Advanced Application 10

Final and Forward 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.

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.

In general, with forward construction stage analysis, we cannot obtain cable pretension loads for each stage which satisfy the initial equilibrium state at the final stage. By using cable pretension loads resulting from backward stage analysis, we can perform forward stage analysis. However, newly added function, Lack-of-Fit Force finds cable pretension loads for each construction stage from cable pretension loads at the final stage without backward stage analysis.

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 110 m center span and 40 m side spans. Fig. 1 below shows the bridge layout.

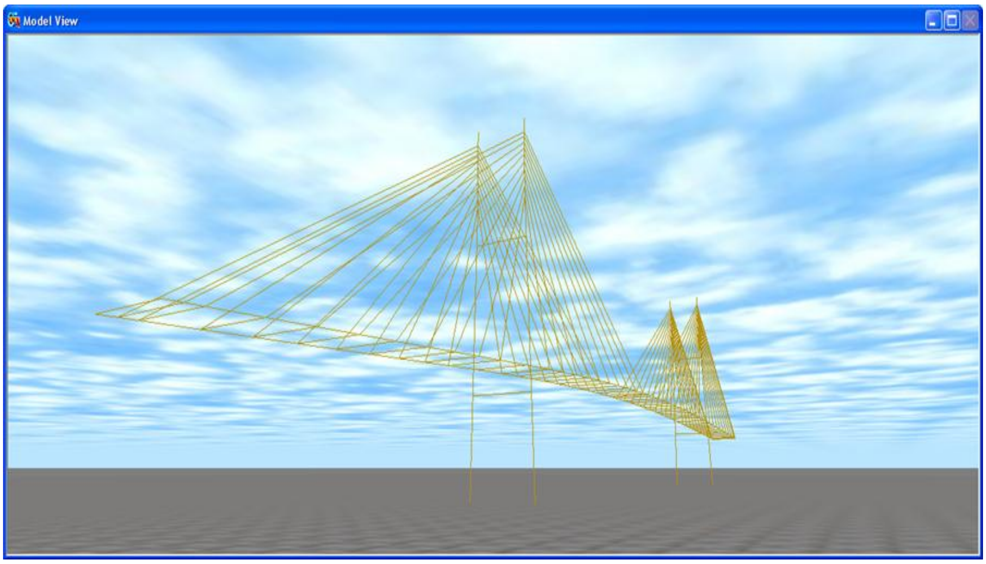


Fig. 1 Cable-stayed bridge analytical model

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
Bridge length	L = 40 *m*+110 *m*+40 *m* = 190 *m*
Bridge Height	Lower part of tower: 20 *m*, Upper part of tower: 40 *m*

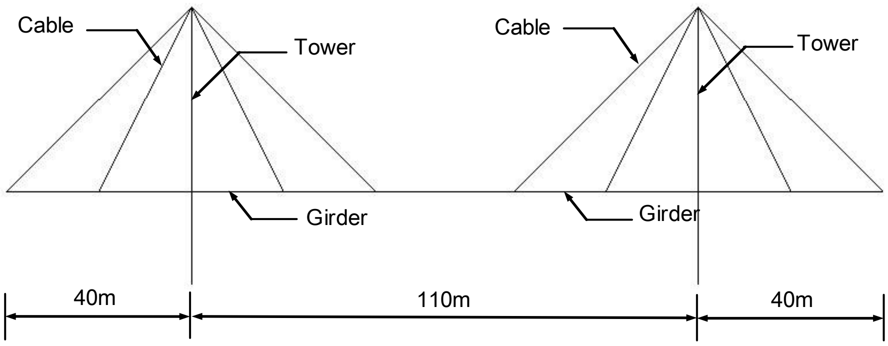


Fig. 2 General layout

Loading

Classification	Loading Type	Loading Value
Dead Load	Self weight	Automatically calculated within the program
Cable Prestess Force	Pretension Load	Cable prestress forces that satisfy initial equilibrium state at the final stage
Derrick Crane	Nodal Load	80 tonf
Jack Up Load	Specified Displacement	10 cm

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

Working Condition Setting

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

> ***New Project***

> *Save* **(Cable Stayed Forward)**

Tools Tab > ***Unit System***

Length>**m**; Force (Mass)>**tonf** ↵

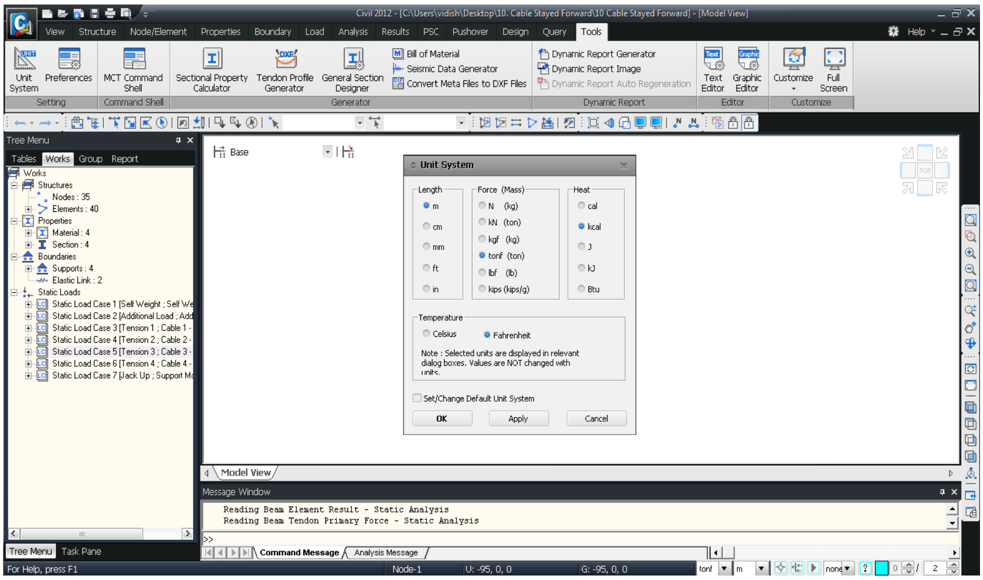


Fig. 3 Assign Working Condition and Unit System

Definition of Material and Section Properties

Input material properties for the main girders, tower-bottom, tower-top and cables. Click Add button under Material tab in Properties dialog box.

Properties Tab > ***Material Properties***

Name **(Girder);** Type of Design>**User Defined**

Modulus of Elasticity **(2.1e7);** Poisson's Ratio **(0.3)**

Weight Density **(7.85)**↵

Input material properties for the tower-bottom, tower-top and cables similarly. The input values are shown in Table 1.

Table 1 Material Properties

ID	Component	Modulus of Elasticity (tonf/m^2)	Poisson's Ratio	Weight Density (tonf/m^3)
1	Girder	2.1×10^7	0.3	7.85
2	Lower Tower	2.5×10^6	0.17	2.5
3	Upper Tower	2.1×10^7	0.3	7.85
4	Cable	1.57×10^7	0.3	7.85

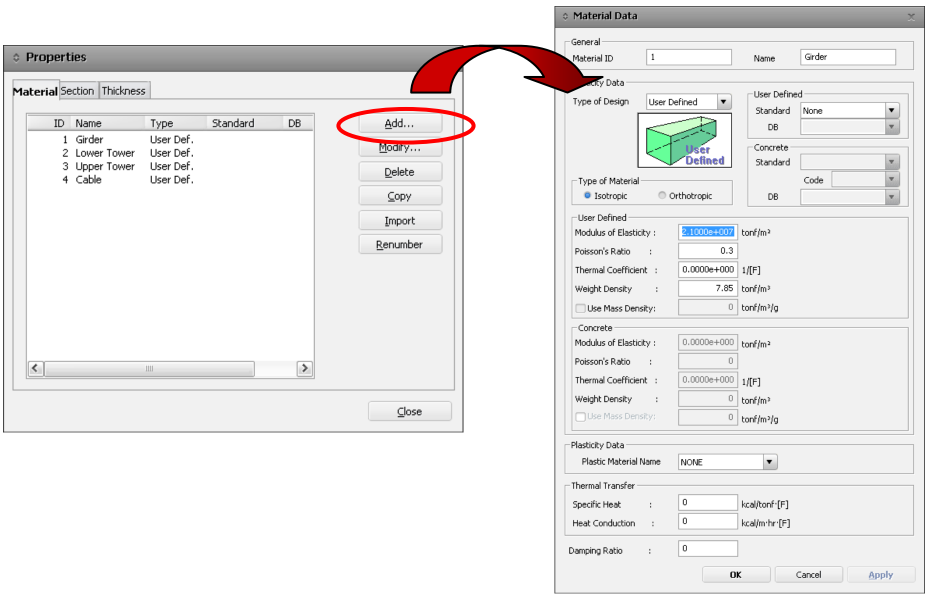


Fig. 4 Defined Material Properties

Input section properties for the girders, tower-bottom, tower-top and cables. Click [Add] button under Section tab in Properties dialog box.

Properties Tab > *Section Properties*

Value tab

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

Section Shape>**Solid Rectangle;** Stiffness>Area **(0.8)** ↵

Input section properties for the tower-bottom, tower-top and cables similarly. The values are shown in Table 2.

Table 2 Section Properties

ID	Component	Area (m^2)	Ixx (m^4)	Iyy (m^4)	Izz (m^4)
1	Girder	0.8	15.0	1.0	15.0
2	Lower Tower	50.0	1000.0	500.0	500.0
3	Upper Tower	0.3	5.0	5.0	5.0
4	Cable	0.005	0.0	0.0	0.0

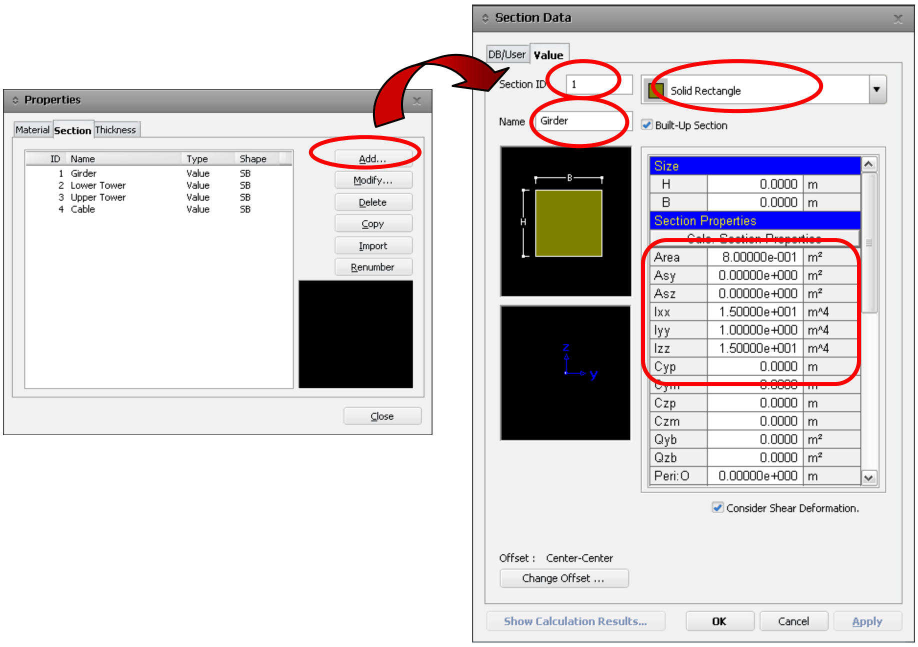


Fig. 5 Defined Section Properties

Final Stage Analysis

After completion of the final stage modeling for the cable-stayed bridge, we calculate the cable initial 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**. 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. 6.

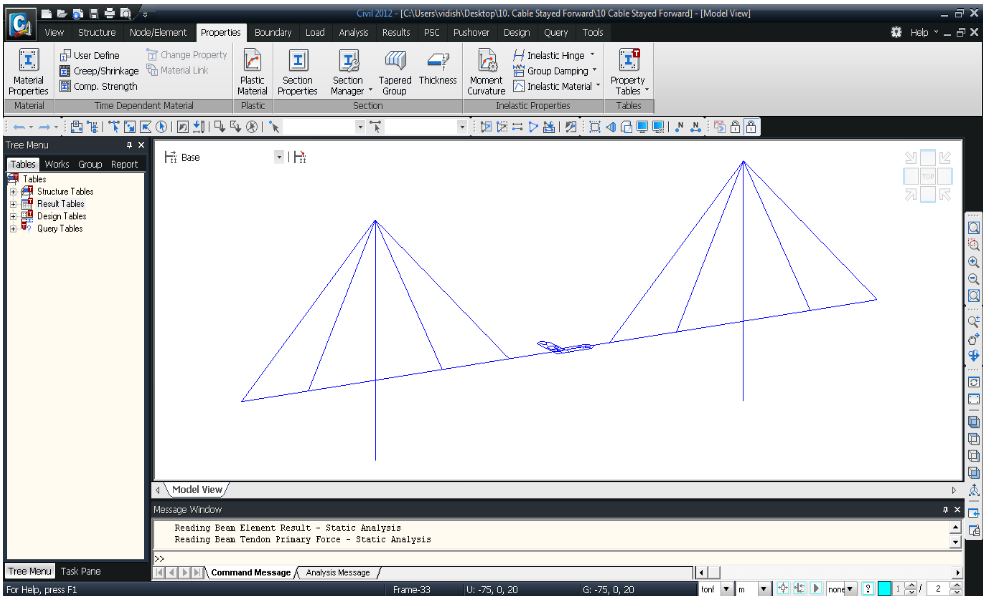


Fig. 6 Final Model for Cable-Stayed Bridge

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. Main Girder Modeling
2. Tower Modeling
3. Cable Modeling
4. Tower Bearing Generation
5. Boundary Condition Input
6. Initial cable Prestress Force Calculation by Unknown Load Factors
7. Loading Condition and Loading Input
8. Perform Structural Analysis
9. Unknown Load Factors Calculation

Main Girder Modeling

First generate nodes, and then model the girder (9@10+2@5+9@10m) by using the Extrude Element function.

