tensioning.

Tools / ***Unit System***

Length>**m** ↵

Model / Elements / ***Change Elements Parameters***

Select identity - Elements

Select Type>Element Type>**Truss** ↵

Parameter Type > **Element Type** (on)

Mode> From> **Truss** (on); To > **Tension only/Hook/Cable**

Cable (on) ; **Pretension=0** ↵

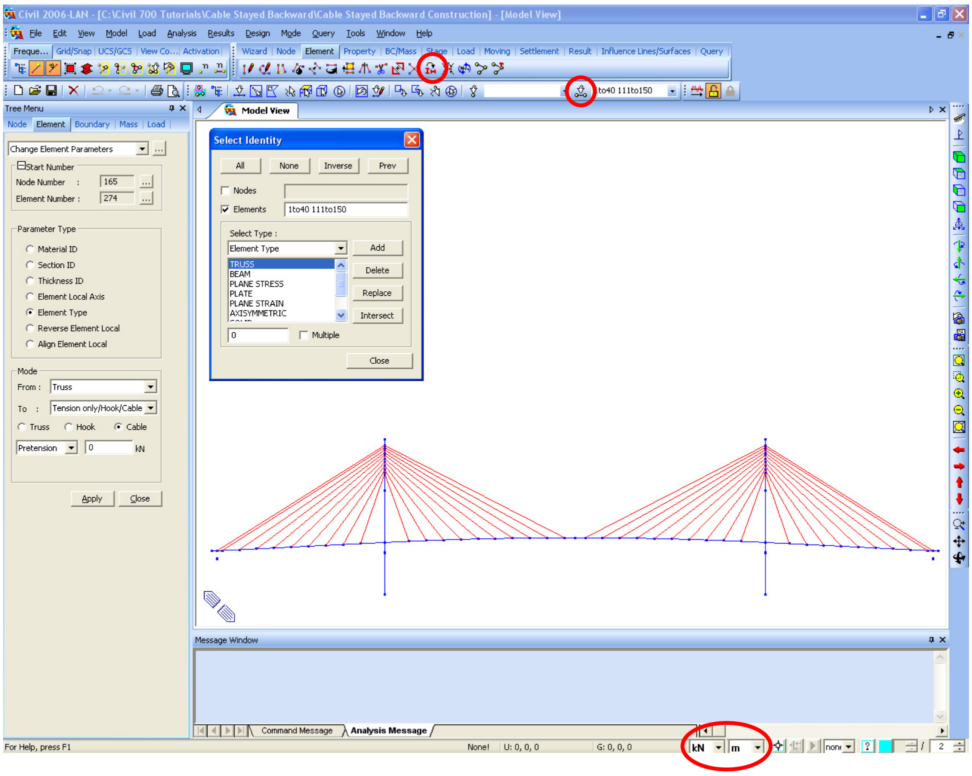


Fig. 40 Change of Truss Element to Cable Element

Input the unknown load factors calculated by optimization technique to individual cable elements as Pretension Loads.

The input method for Pretension Loads is the same as for inputting unit pretension loads for cable elements.

Load / Prestress Loads / ***Pretension Loads***
Zoom Window (A in Fig. 41)
Select Intersect (Elements: A in Fig. 41)
Zoom Window (B in Fig. 41)
Select Intersect (Elements: B in Fig. 41)
Load Case Name > **Pretension**; Load Group Name > **Default** Options > **Add**; Pretension Load **(1101.63)** ↵

Input the pretension loads in Table 6 to each cable element repeatedly.

Table 6. Initial Prestress (Pretension Loading) calculated by Optimization Technique

Element No.	Pretension Loading	Element No.	Pretension Loading
1, 40, 111, 150	1101.63	20, 21, 130, 131	1151.79
2, 39, 112, 149	1050.20	19, 22, 129, 132	1104.23
3, 38, 113, 148	919.01	18, 23, 128, 133	966.34
4, 37, 114, 147	833.67	17, 24, 127, 134	846.77
5, 36, 115, 146	787.47	16, 25, 126, 135	772.57
6, 35, 116, 145	718.19	15, 26, 125, 136	705.01
7, 34, 117, 144	671.96	14, 27, 124, 137	667.43
8, 33, 118, 143	612.34	13, 28, 123, 138	639.52
9, 32, 119, 142	407.08	12, 29, 122, 139	472.78
10, 31, 120, 141	174.78	11, 30, 121, 140	174.67

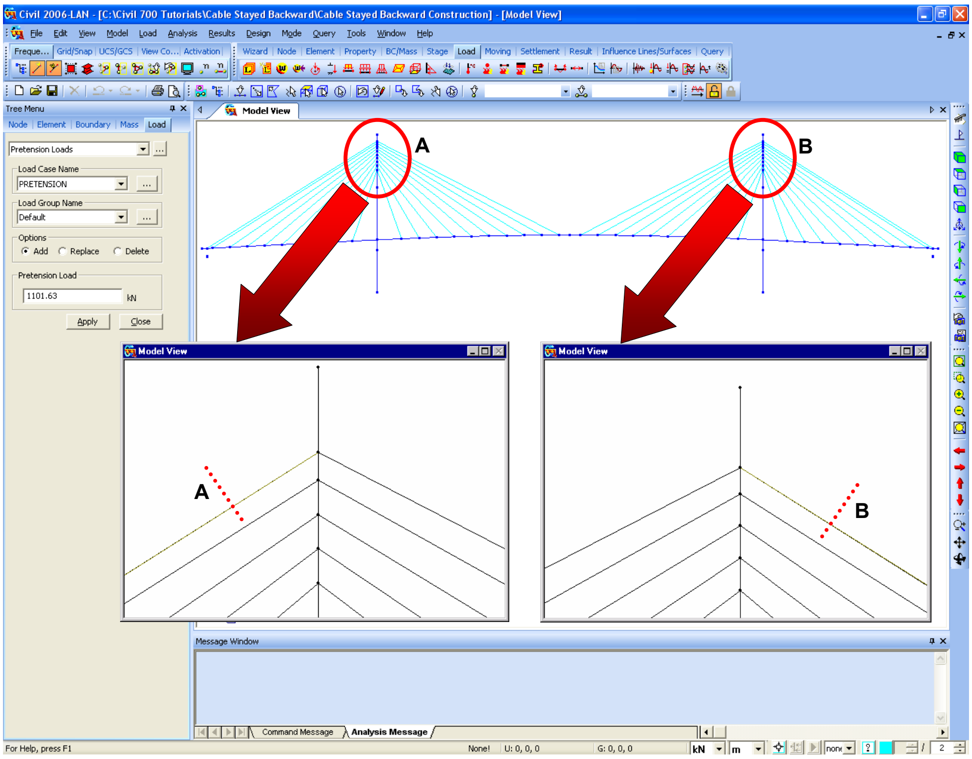


Fig. 41 Input Pretension Loading to Cable Elements

Define Construction Stage

We now define each construction stage to perform backward construction stage analysis. First, we assign each construction stage name in the Construction Stage dialog box. In this tutorial, we will define total 33 construction stages including the final stage.

- Define multiple construction stages simultaneously by assigning numbers to a stage. The generated construction stages will, thus, have having identical names.

- For generating analysis results, the analysis results in each construction stage are saved and subsequently generated.