Front View, ***Node Snap*** (on), ***Element Snap*** (on)
Auto Fitting (on), ***Node Number*** (on)
Node /Element > ***Create Nodes***
Coordinates **(-95, 0, 0)** ↵

Nodes /Elements > ***Extrude Elements***
Select All
Extrude Type>**Node→ Line Element**
Element Attribute>Element Type>**Beam**
Material>**1 : Girder** ; Section>**1 : Girder**
Generation Type>**Translate**
Translation>**Unequal Distance** ; Axis>**x**
Distances>**9@10, 2@5, 9@10** ↵

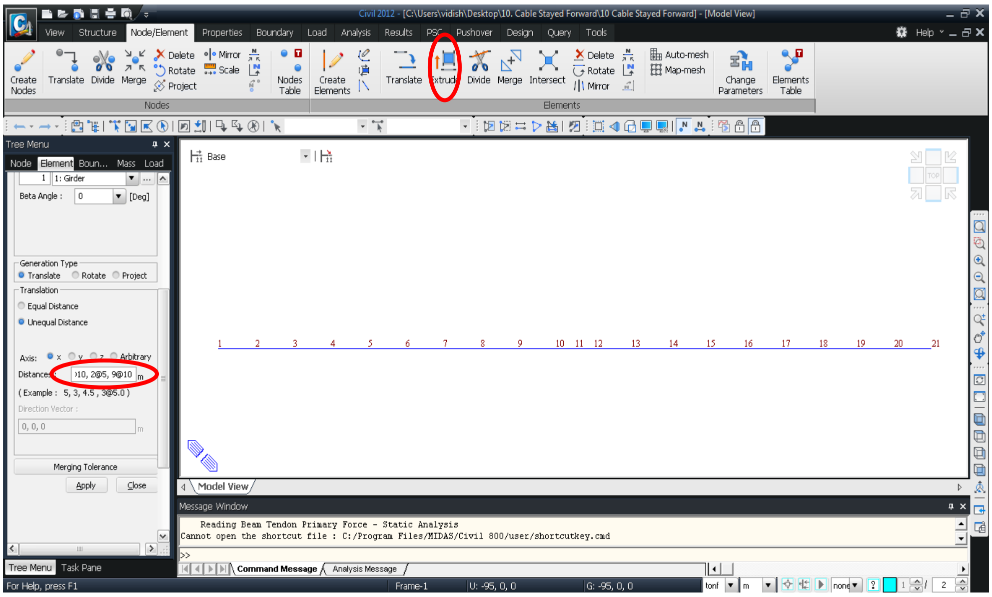


Fig. 7 Generation of Main Girders

Tower Modeling

First generate nodes at the lower ends of the towers, and then model Lower Tower (10m+5m) using the Extrude Element function.

Node /Element> ***Create Nodes***

Coordinates **(-55 , 0, -20)**

Copy>Number of Times **(1)** ; Distance **(110, 0, 0)** ↵

Node /Element > ***Extrude Elements***

Select Window (Nodes : ① in Fig. **8** ; **Node 22, 23**)

Extrude Type>**Node→Line Element**

Element Attribute>Element Type>**Beam**

Material>**2 : Lower Tower** ; Section>**2 : Lower Tower**

Generation Type>**Translate**

Translation>**Unequal Distance** ; Axis>**z**

Distances>**10, 5** ↵

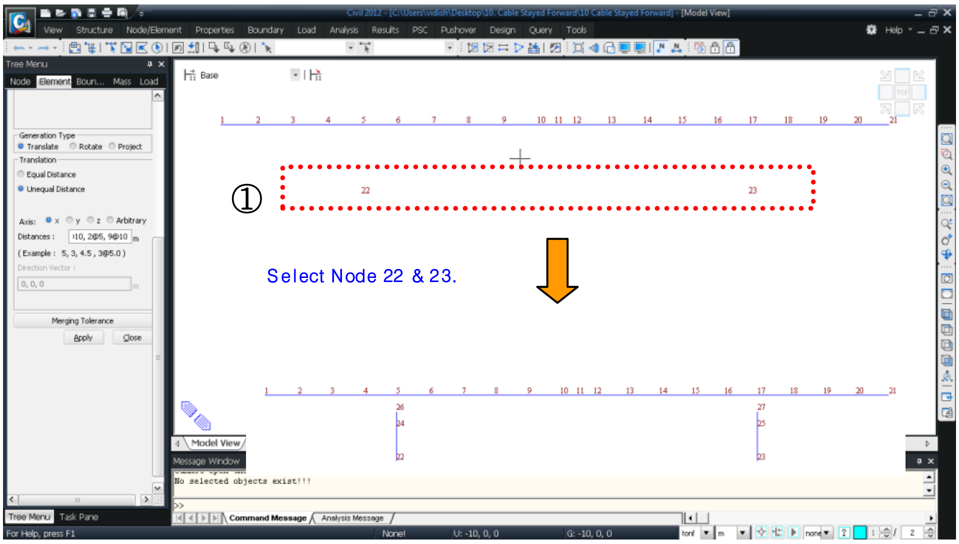


Fig. 8 Generation of Lower Tower

To generate the Upper Tower (10m+5m+3@10m), select nodes and use the Extrude Element function.

Node /Element > ***Extrude Elements***

Select Window (Nodes : ① in Fig. 9 ; **Node 26, 27**)

Extrude Type>**Node→ Line Element**

Element Attribute>Element Type>**Beam**

Material>**3 : Upper Tower** ; Section>**3 : Upper Tower**

Generation Type>**Translate**

Translation>**Unequal Distance** ; Axis>**z**

Distances>**15, 3@10** ↵

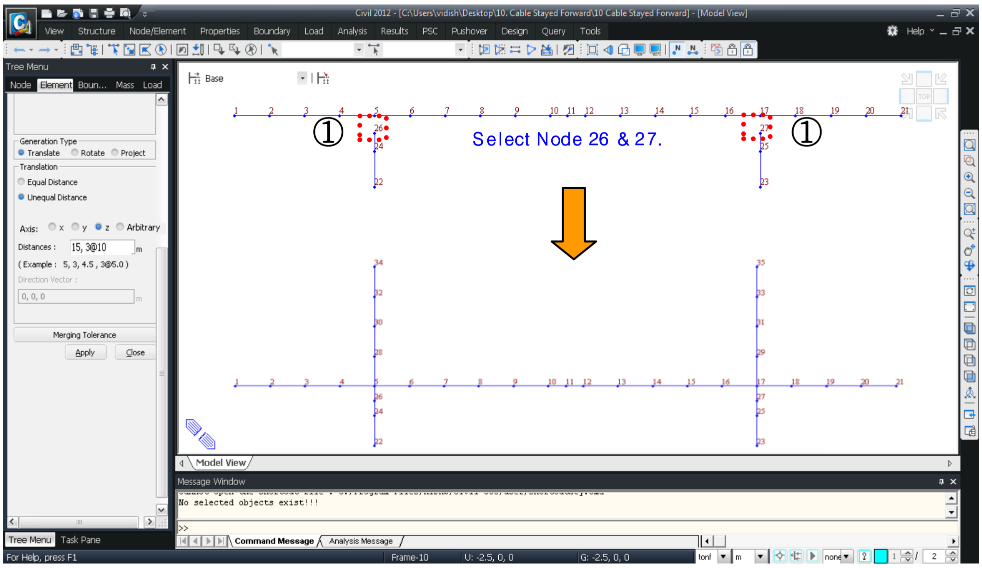
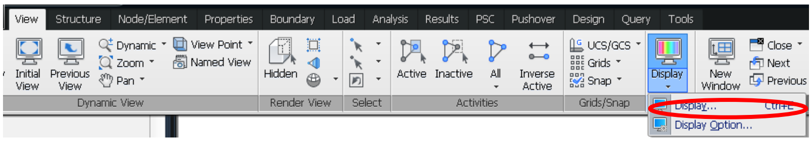


Fig. 9 Generation of Upper Tower

Cable Modeling

Generate cable elements using Truss of the Create Element function. Also check Element's Local Axes during the generation of cables.



Element> **Local Axis**(on) ↵

Node /Element > ***Create Elements***

Element Type>**Truss**

Material>**4: Cable** ; Section>**4: Cable;** Beta Angle **(0)**

Nodal Connectivity **(34, 1)** ; Nodal Connectivity **(34, 3)**

Nodal Connectivity **(34, 7)** ; Nodal Connectivity **(34, 9)**

Nodal Connectivity **(35, 13)** ; Nodal Connectivity **(35, 15)**

Nodal Connectivity **(35, 19)** ; Nodal Connectivity **(35, 21)** ↵

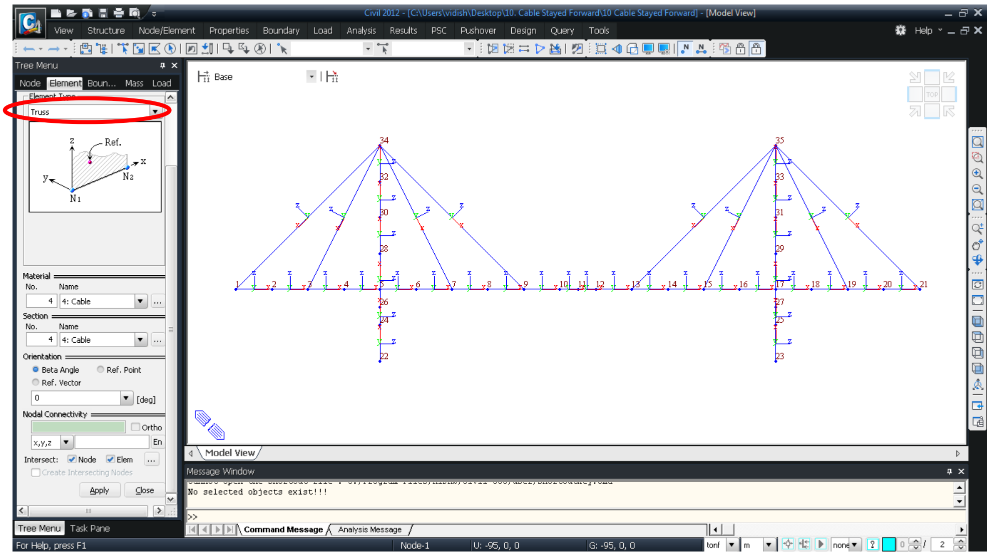


Fig. 10 Generation of Cables

Tower Bearing Generation

Model the tower bearings using the Elastic Link elements.

Bearing properties are as follows:
SDx: 500,000 tonf/m
SDy: 100,000,000 tonf/m
SDz: 1,000 tonf/m

Boundaries Tab > ***Elastic Link***
Zoom Window (① in Fig. 11)
Options > **Add** ; Link Type > **General Type**
SDx (tonf/m) **(500000)** ; SDy(tonf/m) **(100000000)** ; SDz(tonf/m) **(1000)**
Shear Spring Location (on)
Distance Ratio From End I : SDy **(1)** ; SDz **(1)**
Beta Angle > **(0)**
2 Nodes **(26,5)**
2 Nodes **(27,17)** ↵

Stiffness of link is defined as the force required for unit displacement. Rotational Stiffness is defined as moment required for unit rotation (in radians).

Beta Angle is entered to define the orientation of member.

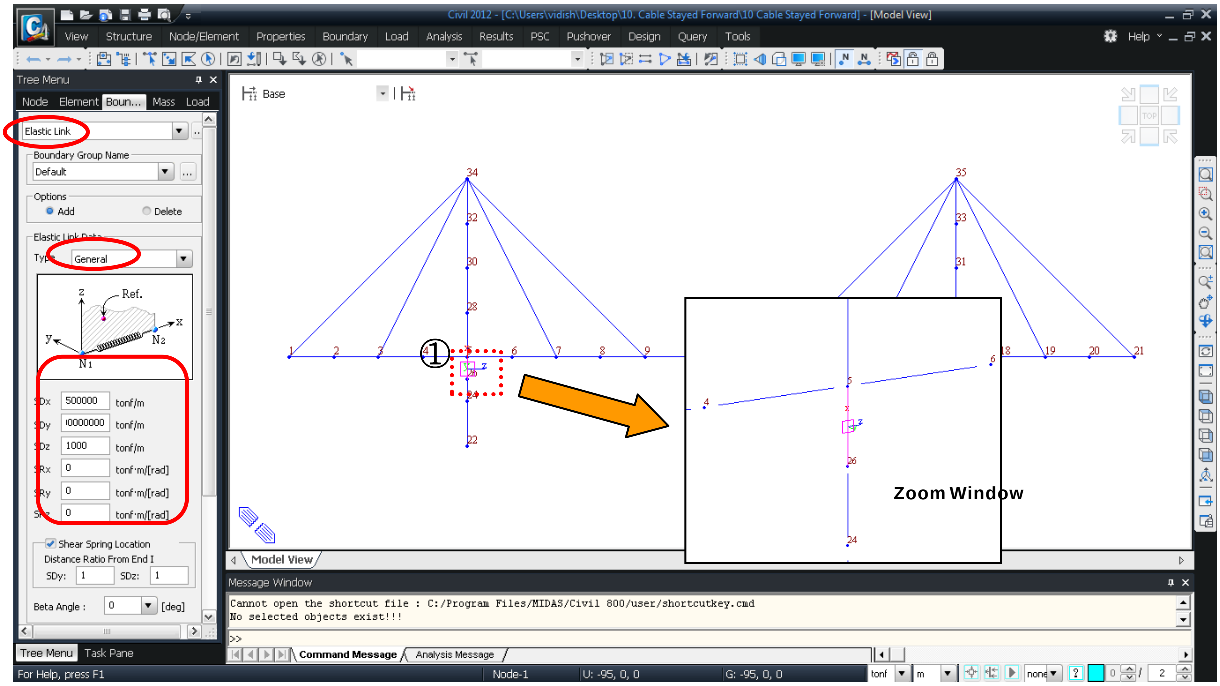


Fig. 11 Tower Bearing Generation

Boundary Condition Input

Boundary conditions for the analytical model are as follows:

- Tower base: Fixed condition (Dx, Dy, Dz, Rx, Ry, Rz)
- Pier base: Hinge condition (Dy, Dz, Rx, Rz)

Input boundary conditions for the tower and pier bases.

Auto Fitting

Boundary Tab > ***Supports***

Select Window (Nodes : ① in Fig. **12** ; **Node 22, 23**)

Boundary Group Name > **Default**

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

Select Window (Nodes : ② in Fig. **12 ; Node 1, 21**)

Boundary Group Name > **Default**

Options > **Add** ; Support Type > **Dy, Dz, Rx, Rz** ↵

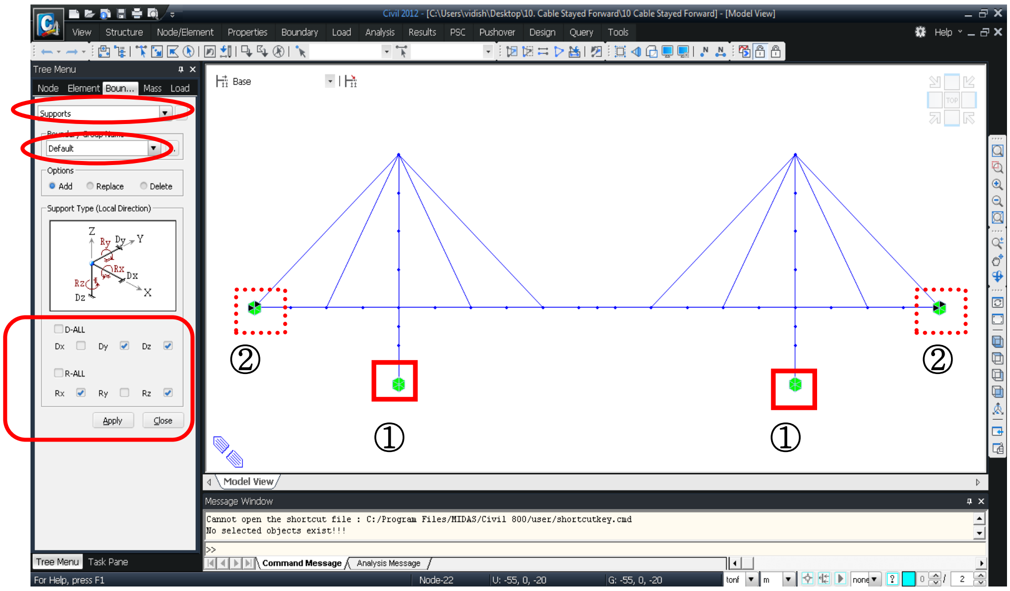


Fig. 12 Specifying Fixed Boundary Conditions for Tower and Pier Bases

Cable Initial Prestress Calculation

The initial cable prestress, which is balanced with dead loads, is introduced to improve section forces in the main girders and towers, 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.