Load / Construction Stage Analysis Data / ***Construction Stage***

Define Construction Stage Generate

Stage>Name **(CS)**; Suffix **(0to32)**

Save Result>**Stage** (on) ↵

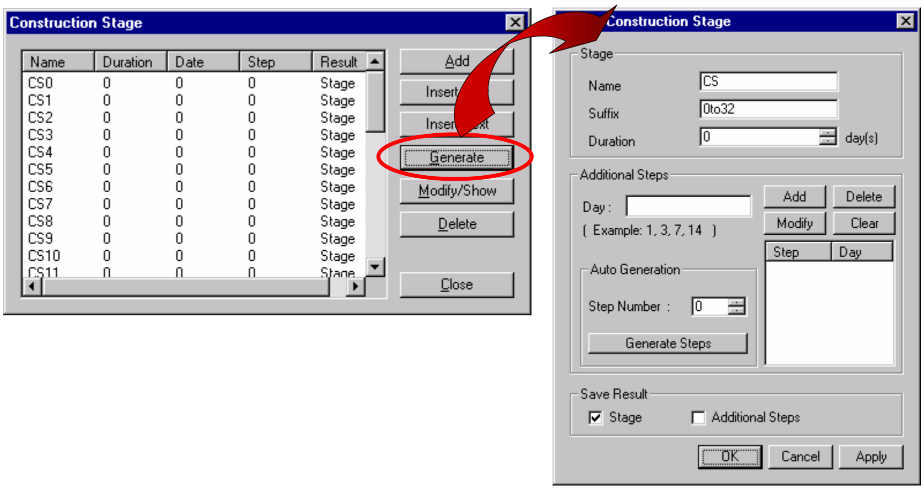


Fig. 42 Construction Stage Dialog Box

Assign Structure Group

Assign the elements, which are added/deleted in each construction stage by Structure Group. After defining the name of each Structure Group, we then assign relevant elements to the Structure Group.

Group Tab

C Group>Structure Group>**New...** (right-click mouse)

Name **(SG)**; Suffix **(0to32)** Add

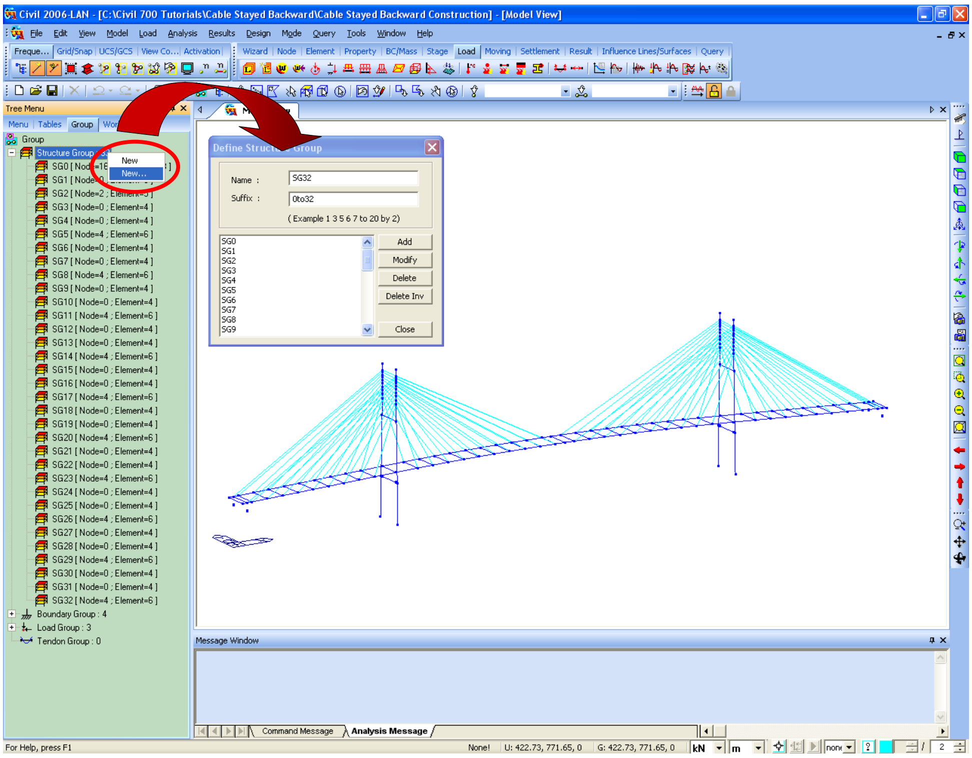


Fig. 43 Defining Structure Group

Assign the elements, which become added/deleted in each construction stage, to each corresponding Structure Group. The final stage is defined as the SG0 Structure Group. We skip the construction stage CS1 because CS1 is a construction stage, which eliminates the superimposed dead load, and as such there are no added/deleted elements involved.

Front View

Group > Structure Group

Select All

SG0 (*Drag & Drop*)

Select Window (Elements: **62, 63, 172, 173, 263** A in Fig. 45)

SG2 (*Drag & Drop*)

C ***Inactivate***

Inactivate previously defined element groups so that they do not overlap with another element group.

Define the Structure Group SG3 to SG32 by eliminating main girders and cables sequentially while referring to Table 5 Cannibalization Stage Category.

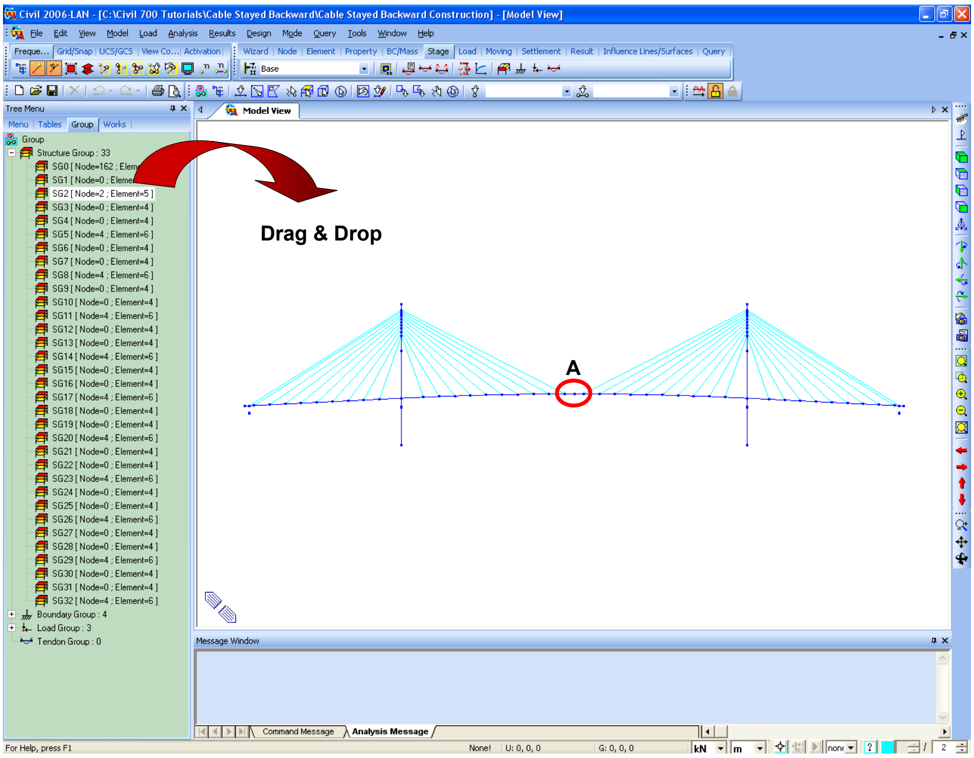


Fig. 44 Defining Structure Group SG2

Assign the Structure Group, which is required to define the last stage (CS32) in backward construction stage analysis.

Construction stage CS32 is the stage in which all the cable elements and main girders in the center span are eliminated, and the temporary bents in the side spans are erected. Actually, this is the 1st stage in the cable-stayed bridge construction.