Table 3. Flowchart for Cable Initial Prestress Calculation

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

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 4, as the bridge is a symmetric cable-stayed bridge, which has 4 cables on each side of each tower. Input loading conditions for each of the 4 cables.

Load Tab > ***Static Load Cases***

Name **(Self Weight)**; Type>**Dead Load**

It may be more convenient to use the MCT Command Shell for the input of loading conditions.

Description **(Self Weight)** ↵

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

Description **(Additional Load)** ↵

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

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

....

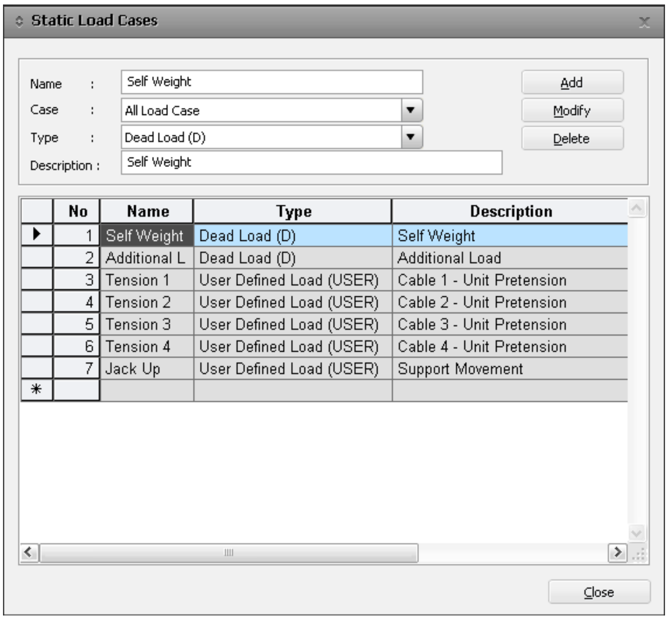
Name **(Tension 4)**; Type>**User Defined Load**

Description **(Cable4- UNIT PRETENSION)** ↵

Name **(Jack Up)**; Type>**User Defined Load**

Description **()** ↵

Input the loading conditions repeatedly from Name (Tension 1) to Name (Tension 4).



No	Name	Type	Description
1	Self Weight	Dead Load (D)	Self Weight
2	Additional L	Dead Load (D)	Additional Load
3	Tension 1	User Defined Load (USER)	Cable 1 - Unit Pretension
4	Tension 2	User Defined Load (USER)	Cable 2 - Unit Pretension
5	Tension 3	User Defined Load (USER)	Cable 3 - Unit Pretension
6	Tension 4	User Defined Load (USER)	Cable 4 - Unit Pretension
7	Jack Up	User Defined Load (USER)	Support Movement

Fig. 18 Generation of Loading Conditions for Dead Loads and Unit Loads

Loading Input

Input the self-weight, superimposed dead load for the main girders, unit loads for the cables and Jack Up loads. 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.

Zoom Fit

Load Tab > ***Self Weight***

Load Case Name>**Self Weight**

Load Group Name>**Default**

Self Weight Factor>Z **(-1)** ↵

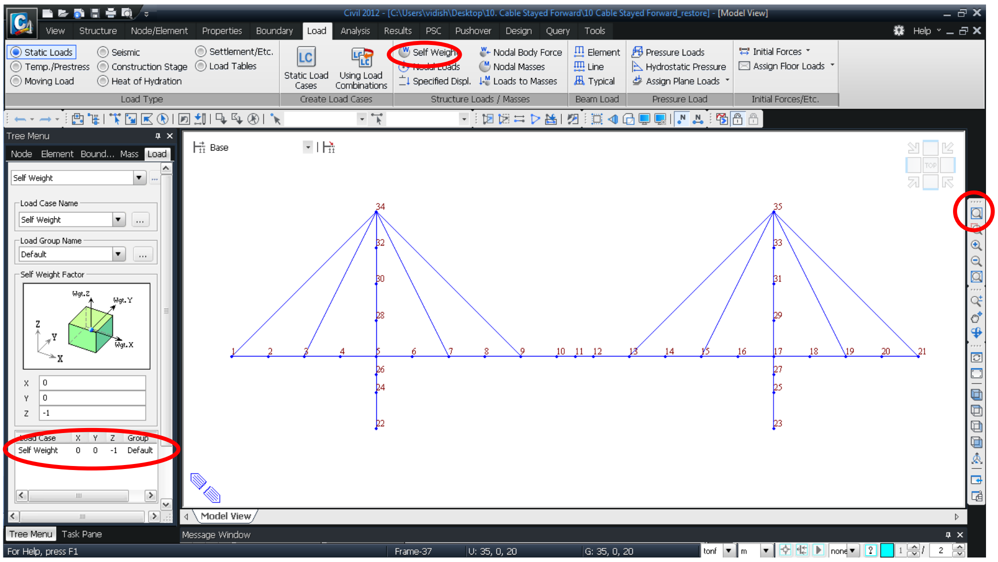


Fig. 19 Entering Self-Weight

Load superimposed dead loads for the main girders.

Input the superimposed dead load –3.0 tonf/m, which is due to barriers, pavement, etc by the ***Element Beam Loads*** function.

Load Tab > ***Element***

Select Window (Nodes : ① in Fig. **20** ; **Node 22, 23**)

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

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

Projection>**No**

Value>**Relative**; x1 **(0)**, x2 **(1)**, W **(-3)** ↵

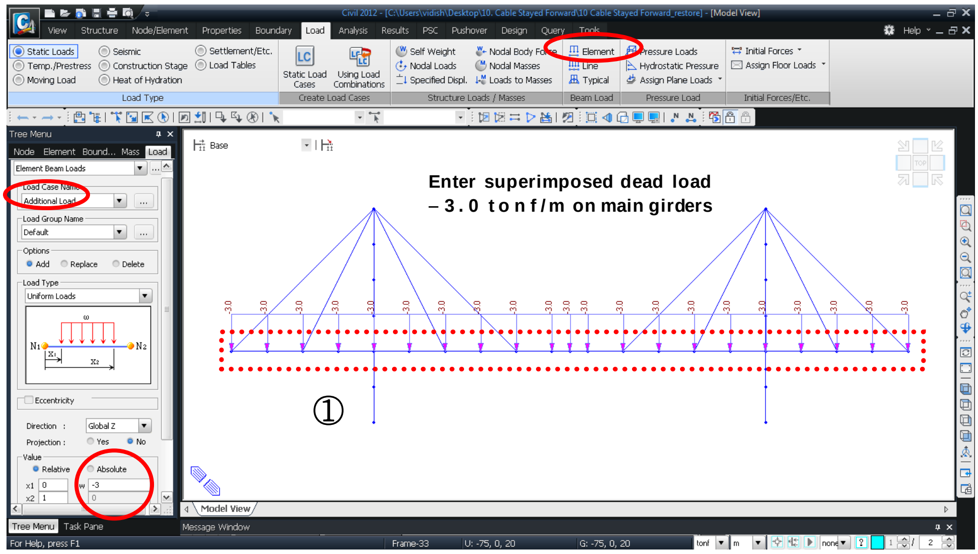


Fig. 20 Entering Superimposed Dead Loads to Main Girders

Input a unit pretension load to each cable. For the case of a symmetric cable-stayed bridge, identical cable initial 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.

Load Tab > Temp./ Prestress Option > ***Pretension Loads***

View Tab > Select Section > ***Select Intersect*** (Elements: ① in Fig. 21 ; **Element : 33, 40**)
Load Case Name>**Tension 1**; Load Group Name>**Default**
Options>**Add**; Pretension Load (**1**) ↵
…
Load Case Name>**Tension 4**; Load Group Name>**Default**
Options>**Add;** Pretension Load (**1**) ↵

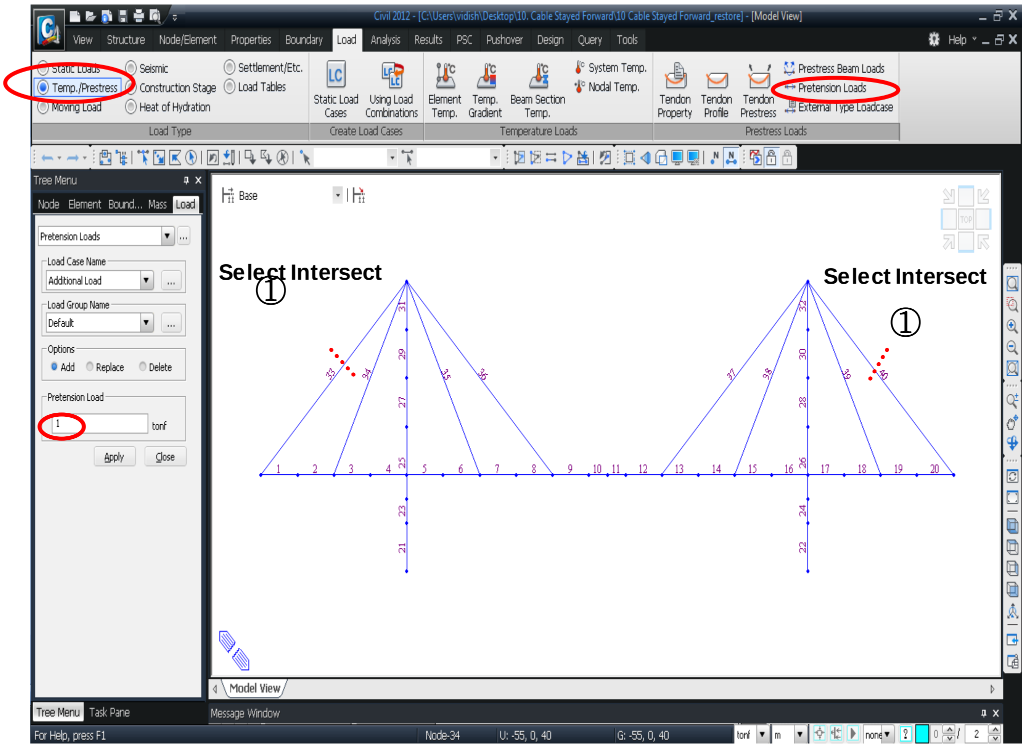


Fig. 21 Entering Unit Pretension Load to Cables

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

Table 4. Loading Conditions and Element Numbers

Load Case	Element No.	Load Case	Element No.
Tension 1	33, 40	Tension 3	35, 38
Tension 2	34, 39	Tension 4	36, 37

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

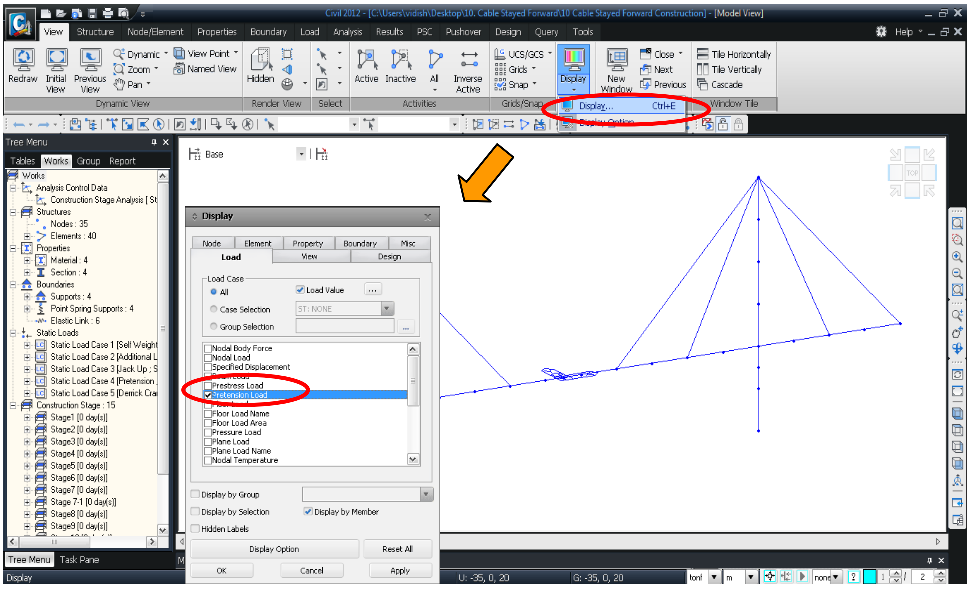


Fig. 22 Unit Pretension Loads entered for Cables

Enter Jack Up loads to the piers at each side span by the ***Specified Displacements of Supports***.

Jack Up load is as follows:

Vertical Displacement : 0.01 m

Specified displacements of supports are entered for arbitrary loads.

Load Tab > ***Specified Displacements of Supports***

Select Window (Nodes : ① in Fig. **23** ; **Node 1, 21**)

Load Case Name>**Jack Up** ; Options>**Add**

Displacements> Dz **(0.01)** ↵

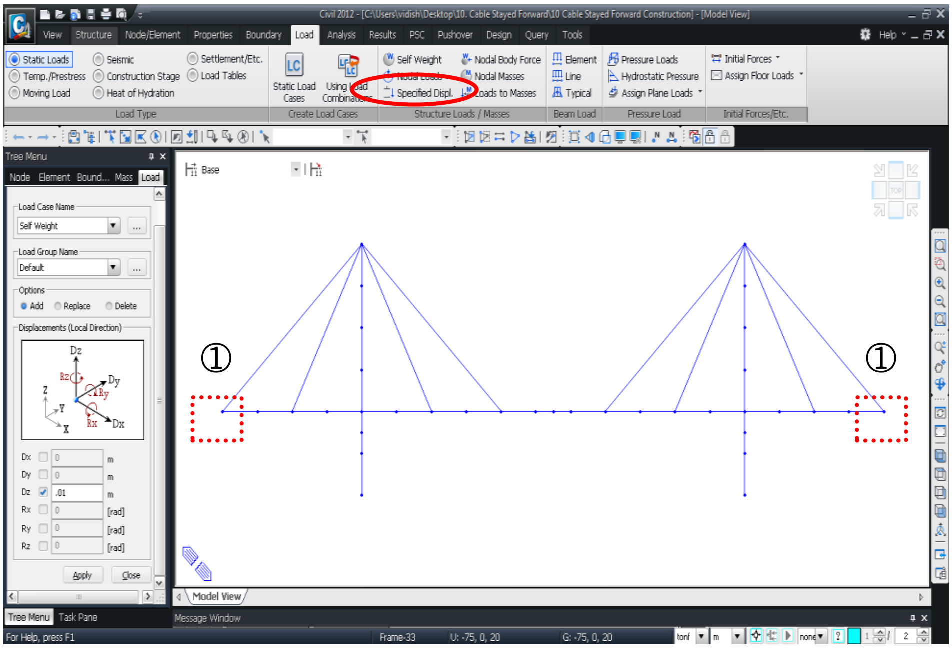


Fig. 23 Entering Jack Up Loads

Perform Structural Analysis

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

Analysis / ***Perform Analysis*** ↵

Final Stage Analysis Results Review

Load Combination Generation

Create load combinations using the 4 loading conditions for cable unit pretension loading, self-weights, superimposed dead loads and Jack Up loads.