Select Window (A in Fig. 45)

SG32 (***Drag & Drop***)

C ***Inactivate***

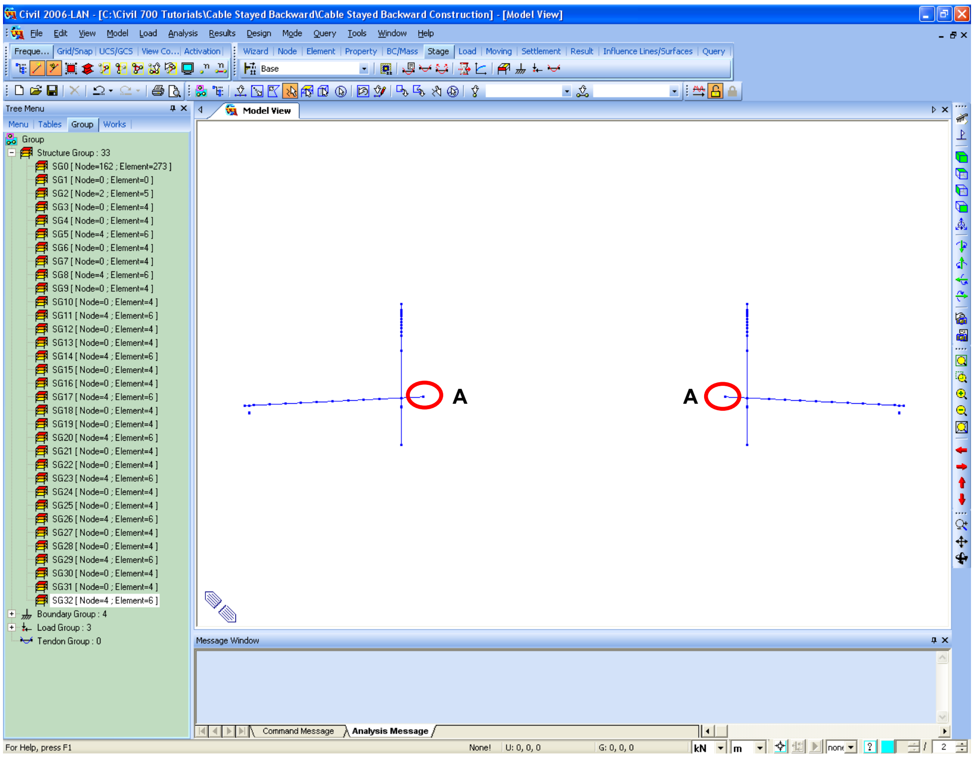


Fig. 45 Defining Structure Group SG32

Assign Boundary Group

Assign the boundary conditions, which become added/deleted in each construction stage, to each corresponding Boundary Group. After defining the name of each Boundary Group, we then assign relevant boundary conditions to each Boundary Group.

 ***Activate All***

Group Tab

C Group>Boundary Group>**New…** (right-click mouse)

Name **(Fixed Support)** ↵

Name **(Elastic Link)** ↵

Name **(Bent)** ↵

Name **(Rigid Link)** ↵

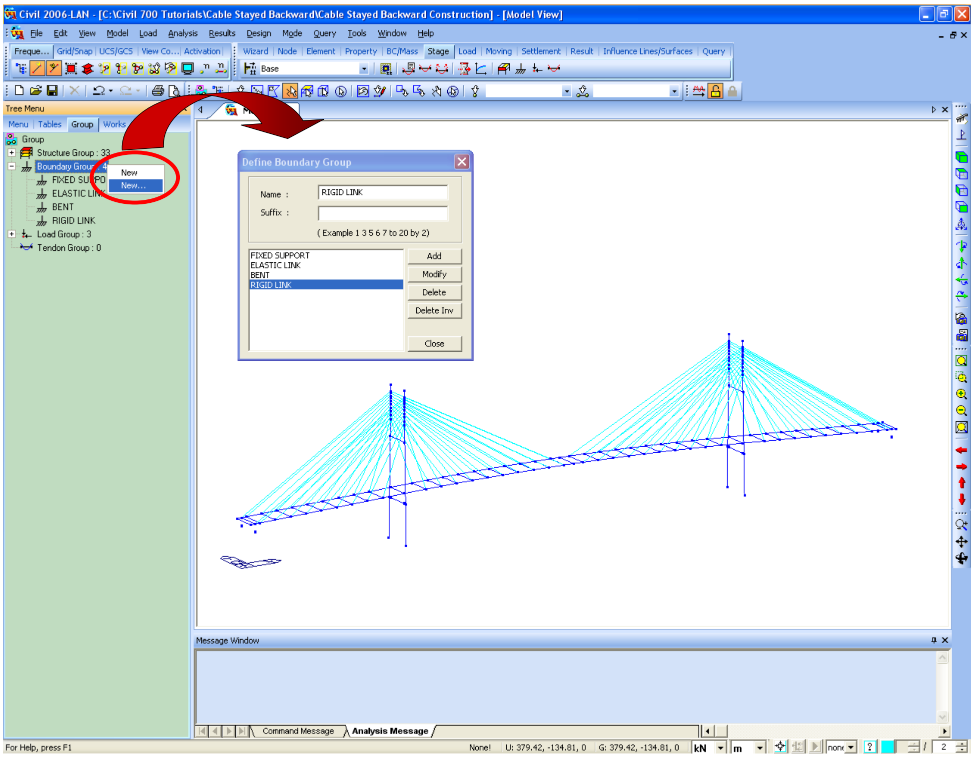


Fig. 46 Defining Boundary Group

Reassign the fixed support, Elastic Link and Rigid Link conditions, which were already defined for the final stage analysis, to Boundary Group for the construction stage analysis.

Group>Boundary Group

Select All

Fixed Support (*Drag & Drop*)

Select Boundary Type>**Support** (on) ↵

Select All

Elastic Link (*Drag & Drop*)

Select Boundary Type>**Elastic Link** (on) ↵

Select All

Rigid Link (*Drag & Drop*)

Select Boundary Type>**Rigid Link** (on) ↵

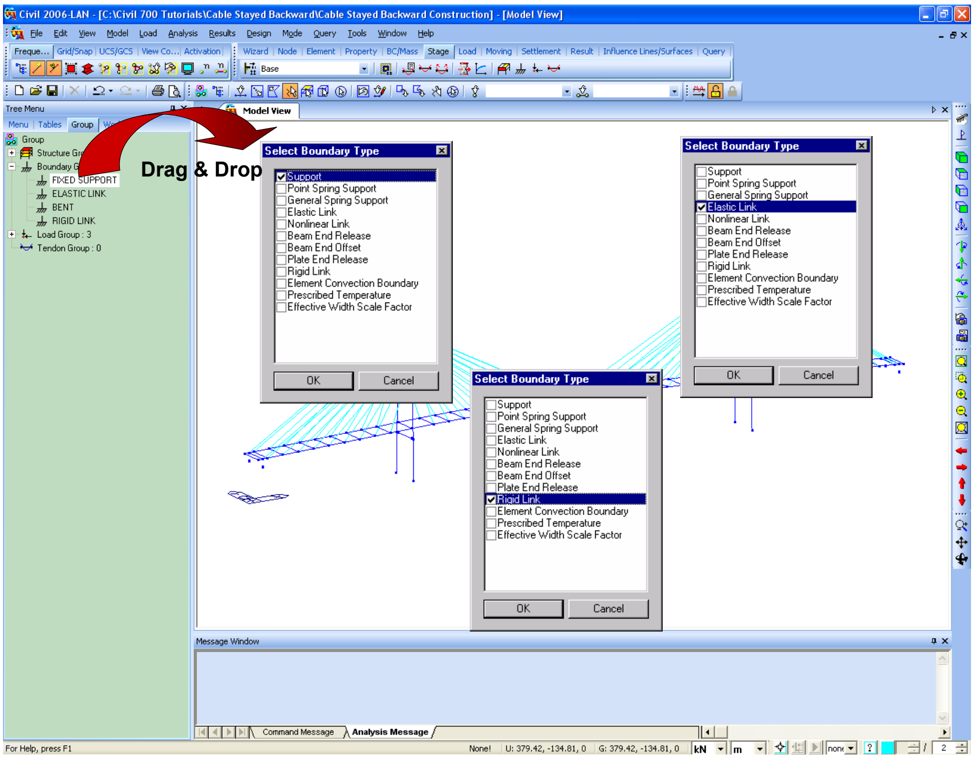


Fig. 47 Generating Fixed Support, Elastic Link and Rigid Link Conditions

We also assign the boundary condition for the temporary bents to a Boundary Group. We will input the boundary condition as hinge condition (Dx, Dy, Dz, Rz) at the centers of the side spans.