Results Tab / ***Load Combination***

Load Combination List>Name>**LCB 1**

Active>**Active** ; Type>**Add**

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

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

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

LoadCase>**Tension 2(ST)**; Factor **(1.0)**

LoadCase>**Tension 3(ST)**; Factor **(1.0)**

LoadCase>**Tension 4(ST)**; Factor **(1.0)**

LoadCase > **Jack Up (ST)** ; Factor **(1.0)** ↵

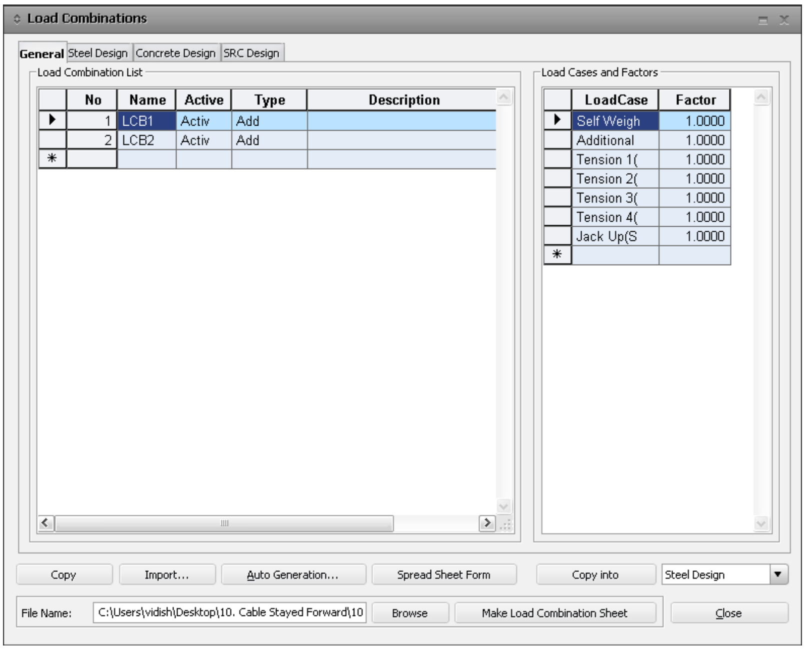


Fig. 24 Creating Load Combinations

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 horizontal deflection (Dx) of the tower and the bending moment (My) 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 Tab /Cable Control / ***Unknown Load Factor***

Unknown Load Factor Group> Add New

Item Name **(Unknown)**; Load Comb>**LCB 1**

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

LCase > **Tension 1** (on)

LCase > **Tension 2** (on)

LCase > **Tension 3** (on)

LCase > **Tension 4** (on)

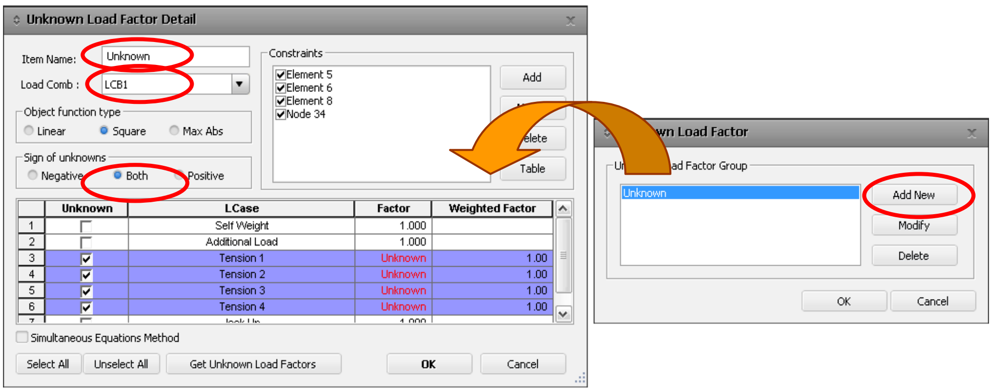


Fig. 25 Unknown Load Factor Detail Dialog Box

Specify the constraining conditions, which restrict the horizontal displacement (Dx) of the tower and the bending moment (My) of the main girders by the ***Constraints*** function.

Constraints > Add

Constraint Name **(Node 34)**

Constraint Type > **Displacement**

Node ID **(34)**

Component > **Dx**

Equality/Inequality Condition > **Equality** ;

Value **(0)** ↵

In this tutorial, we will apply constraints to restrict the horizontal displacement of the tower and the bending moment of the main girders. Since the analytical model is symmetric, we will apply the constraints to only half of the tower and the main girders.

Constraints > Add

Constraint Name **(Element 5)**

Constraint Type > **Beam Force**

Element ID **(5)**

Point > I-end

Component > **My**

Equality/Inequality Condition > **Equality** ;

Value **(-300)** ↵

Constraints > Add

Constraint Name **(Element 6)**

Constraint Type > **Beam Force**

Element ID **(6)**

Point > J-end

Component > **My**

Equality/Inequality Condition > **Equality** ;

Value **(-200)** ↵

Constraints > Add

Constraint Name **(Element 8)**

Constraint Type > **Beam Force**

Element ID **(8)**

Point > J-end

Component > **My**

Equality/Inequality Condition > **Equality** ;

Value **(-400)** ↵

The constraints for calculating Unknown Load Factors can be easily entered by MCT command Shell.

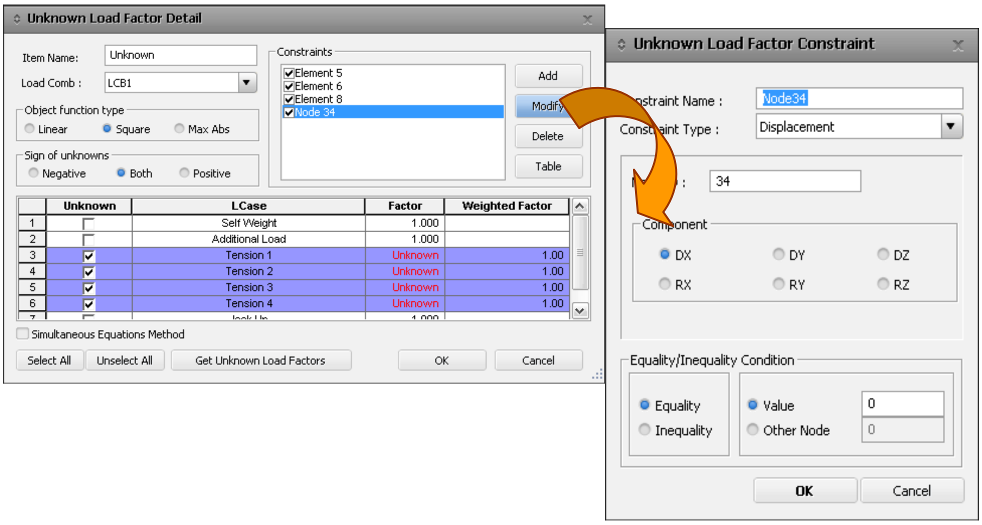


Fig. 26 Constraint Dialog Box

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

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

Unknown Load Factor Group> Get Unknown Load Factors !

Fig. 27 shows the analysis results for unknown load factors calculated by ***Unknown Load Factor***.

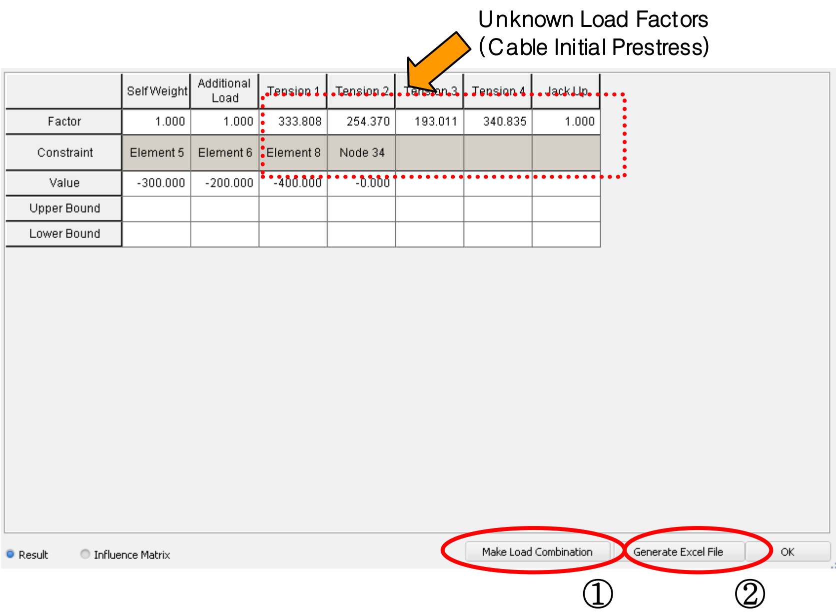


Fig. 27 Analysis Results for Unknown Load Factors

We now check to see if the calculation results satisfy the constraints by auto-generating a new loading combination using the unknown load factors by the ***Make Load Combination*** (① in Fig. 27). The Unknown Load Factors can also be generated by clicking on ***Generate Excel File*** (② in Fig. 27).

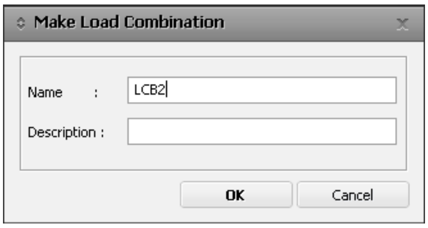


Fig. 28 Auto-generation of LCB2 Using Unknown Load Factors

Results Tab / ***Load Combinatio***

From Tension 1 (ST) to Tension 4 (ST), all the load factors obtained from the analytical results as shown in Fig. 27 are automatically entered.

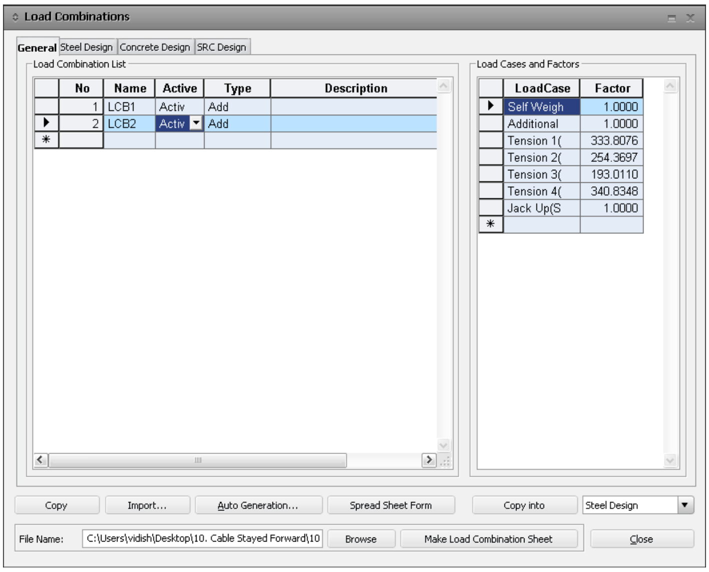


Fig. 29 New Load Combination Auto-generated by Unknown Load Factors

Deformed Shape Review

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

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

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

Components>**DXYZ**

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

Values > Value Output Details

Decimal Points > **(4)** ; **MinMax Only** (on) ; **Min & Max** (on) ↵

Deform > Deformation Scale Factor **(0.5)** ↵

Zoom Window

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

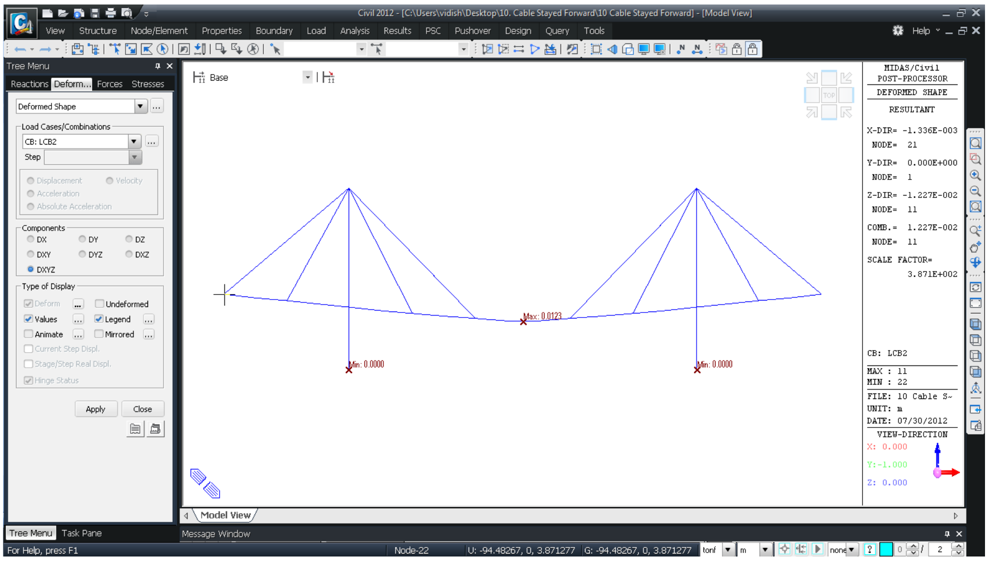


Fig. 30 Check Deformed Shape

Forward Construction Stage Analysis

When a cable stayed bridge is designed, the structural configuration, cable sections and tension forces are generally calculated from the overall analysis of the completed state.

Apart from the analysis for the completed state, construction stage analysis is also required for design of the cable stayed bridge. Depending on the temporary support method, the structural system of a cable stayed bridge changes drastically during construction. The structural system may become unsafe and/or unstable during the construction compared to the completed state. This necessitates a construction stage analysis, and the analysis based on the construction sequence is referred to as Forward Analysis. Stresses, deflection, sequence, constructability, etc. can be checked through Forward Analysis. One of the difficulties associated with forward stage analysis is to find tension forces at construction stages. With the facility of the lack of fit force functionality, additional pretension loads, which are introduced during the installation of cables, are calculated, and member forces are preloaded at Key Segment such that member forces at key segment closure are the same as those at the completed state. Using these pretension and member forces, forward stage analyses are performed.

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.