Iso View

Model / Boundary / ***Supports***

Select Identity- Node (Nodes: **86, 29, 130, 63**) ↵

Boundary Group Name>**Bent**

Options>**Add**

Support Type>**D-ALL** (on); **Rz** (on) ↵

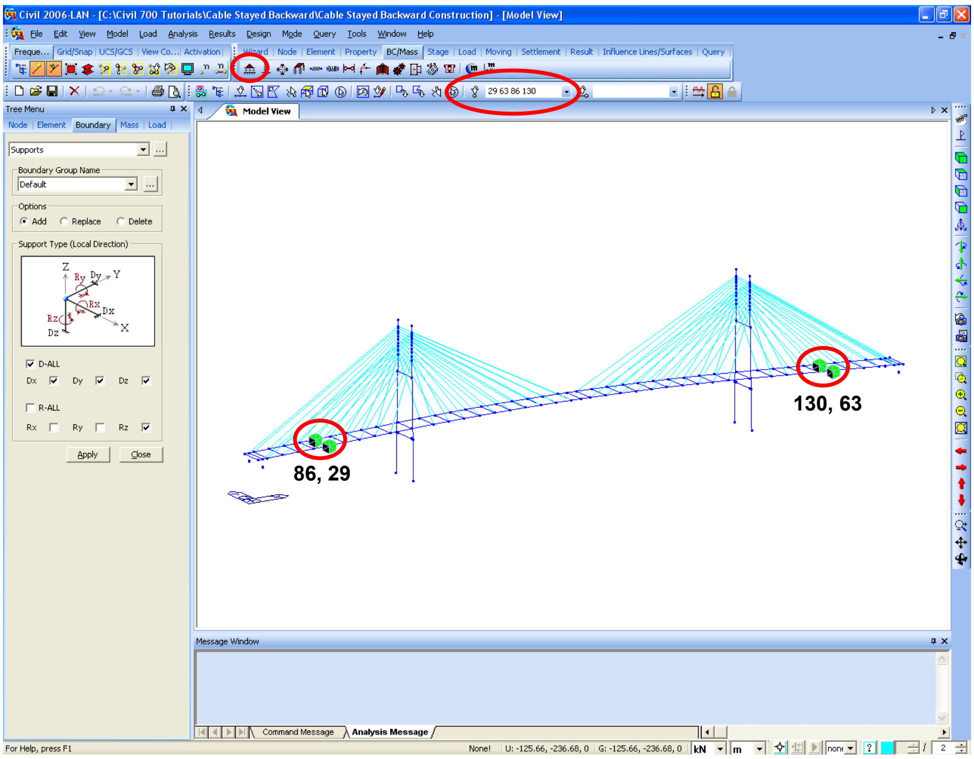


Fig. 48 Generating Boundary Condition for Temporary Bents

Assign Load Group

Assign the loading conditions, which become added/deleted in each construction stage, to each corresponding Load Group. The loads considered in this backward construction stage analysis are self-weight, superimposed dead load and initial cable prestress. First, we generate the name of each Load Group and then assign corresponding loading conditions to each Load Group.

Group Tab

C Group>Load Group> **New…** (right-click mouse)

Name **(SelfWeight)** ↵

Name **(Additional Load)** ↵

Name **(Pretension Load)** ↵

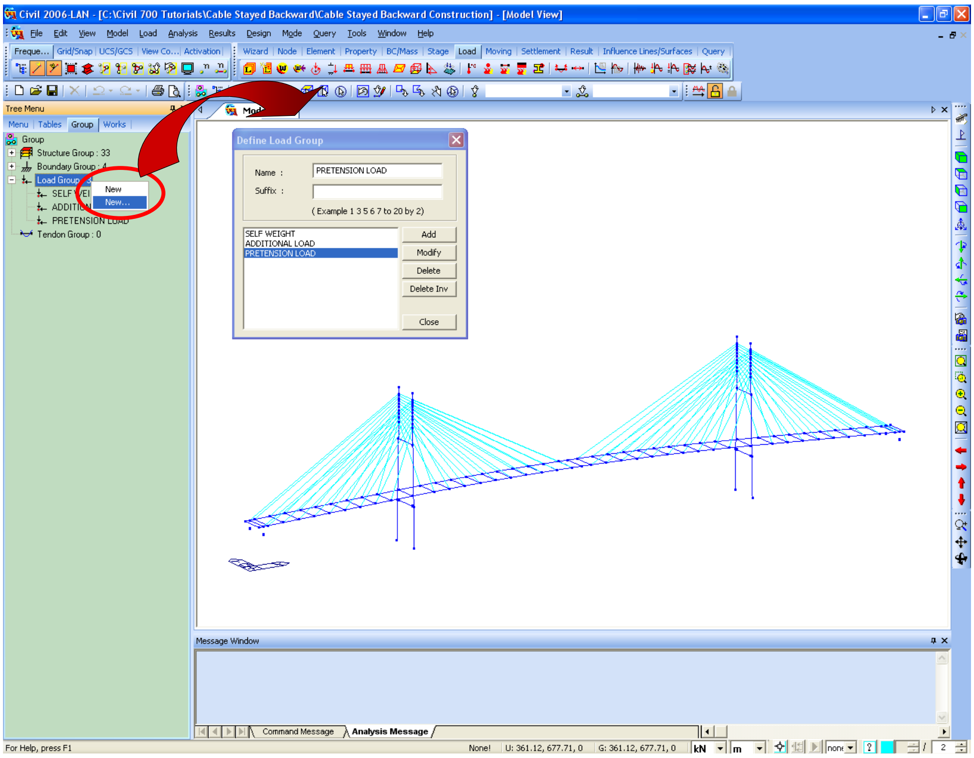


Fig. 49 Defining Load Group

Modify the Load Group "Default", which was defined for self-weight in the final stage analysis, to "Self Weight".