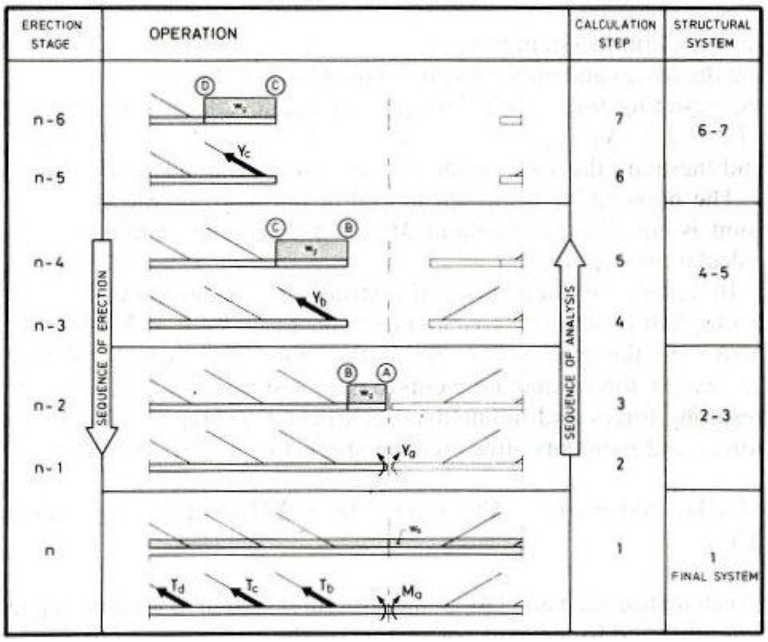
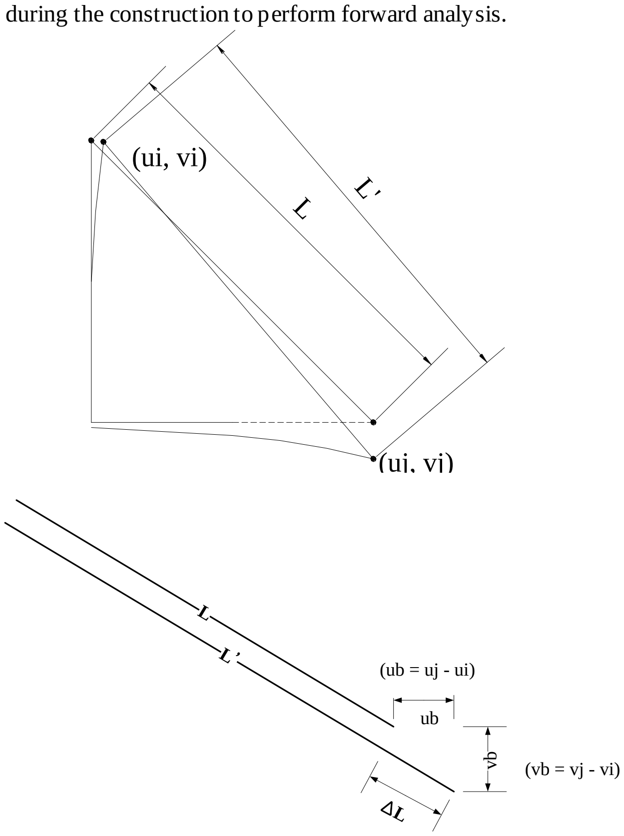


Fig. 31Construction Sequence and Analysis Sequence for a Cable Stayed Bridge

(1) Calculating Lack of Fit Force - Truss

First, displacements at each end of cables are calculated at a stage immediately before the cables are installed. Using the displacements at each cable end, the program calculates the additional cable pretension (ΔT), the difference between the cable length (L) at the completed state and the cable length (L') during the construction. This additional cable pretension (ΔT) is added to the initial Pretension (T) determined from the initial configuration analysis; that is, it is entered as Pretension during the construction to perform forward analysis.



$$L' - L = \Delta L = Vb\,Cos\theta + Ub\,Sin\theta$$

$$\Delta T = \frac{EA}{L}\Delta L$$

$$T_f = T_i + \Delta T$$

Fig. 32 Calculating Lack of Fit Force of Truss

(2) Calculating Lack of Fit Force – Beam

At the time of key segment closure for a 3-span continuous cable stayed bridge, cantilevers of the center span are deflected. If the key segment is closed in this state, no member force takes place at the key segment (only member forces due to self-weight take place) and there is discontinuity between the cantilevers and the key segment. To connect the key segment to each cantilever member continuously, Lack-of-Fit Force function calculates specified displacements required at each end of the key segment and converts the specified displacements into member forces to apply these forces to the key segment.

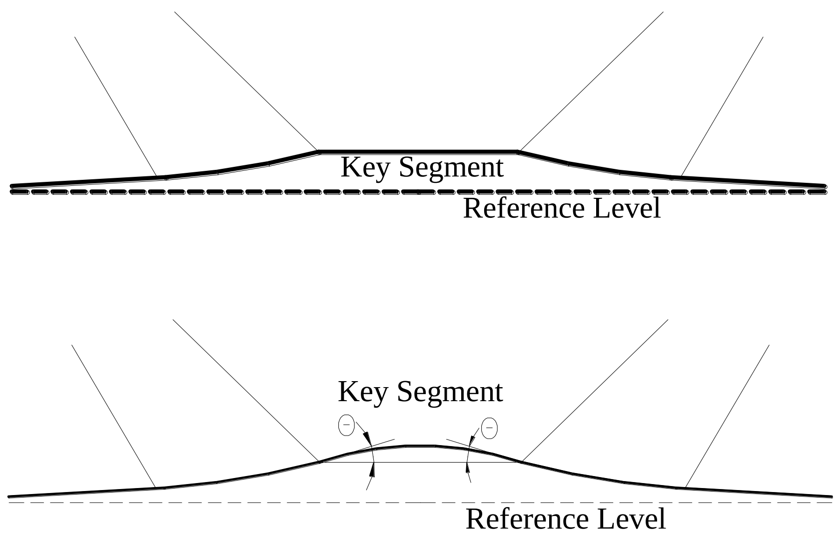


Fig. 33 Calculating Lack of Fit Force of Beam

Construction Stage Category

In this tutorial, 13 construction stages are generated to simulate the changes of loading and boundary conditions.

Forward analysis is performed using Cable Pretension obtained from the initial equilibrium state analysis. Lack-of-Fit Force function is applied to cables, key segment in the center span and side span girders activated at Stage 2. We apply Lack-of-Fit Force function to a side span girder erection stage where girders are connected to the supports and accordingly structural system changes, not to mention the key segment closure in the center span.

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

Table 5 Construction Stage Category

Stage	Content	Remark
Stage 1	Install towers, end supports in the side spans, temporary bents, and temporary connection between tower-girder	
Stage 2	Install side spans (Elements 1 to 5 & 16 to 20)	Lack-of-Fit Force
Stage 3	Apply Derrick Crane1 load	
Stage 4	Remove temporary bents and generate cables (Element 34, 39)	Lack-of-Fit Force
Stage 5	Generate main girders (Element 6, 7, 14, 15)	
Stage 6	Generate cables (Element 35, 38)	Lack-of-Fit Force
Stage 7	Remove Derrick Crane1 load and apply Derrick Crane2 load	
Stage 7-1	Generate cables (Element 33, 40)	Lack-of-Fit Force
Stage 8	Generate main girders (Element 8, 9, 12, 13)	
Stage 9	Generate cables (Element 36, 37)	Lack-of-Fit Force
Stage 10	Remove Derrick Crane2 load and apply Derrick Crane3 load	
Stage 11	Remove Derrick Crane3 load	
Stage 11-1	Generate KEY SEG (Element 10, 11)	Lack-of-Fit Force
Stage 12	Replace the connection between tower-girder and apply Jack Up load	Rigid link→ Elastic link
Stage 13	Apply additional dead loads (Final Stage)	

Forward Construction Stage Analysis

Forward analysis reflects 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 forward construction stage analysis is as shown in Fig. 34.

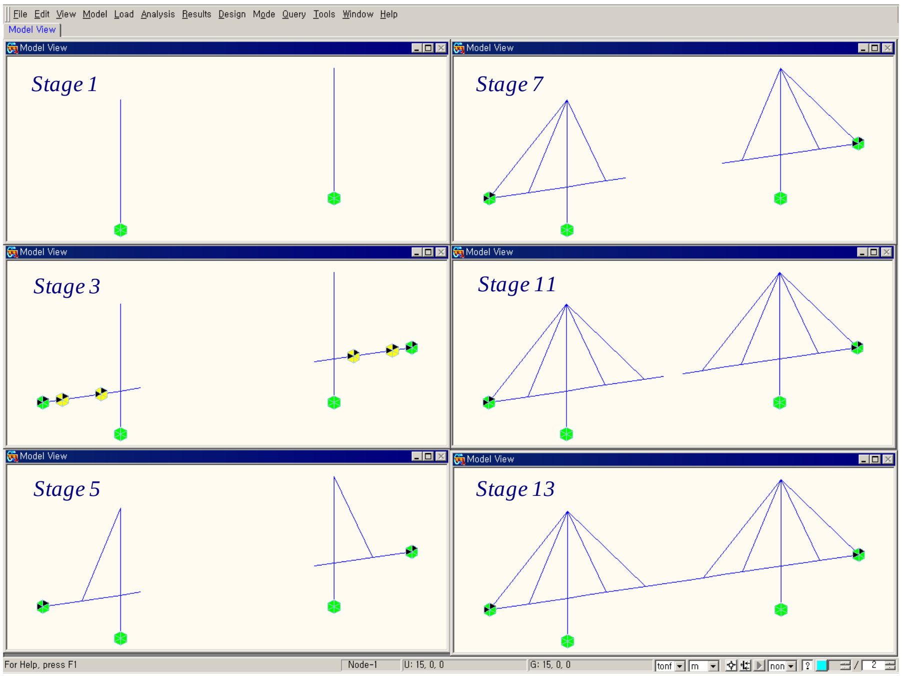


Fig. 34 Analysis Sequence by Forward Construction Stage Analysis

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.

> *Save As* **(Cable Stayed Forward Construction)**

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

1. **Save a forward stage analytical model**
   Change the truss element used in the final stage analysis to cable element.
   Define load cases for forward analysis.

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 and to which Lack-of-Fit Force is applied.

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.

Save a Forward Stage Analytical Model

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 4. To input Pretension Loads calculated by forward stage analysis, define a new loading case for Cable Pretension.

Results Tab > ***Load Combination***

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

Load Tab > ***Static Load Cases***

Name **(Tension 1)** ~ Name **(Tension 4)** Delete

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

Description **(Pretension from Forward Analysis)** ↵

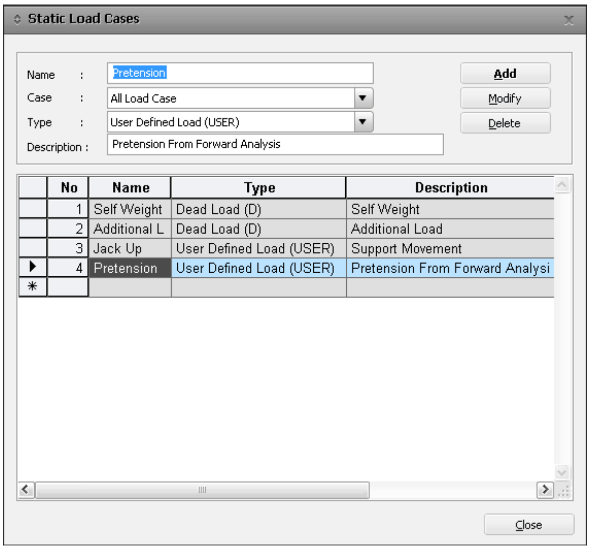


Fig. 35 Entering Initial Prestress Loading Condition

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.

Node/ Element Tab > ***Change Elements Parameters***

Select identity - Elements

Select Type>**Element Type**

Nodes (off) ; **Elements** (on)

(Truss) > Add ↵

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

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

Cable (on) ↵

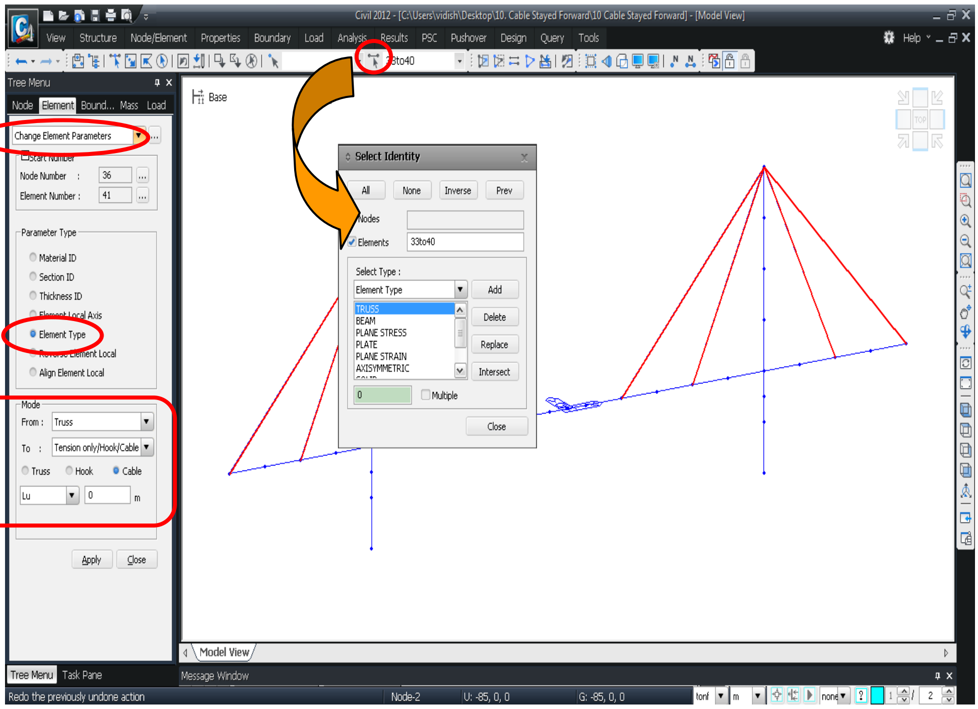


Fig. 36 Change of Truss Element to Cable Element

Define Construction Stage

We now define each construction stage to perform forward construction stage analysis. First, we assign each construction stage name in the Construction Stage dialog box. In this tutorial, we will define total 13 construction stages including the final stage.

Define multiple construction stages simultaneously by assigning numbers to a particular name.