Model / Load / ***Self Weight***

Load Case Name>**SelfWeight**

Load Group Name>**SelfWeight**

Operation>Modify

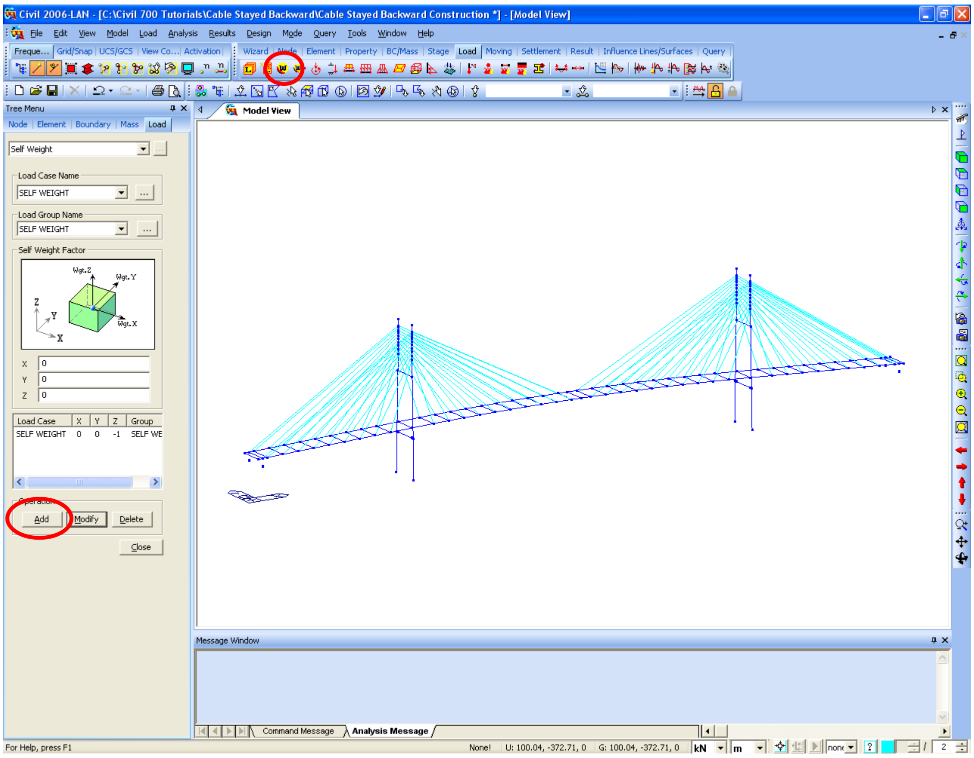


Fig. 50 Modifying Load Group for Self-Weight

Reassign the superimposed dead load and initial cable prestress, which were defined for the final stage analysis, to Load Group.

Select All

Group > Load Group

  Additional Load (*Drag & Drop*)

  Select Load Type > **Beam Loads** (on) ↵

Select All

Group > Load Group

  Pretension Load (*Drag & Drop*)

  Select Load Type > **Pretension Loads** (on) ↵

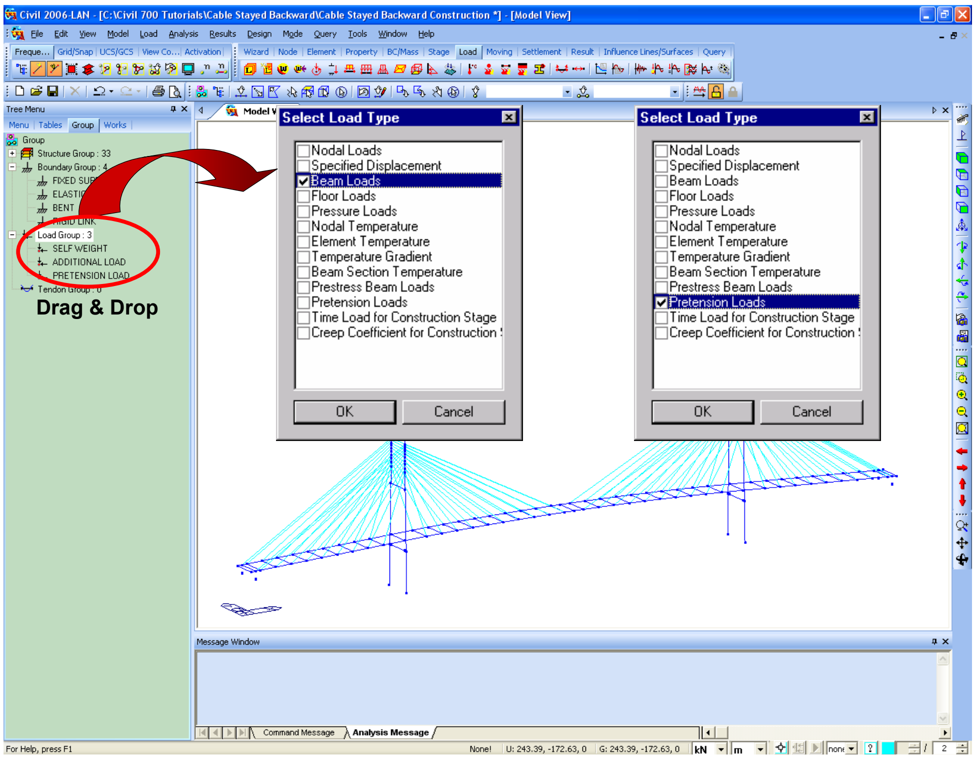


Fig. 51 Defining Load Group for Superimposed Dead Load and Initial Cable Prestress

Assign Construction Stage

We now assign the predefined Structure Group, Boundary Group and Load Group to each corresponding construction stage. First, we assign the final stage (CS0) to Construction Stage as the 1st stage in backward analysis.

Load / Construction Stage Analysis Data / ***Define Construction Stage***

CS0 Modify/Show

Save Result>**Stage** (on)

Element tab>Group List > **SG0**; Activation> Add

Boundary tab>Group List > **Fixed Support, Elastic Link, Rigid Link**

Support / Spring Position>**Original**

Activation> Add

Load tab> Group List>**SelfWeight, Additional Load, Pretension**

Activation> Add ↵

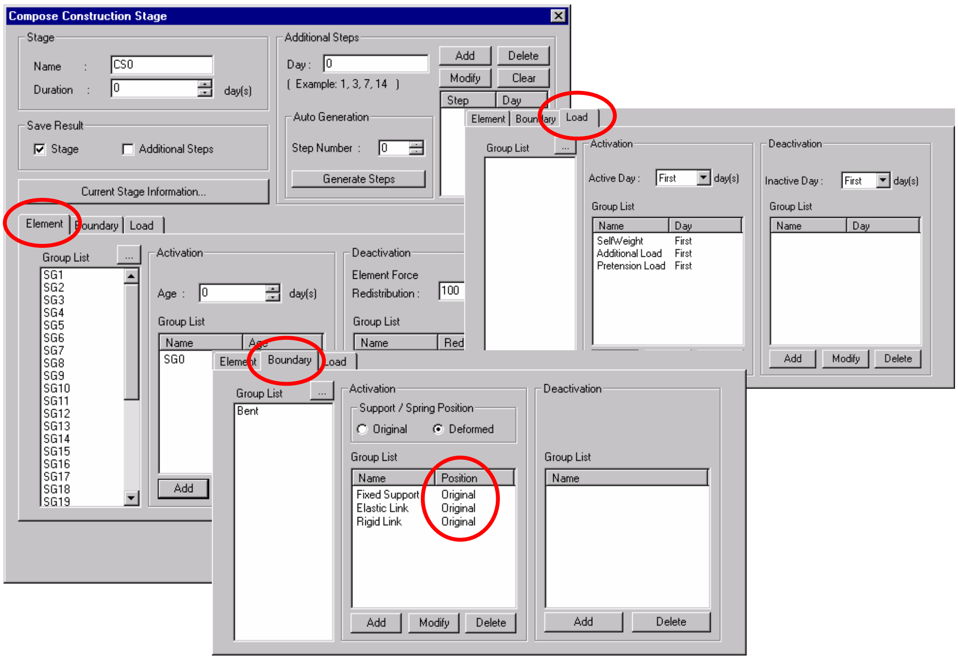


Fig. 52 Defining Elements, Boundary Conditions and Loads for Construction Stage CS0

Define Construction Stage for each construction stage from CS1 to CS32 using Table 5 Cannibalization Stage Category as follows:

CS1 Modify/Show

Save Result>**Stage** (on)

Load tab> Group List> **Additional Load**

Deactivation> Add ↵

CS2 Modify/Show

Save Result>**Stage** (on)

Element tab>Group List > **SG2**; Deactivation> Add

Element Force Redistribution> **100%**

Boundary tab>Group List > **Bent;** Support / Spring Position>**Original**

Activation> Add

CS3 to CS32 Modify/Show

Save Result>**Stage** (on)

Element tab>Group List > **SG3 to SG32**; Deactivation> Add

Element Force Redistribution> **100%**

Input Construction Stage Analysis Data

Analysis / ***Construction Stage Analysis Control***

Final Stage>**Last Stage** (on)

Analysis Option>**Include Time Dependent Effect** (off) ↵

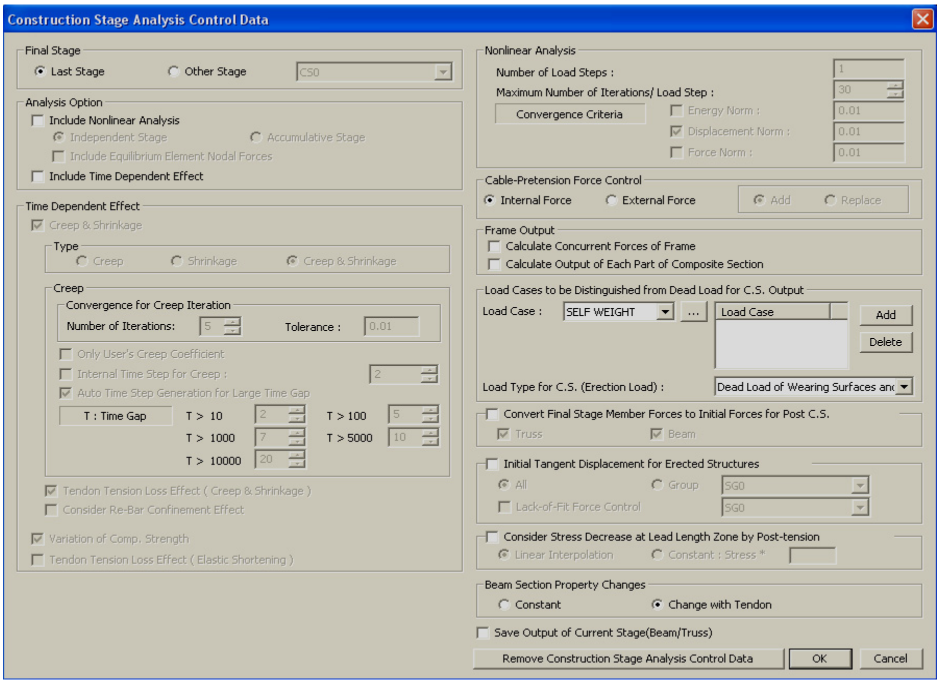


Fig. 53 Construction Stage Analysis Control Data Dialog Box

Perform Structural Analysis

Perform construction stage analysis for self-weight, superimposed dead load and initial cable prestress.

Analysis / ***Perform Analysis*** ↵

Review Construction Stage Analysis Results

Review the changes of deformed shapes and section forces for each construction stage by construction stage analysis.

Review Deformed Shapes

Review the deformed shape of the main girders and towers for each construction stage.

If the Stage Toolbar is active, the analysis results can be easily monitored in the Model View by selecting construction stages using the arrow keys on the keyboard.

If the default Deformation Scale Factor is too large, we can adjust the Scale Factor.

Stage Toolbar>**CS 5** (A in Fig. 54)

Result / Deformations / *Deformed Shape*

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

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

Deform ...

Deformation Scale Factor **(0.5)** ↵

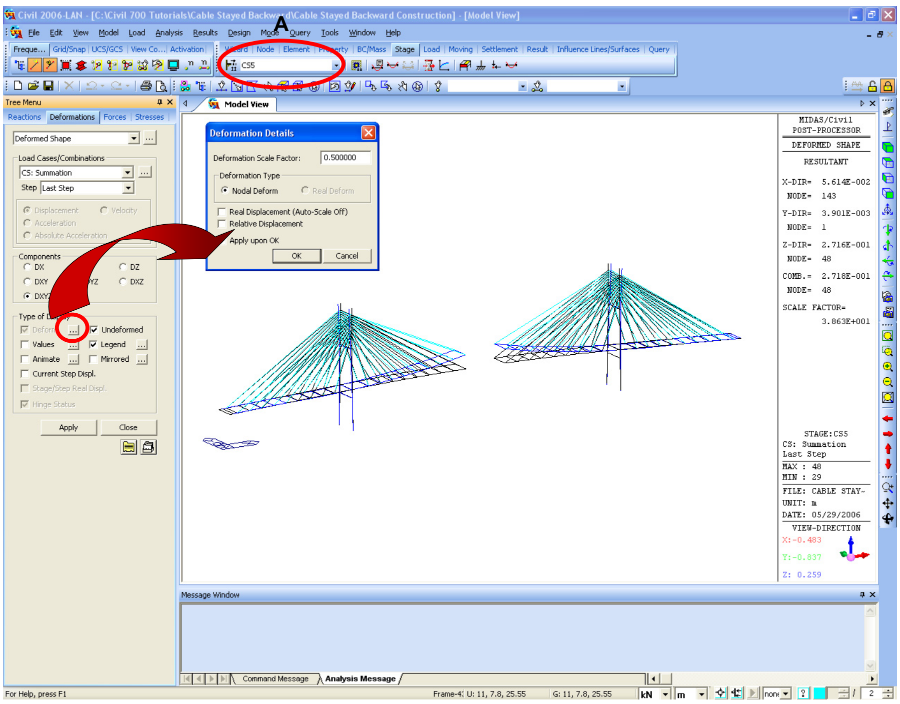


Fig. 54 Deformed Shape for Each Construction Stage from Backward Analysis

Review Bending Moments

For each construction stage, we review bending moments for the main girders and towers.