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

Load Tab > Construction Stage Option > ***Define Construction Stage***

Generate

Stage>Name **(Stage)**; Suffix **(1to7)**

Save Result>**Stage** (on) OK ↵

Generate

Stage>Name **(Stage7-1)**

Save Result>**Stage** (on) OK ↵

Generate

Stage>Name **(Stage)**; Suffix **(8to11)**

Save Result>**Stage** (on) OK ↵

Generate

Stage>Name **(Stage11-1)**

Save Result>**Stage** (on) OK ↵

Generate

Stage>Name **(Stage)**; Suffix **(12to13)**

Save Result>**Stage** (on) OK ↵

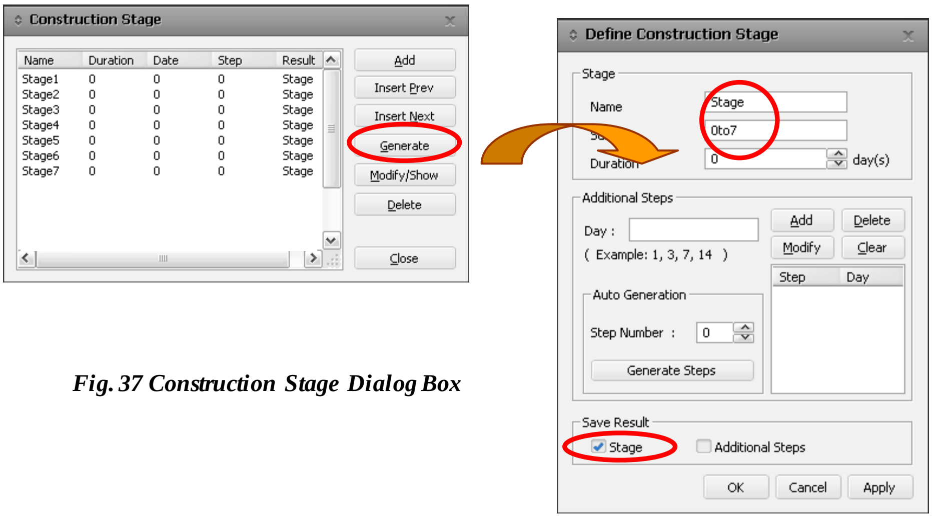


Fig. 37 Construction Stage Dialog Box

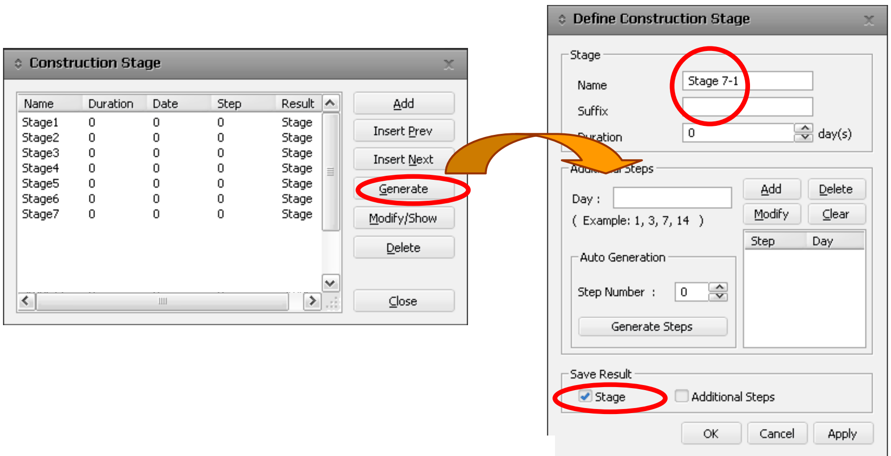


Fig. 38 Construction Stage Dialog Box

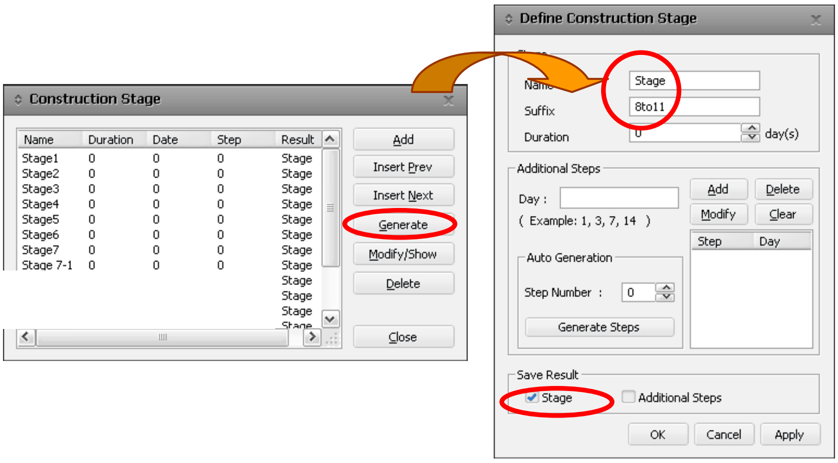


Fig. 39 Construction Stage Dialog Box

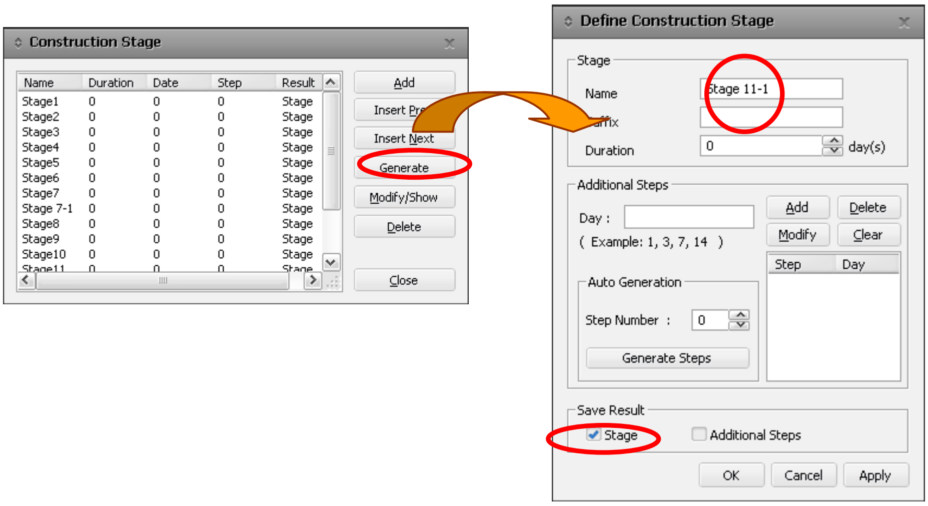


Fig. 40 Construction Stage Dialog Box

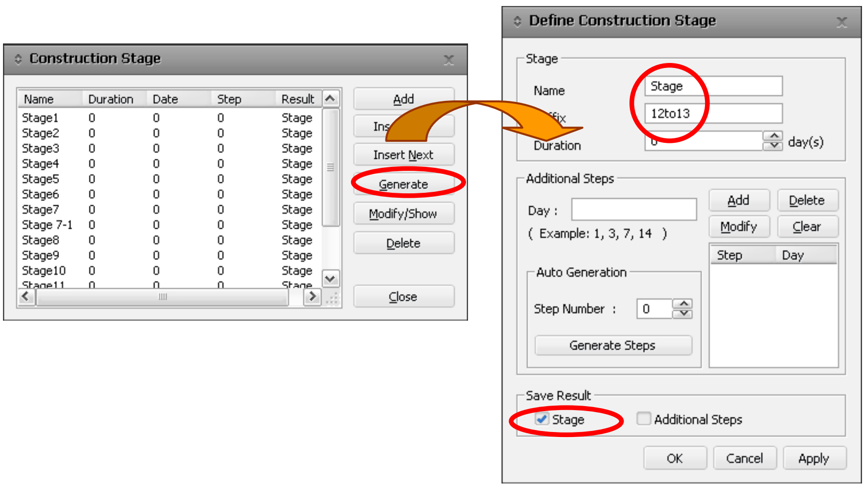


Fig. 41 Construction Stage Dialog Box

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

Group>Structure Group>**New…**

Name **(Stage)**; Suffix **(1to13)** Add

Name **(Lack of Fit Force)** Add

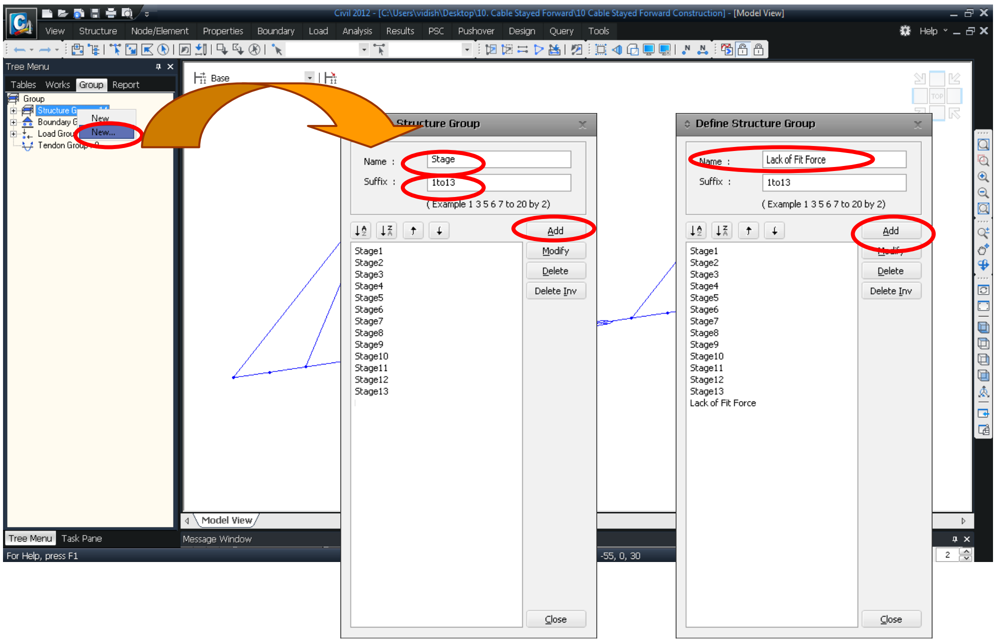


Fig. 42 Defining Structure Group

Assign the elements, which become added/deleted in each construction stage, to each corresponding Structure Group. The tower erection stage is defined as the Stage 1 Structure Group. We skip Stage 3 and Stage 10 because they are construction stages, in which Derrick Crane load at the center span is applied, and as such there are no added/deleted elements involved.

Group > Structure Group

Select Window (① in Fig. 43)

Stage 1 (***Drag & Drop***)

Select Window (② in Fig. 43)

Stage 2 (***Drag & Drop***)

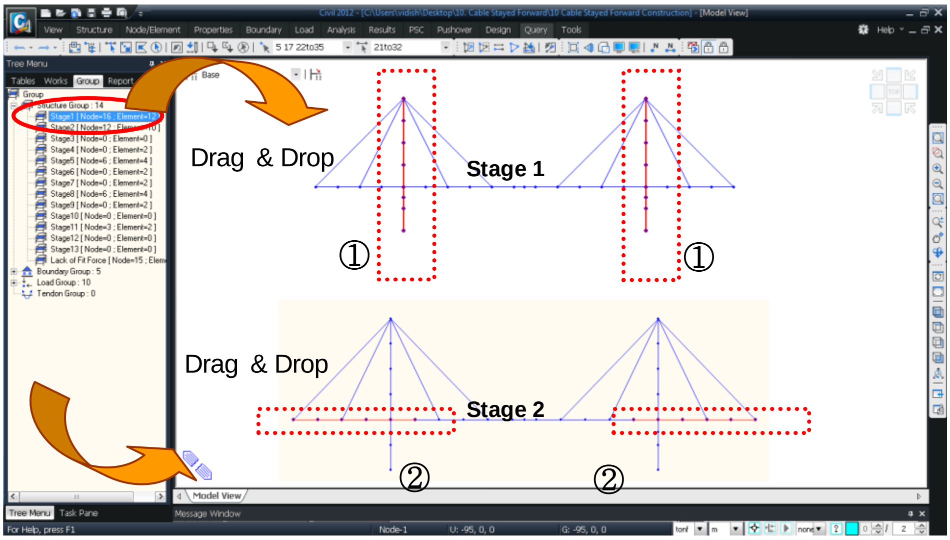


Fig. 43 Defining Structure Group Stage 1 ~ Stage 2

Group > Structure Group

Select Intersect (① in Fig. 44)

Stage 4 (***Drag & Drop***)

Select Window (② in Fig. 44)

Stage 5 (***Drag & Drop***)

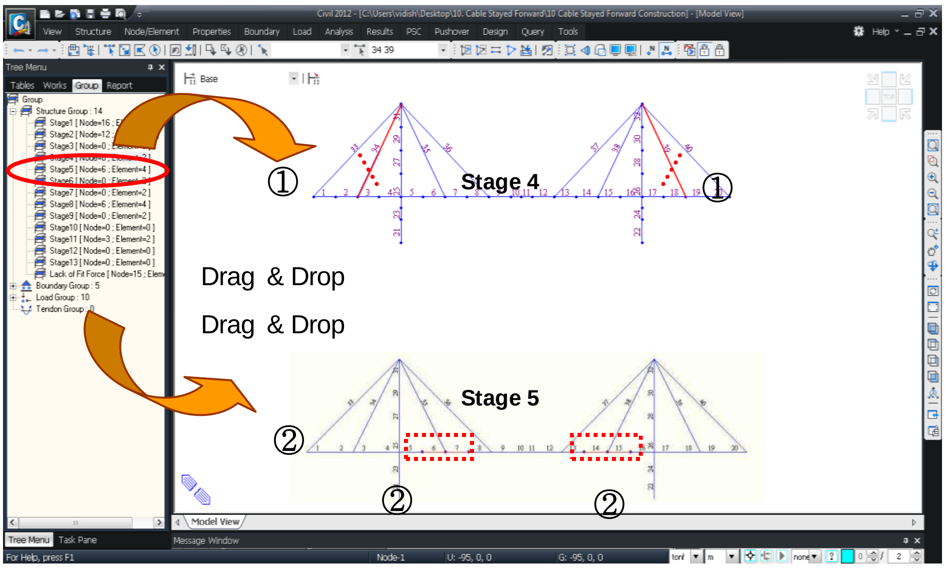


Fig. 44 Defining Structure Group Stage 4~ Stage 5

Group > Structure Group

Select Intersect (① in Fig. 45)

Stage 6 (***Drag & Drop***)

Select Intersect (② in Fig. 45)

Stage 7 (***Drag & Drop***)

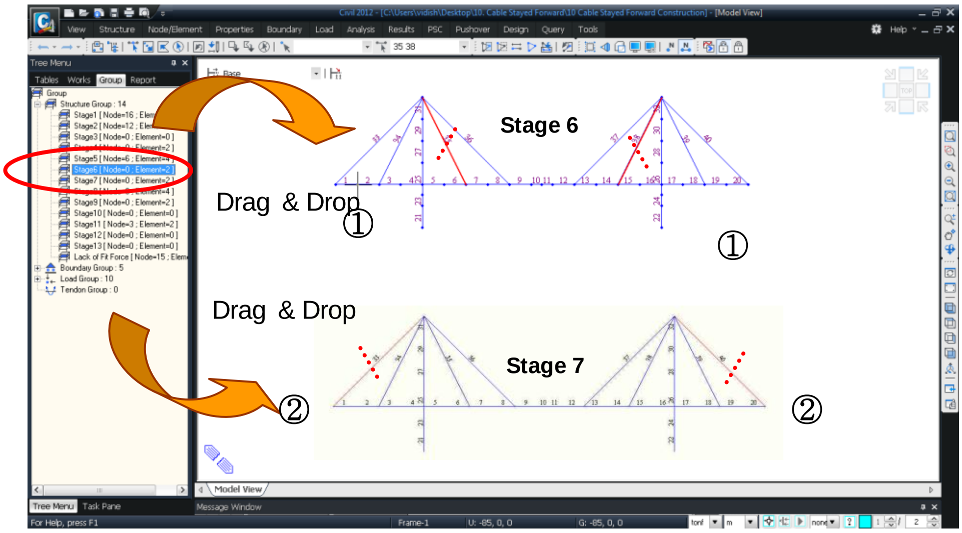


Fig. 45 Defining Structure Group Stage 6~ Stage 7

Group > Structure Group

Select Window (① in Fig. 46)