Stage Toolbar>**CS 7**

Result / Forces / ***Beam Diagrams***

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

Components>**My**

Display Options>**5 Points**; **Line Fill ;** Scale>**(1.0000)**

Type of Display>**Contour** (on); **Deform** (off), **Legend** (on) ↵

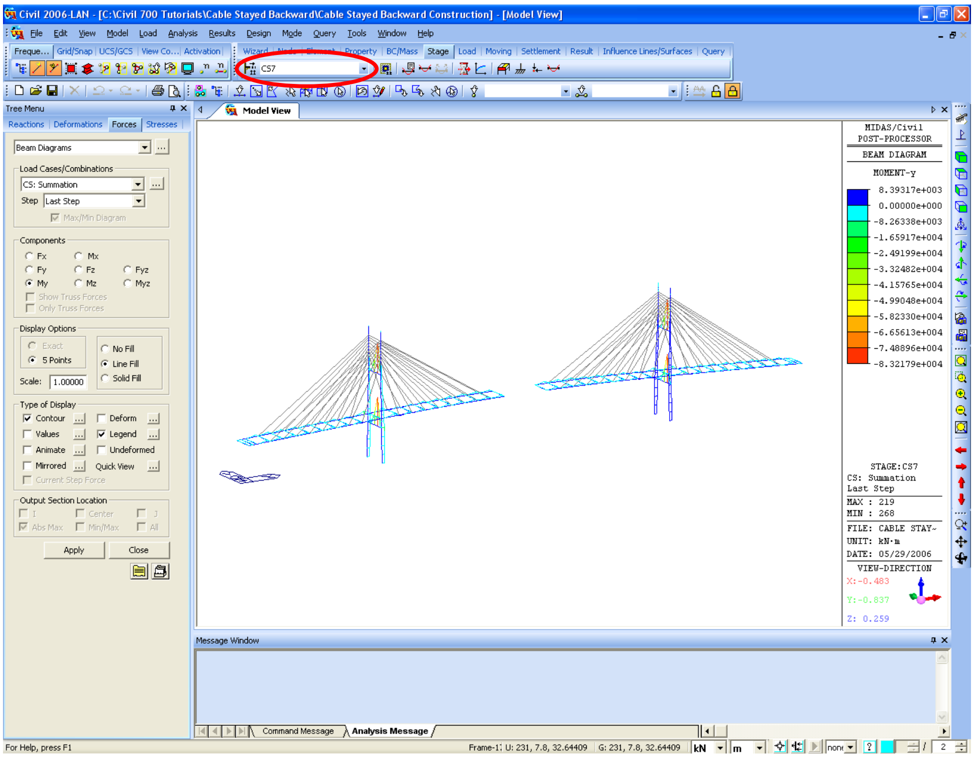


Fig. 55 Bending Moment Diagram for Each Construction Stage from Backward Analysis

Review Axial Forces

For each construction stage, we review axial forces for cables.

Stage Toolbar>**CS 15**

Result / Forces / ***Truss Forces***

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

Force Filter>**All;** Type of Display>**Legend** (on) ↵

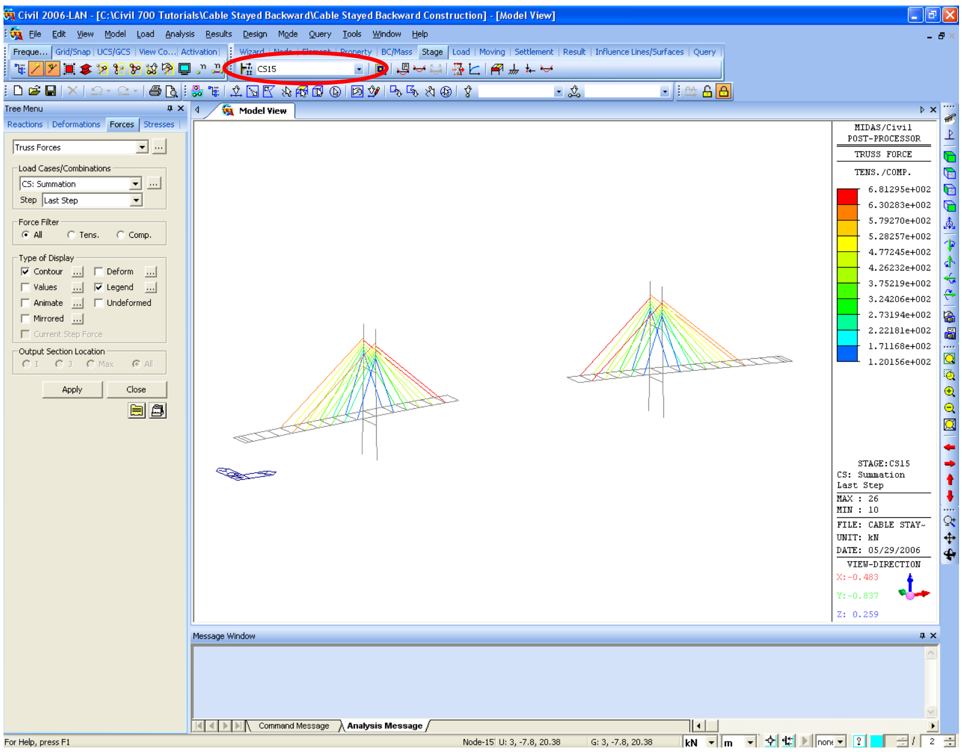


Fig. 56 Axial Forces for Each Construction Stage from Backward Analysis

Construction Stage Analysis Graphs

We will review deformed shapes of the main girders and towers for each construction stage using construction stage analysis graphs. For each construction stage, we review horizontal displacements for the towers and vertical displacements for the main girders at the ¼ point location of a side span.

Status Bar > kN, mm

Result / ***Stage/Step History Graph***

Define Function>**Displacement>** Add New Function

Displacement>Name **(Horizontal Disp.)**; Node Number **(1);** Components>**DX** ↵

Define Function>**Displacement>** Add New Function

Displacement>Name (**Vertical Disp.**); Node Number **(27);** Components>**DZ** ↵

Mode>**Multi Func.**; Step Option>**Last Step**; X-Axis>**Stage/Step**

Check Functions to Plot>**Horizontal Disp.** (on)**, Vertical Disp.** (on)

Load Cases/Combinations>**Summation**

Graph Title **(Horizontal & Vertical Displacements for each CS**), Graph

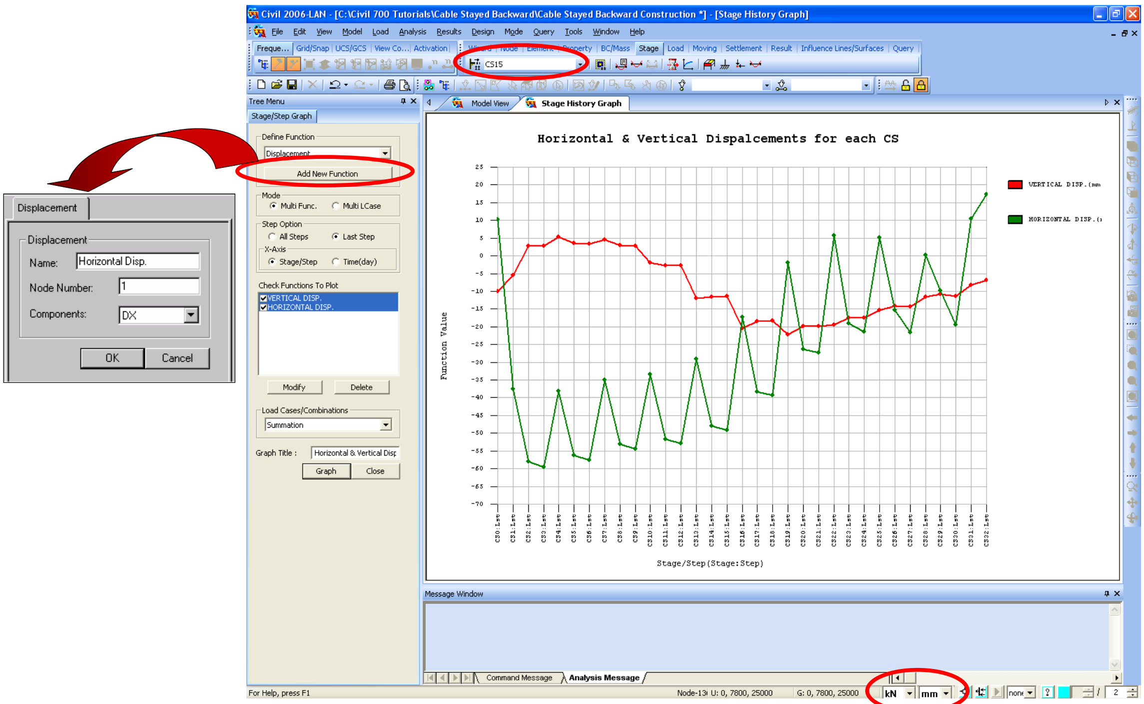


Fig. 57 History Graph of Deformed Shape for Each Construction Stage

Review the variation of cable prestress by using the Step History Graph function. Check the variation of cable tension forces for each construction stage for inner cables in the tower area from the final stage (CS0) to the last stage (CS32) in construction stage analysis.

Result / ***Stage/Step History Graph***

Define Function>**Truss Force/Stress**> Add New Function

Truss Force/Stress>Name **(Cable 10)**; Element No **(10)**; **Force** (on); Point>**I- Node** ↵

Define Function>**Truss Force/Stress**> Add New Function

Truss Force/Stress>Name **(Cable 11)**; Element No **(11)**; **Force** (on); Point>**I- Node** ↵

Mode>**Multi Func.**; Step Option>**Last Step**; X-Axis>**Stage/Step**

Check Functions to Plot>**Cable 10** (on)**, Cable 11** (on)

Load Cases/Combinations>**Summation**

Graph Title **(Variation of Cable Tension for each CS)** Graph

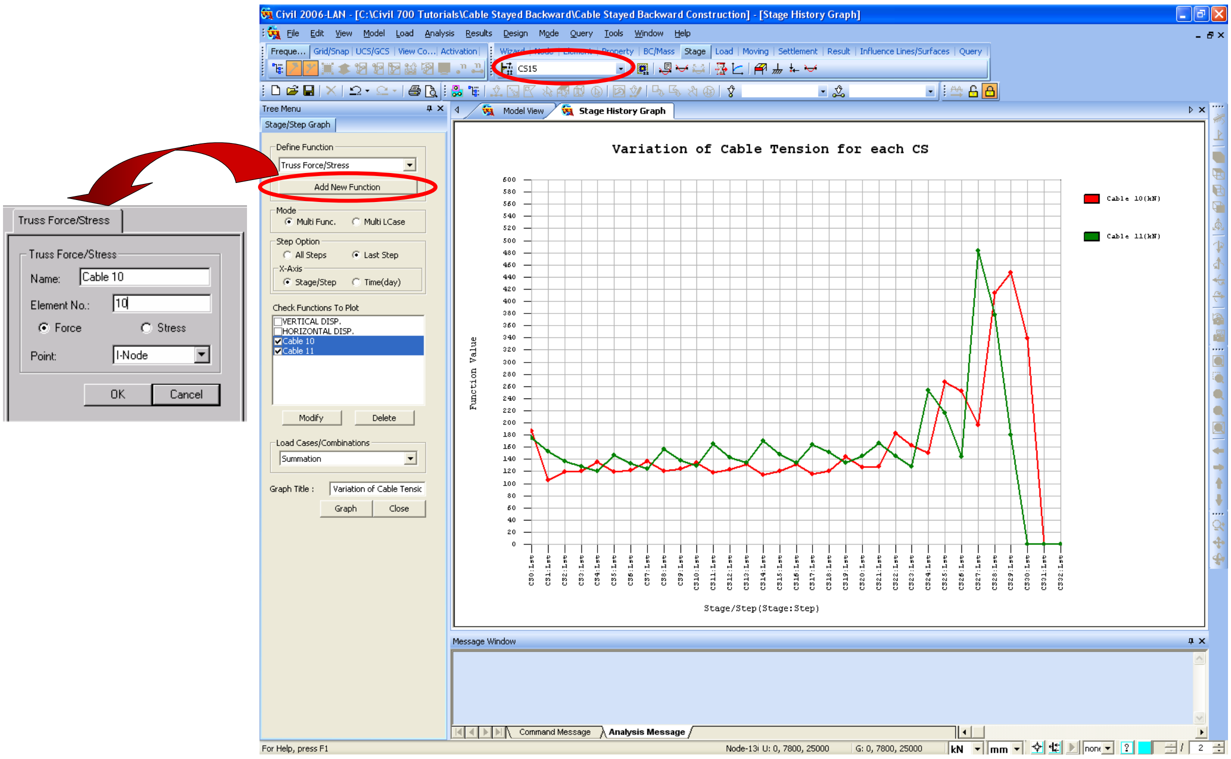


Fig. 58 Cable Tension Force Variation Graph for Each Construction Stage

Review the variation in the bending moments for the main girders and towers by using the Step History Graph function. Review the variation of bending moments for each construction stage for the lower part of the tower and ¼ point location of the main girder in a side span.

Status Bar > kN, m

Result / ***Stage/Step History Graph***

Define Function>**Beam Force/Stress,** Add New Function

Beam Force / Stress>Name **(Moment of Girder)**; Element No **(45); Force** (on)

Point>**I- Node;** Components>**Moment–y** ↵

Define Function>**Beam Force/Stress,** Add New Function

Beam Force / Stress>Name **(Moment of Tower)**; Element No **(108); Force** (on)

Point>**I- Node;** Components>**Moment–y** ↵

Mode>**Multi Func.**; Step Option>**Last Step**; X-Axis>**Stage/Step**

Check Functions to Plot>**Moment of Girder** (on), **Moment of Tower** (on)

Load Cases/Combinations>**Summation**

Graph Title **(Bending Moment for each CS)**, Graph

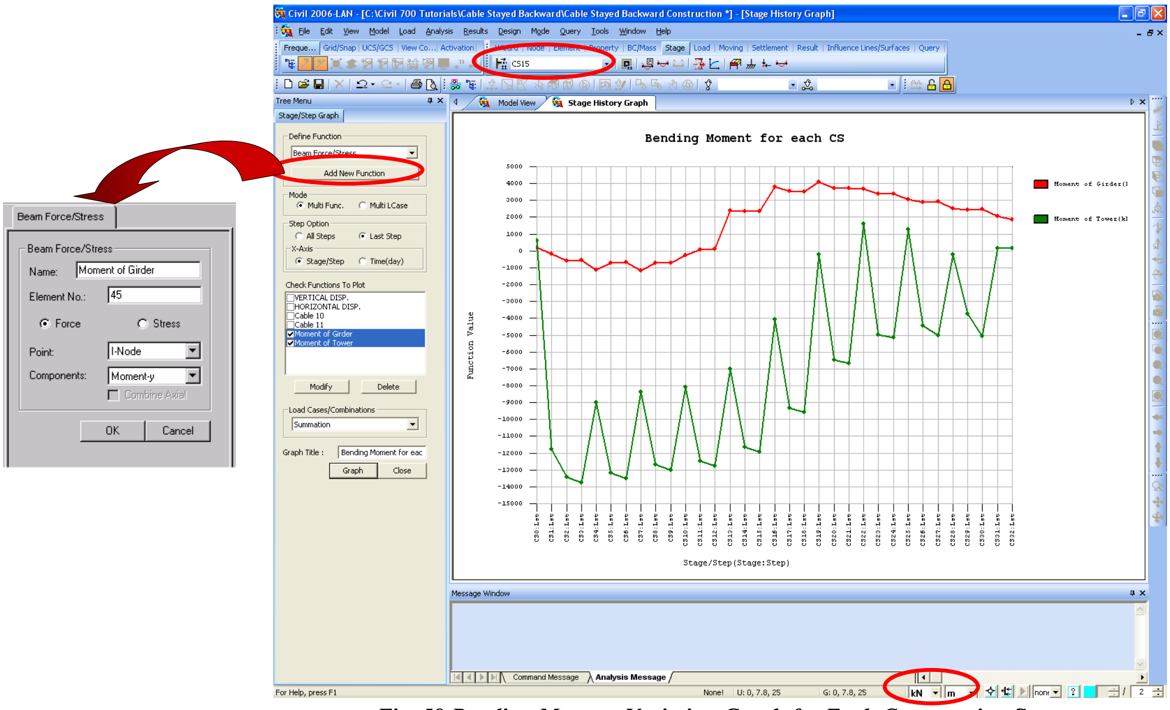


Fig. 59 Bending Moment Variation Graph for Each Construction Stage