Stage 8 (***Drag & Drop***)

Select Intersect (② in Fig. 46)

Stage 9 (***Drag & Drop***)

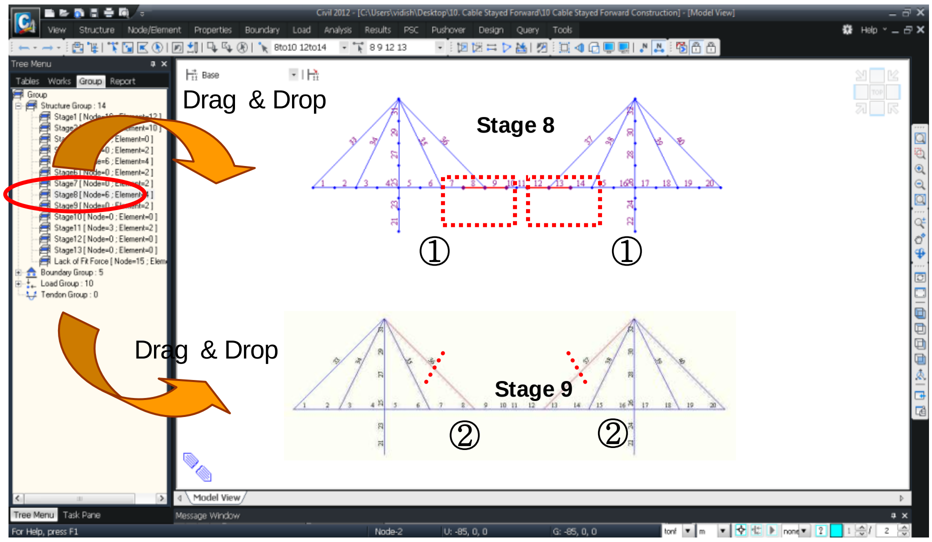


Fig. 46 Defining Structure Group Stage 8~ Stage 9

Assign the Structure Group, which is required to define the stage (Stage 11) to which key segment is added in forward construction stage analysis.

Stage 11 is the stage in which key segment is installed and at the same time the center span is closed.

Select Window (① in Fig. 45)

Stage 11 (***Drag & Drop***)

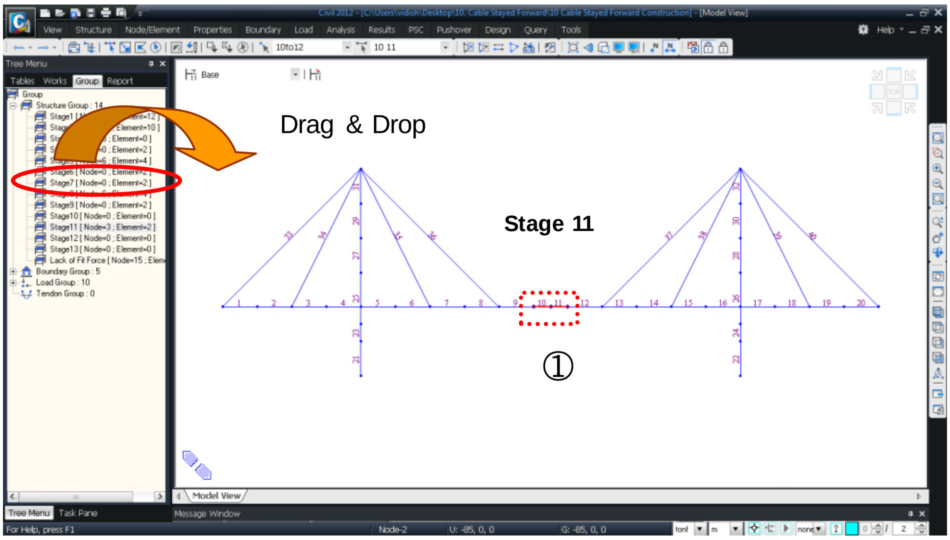


Fig. 47 Defining Structure Group Stage 11

Group > Structure Group

Select Node : 1to6 10to12 16to21 **(**① in Fig. 48 **)**

Select Element : 1to5 10 11 16to20 33to36 37to40 **(**① in Fig. 48**)**

Lack of Fit Force (***Drag & Drop***)

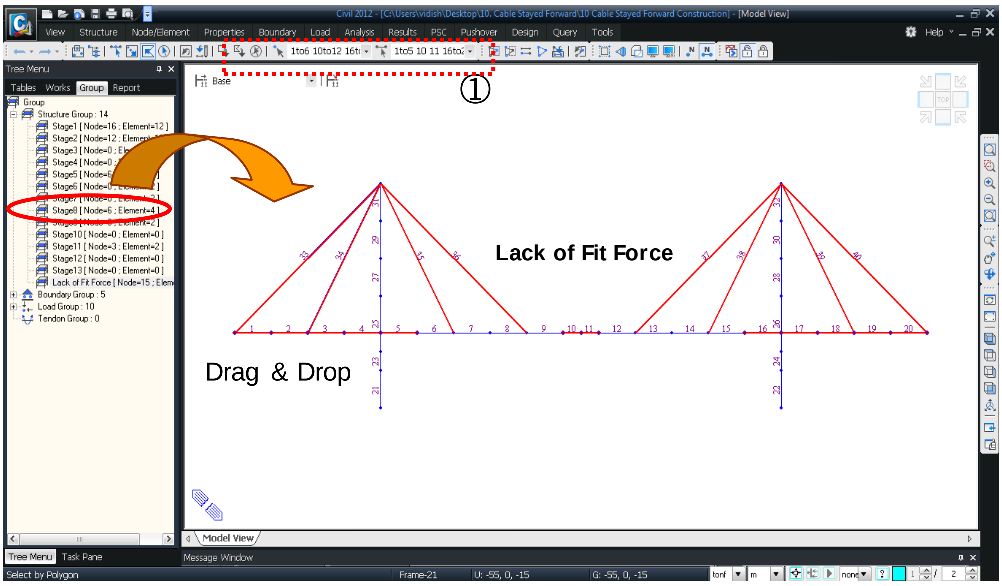


Fig. 48 Defining Structure Group Lack of Fit Force

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.

Active All

Group tab

Group>Boundary Group>**New…**

Name **(Fixed Support (Tower))** ↵

Name **(Hinge Support (Pier))** ↵

Name **(Elastic Link (Tower))** ↵

Name **(Temporary Support (Tower))** ↵

Name **(Temporary Bent)** ↵

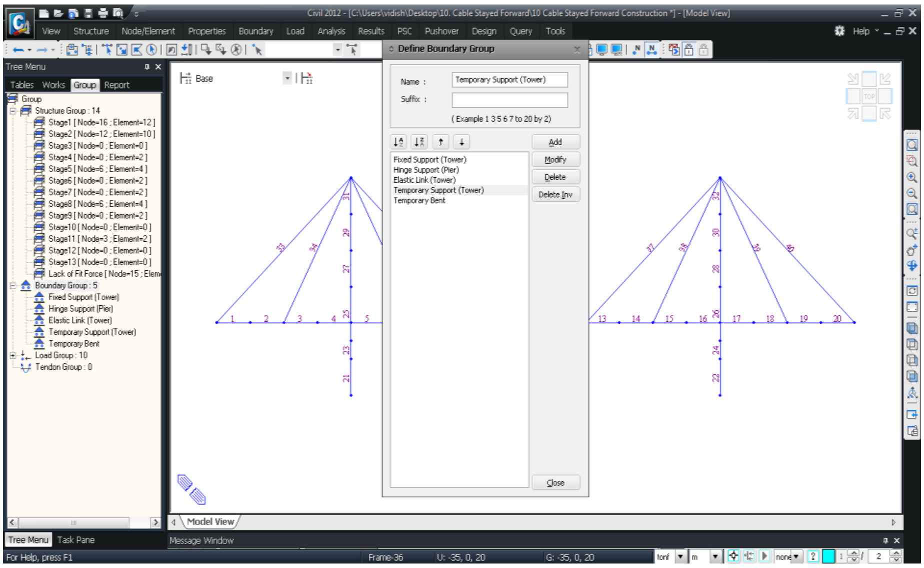


Fig. 49 Defining Boundary Group

Reassign the Fixed Support (Tower) and Hinge Support (Pier) conditions, which were already defined for the final stage analysis, to Boundary Group for the construction stage analysis.

Group>Boundary Group

Select Window (① in Fig. 50)

Fixed Support (Tower) (***Drag & Drop***)

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

Select Window (② in Fig. 50)

Hinge Support (Pier) (***Drag & Drop***)

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

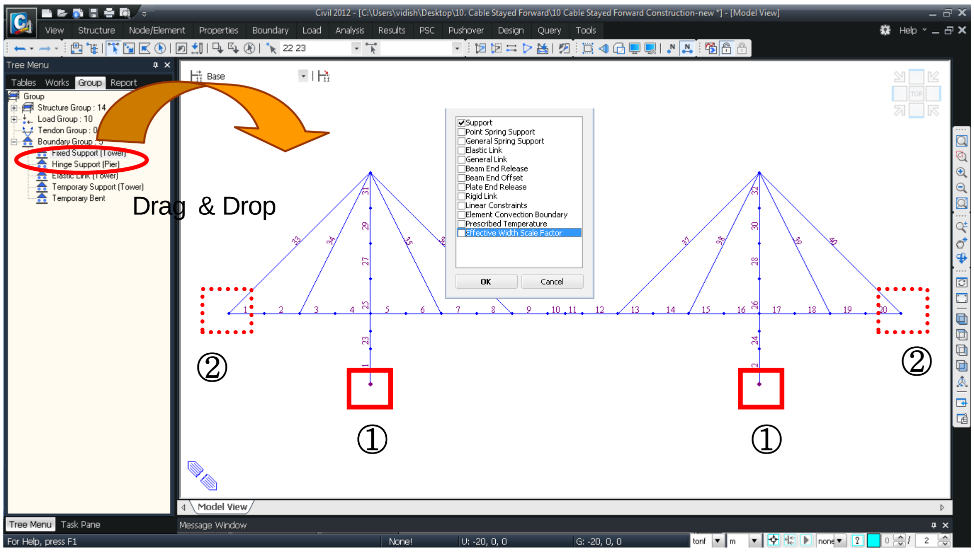


Fig. 50 Generating Fixed Support (Tower) and Hinge Support (Pier) Conditions

We also reassign the boundary condition for the Elastic Link (Tower) to a Boundary Group. We will input the boundary condition as ***Elastic Link-General Type*** between the tower and the girder. The stiffness is as follows:

SDx : **500,000** tonf/m, SDy : **10,000,000** tonf/m, SDz : **10,000,000** tonf/m

SRx : **0** tonf/m, SRy : **0** tonf/m, SRz : **0** tonf/m

Boundary Tab > ***Elastic Link***

Boundary Group Name > **Elastic Link (Tower)**

Options > **Add** ; Link Type > **General Type**

SDx (tonf/m) **(500000)** ; SDy(tonf/m) **(10000000)** ; SDz(tonf/m) **(10000000)**

SRx (tonf/m) **(0)** ; SRy(tonf/m) **(0)** ; SRz(tonf/m) **(0)** ↵

2 Nodes **(26, 5)**

2 Nodes **(27, 17)** ↵

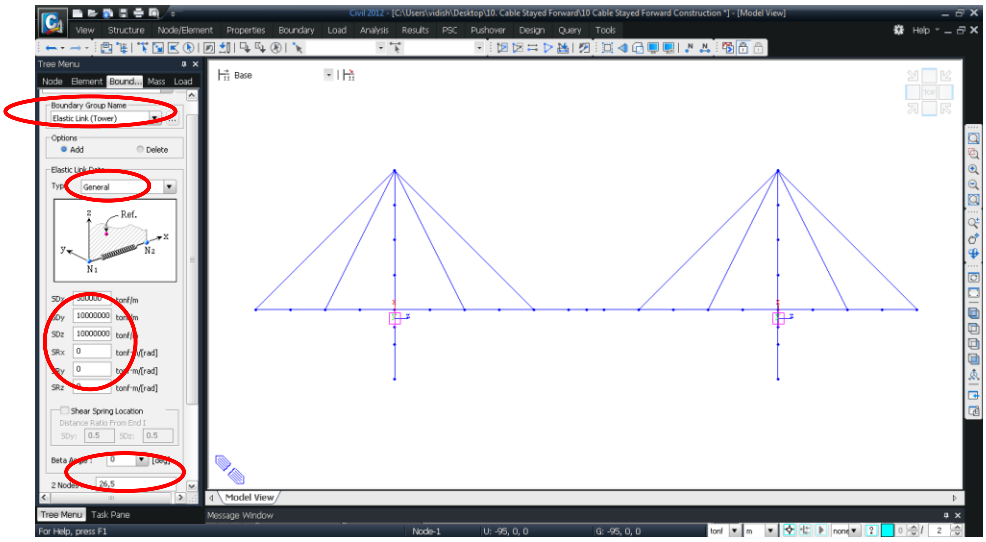


Fig. 51 Generating Boundary Condition for Elastic Link (Tower)

We also reassign the boundary condition for the Temporary Support (Tower) to a Boundary Group. We will input the boundary condition as ***Elastic Link- Rigid Type*** between the tower and the girder.

Boundary Tab > ***Elastic Link***

Boundary Group Name > **Temporary Support (Tower)**

Options > **Add** ; Link Type > **Rigid Type**

2 Nodes **(26, 5)**

2 Nodes **(27, 17)** ↲

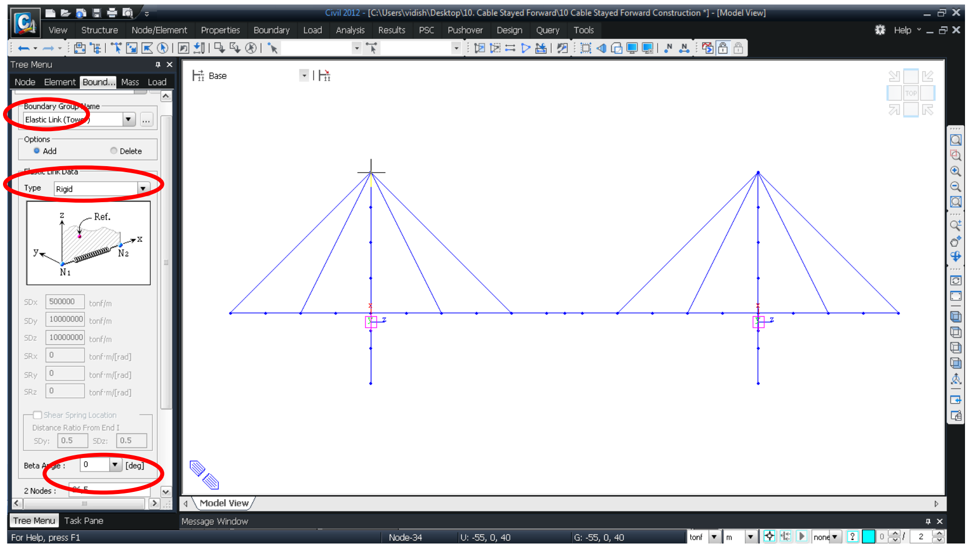


Fig. 52 Generating Boundary Condition for Temporary Support (Tower)

We also assign the boundary condition for the temporary bents to a Boundary Group. We will input the boundary condition as ***Point Spring Support***. The stiffness is as follows:

SDx : **0** tonf/m, SDy : **10,000,000** tonf/m, SDz : **10,000,000** tonf/m
SRx : **10,000,000** tonf/m, SRy : **0** tonf/m, SRz : **10,000,000** tonf/m

---

Boundary Tab > ***Point Spring***

Select Window (① in Fig. 53; **Node 2, 4, 18, 20**)

Boundary Group Name>**Temporary Bent**

Options>**Add**

SDx (tonf/m) **(0)** ; SDy(tonf/m) **(10000000)** ; SDz(tonf/m) **(10000000)**

SRx (tonf/m) **(10000000)** ; SRy(tonf/m) **(0)** ; SRz(tonf/m) **(10000000)** ↵

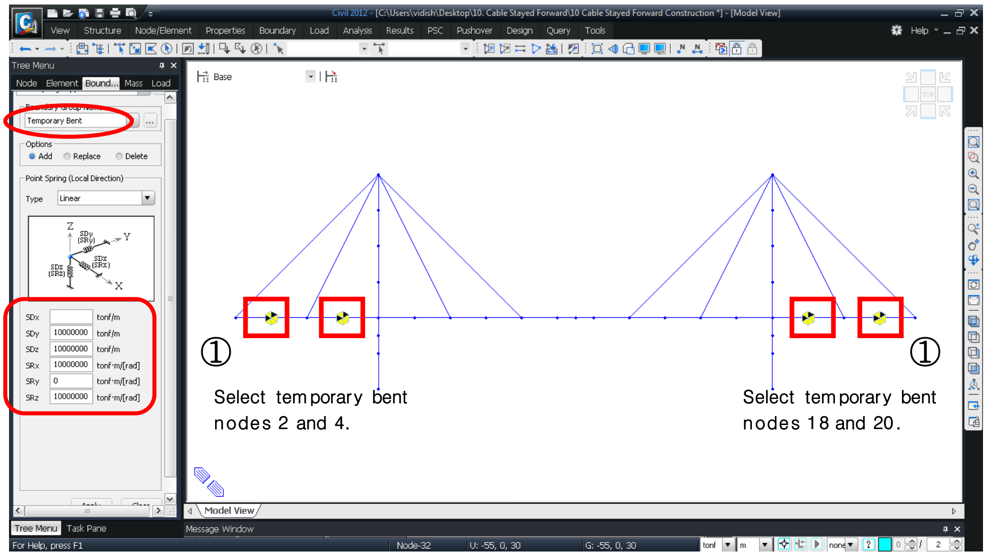


Fig. 53 Generating Boundary Condition for Temporary Bents

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 forward construction stage analysis are self-weight, superimposed dead load, cable pretension 1, 2, 3, 4, Jack Up load and Derrick Crane load 1, 2, 3. First, we generate the name of each Load Group and then assign corresponding loading conditions to each Load Group.

Group tab

Group>Load Group> **New…**

Name (**Self Weight**) ↵

Name (**Additional Load**) ↵

Name (**Pretension Load 1**) ↵

Name (**Pretension Load 2**) ↵

Name (**Pretension Load 3**) ↵

Name (**Pretension Load 4**) ↵

Name (**Jackup Load**) ↵

Name (**Derrick Crane 1**) ↵

Name (**Derrick Crane 2**) ↵

Name (**Derrick Crane 3**) ↵

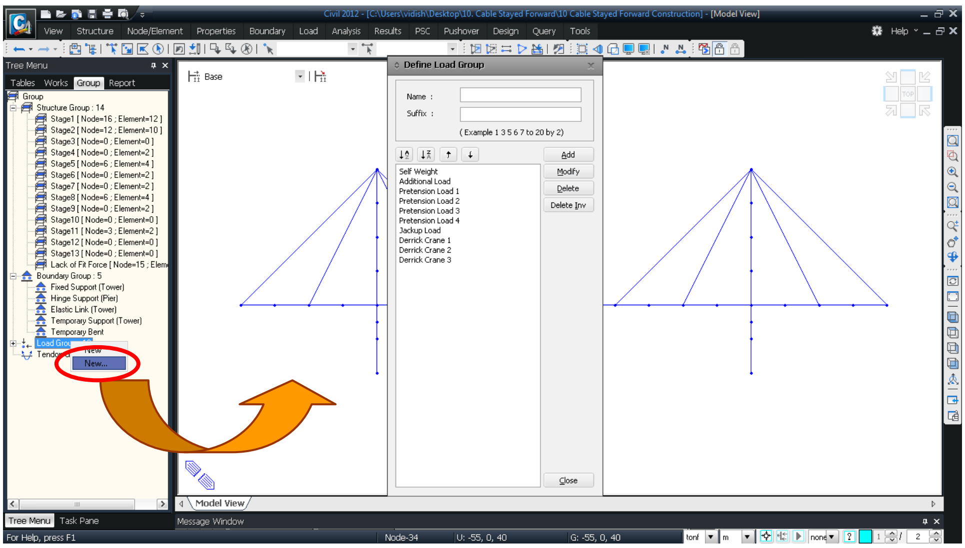


Fig. 54 Defining Load Group

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

Load Tab > ***Self Weight***

Load Case Name>**Self Weight**

Load Group Name>**Self Weight**

Operation>Modify

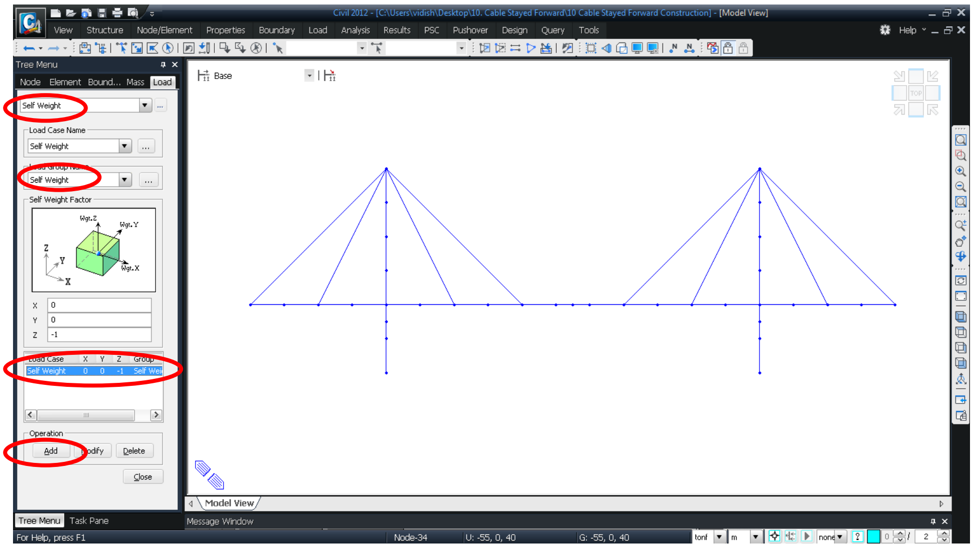


Fig. 55 Modifying Load Group for Self-Weight

Reassign the superimposed dead load and Jack Up load, which were defined for the final stage analysis, to Load Group.

Group > Load Group
  Select All
  Additional Load (***Drag & Drop***)
  Select Load Type>**Beam Loads** (on) ↵
  Select All
  Jack Up Load (***Drag & Drop***)
  Select Load Type>**Specified Displacements of Supports** (on) ↵

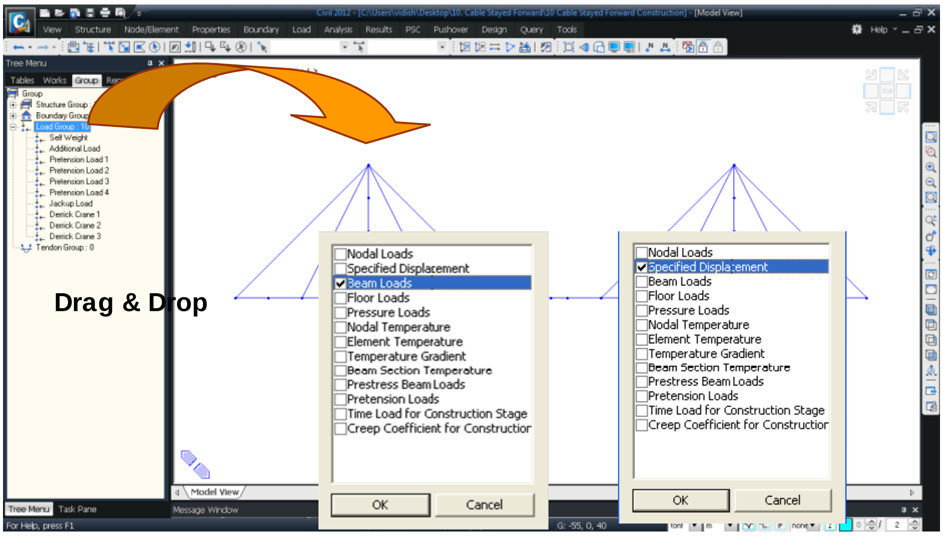


Fig. 56 Defining Load Group for Superimposed Dead Load and Jack Up Load

Input Derrick Crane loads. Derrick Crane shifts its position according to construction sequence. Apply Derrick Crane 1, 2, 3 loads to the corresponding positions. In order to input Derrick Crane loads, define a new loading case for Derrick Crane.

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

Name (**Derrick Crane**) ; Type > **Construction Stage Load**

Description (**Derrick Crane Load**) ↵

Load Tab / ***Nodal Loads***

Select Window (① in Fig. 58; **Node 6, 16**)

Load Case Name>**Derrick Crane**

Load Group Name> **Derrick Crane 1** ; Options>**Add**

Nodal Loads>FZ **(-80)** ↵

Select Window (② in Fig. 58; **Node 8, 14**)

Load Case Name> **Derrick Crane**

Load Group Name> **Derrick Crane 2** ; Options>**Add**

Nodal Loads>FZ **(-80)** ↵

Select Window (③ in Fig. 58; **Node 10, 12**)

Load Case Name> **Derrick Crane**

Load Group Name> **Derrick Crane 3** ; Options>**Add**

Nodal Loads>FZ **(-80)** ↵

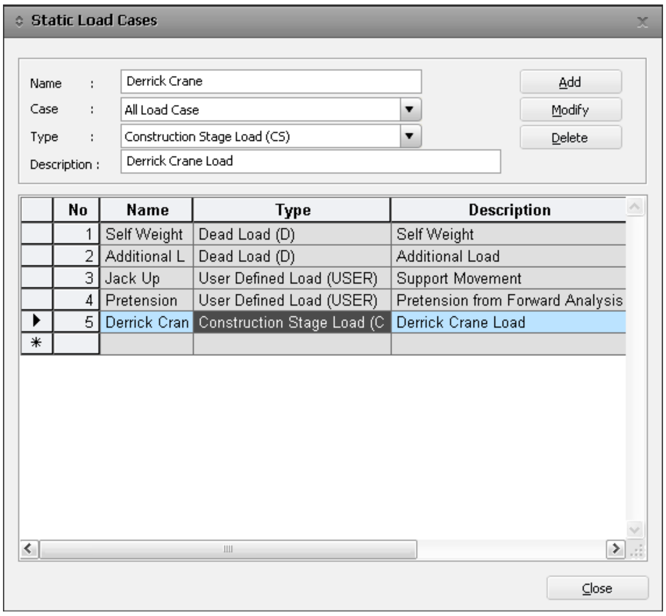


Fig. 57 Generation of a Loading Condition for Derrick Crane

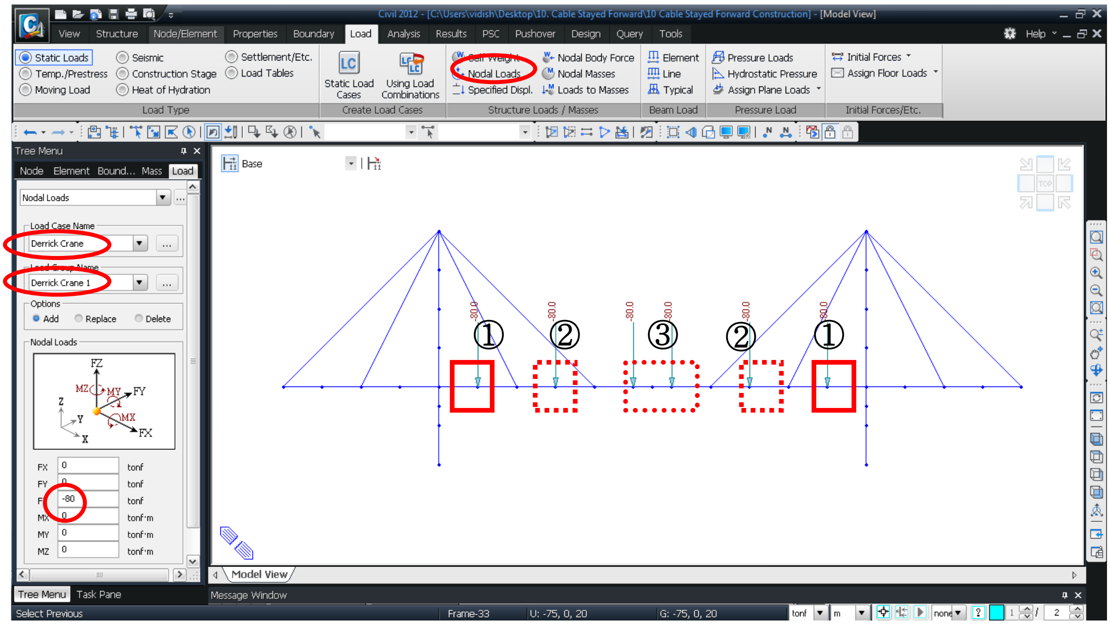


Fig. 58 Entering Derrick Crane Loads

Input Cable Pretension calculated by the final stage analysis to individual cable elements as *Pretension Loads*.

Load Tab > Temp./Prestress Option > ***Pretension Loads***

Select Intersect (Elements : ① in Fig. 59; **Element : 36, 37**)

Load Case Name> **Pretension** ; Load Group Name > **Pretension Load** 1

Options > **Add** ; Pretension Load **(340.835)** ↵

…

Load Case Name> **Pretension** ; Load Group Name > **Pretension Load** 4

Options > **Add** ; Pretension Load **(254.370)** ↵

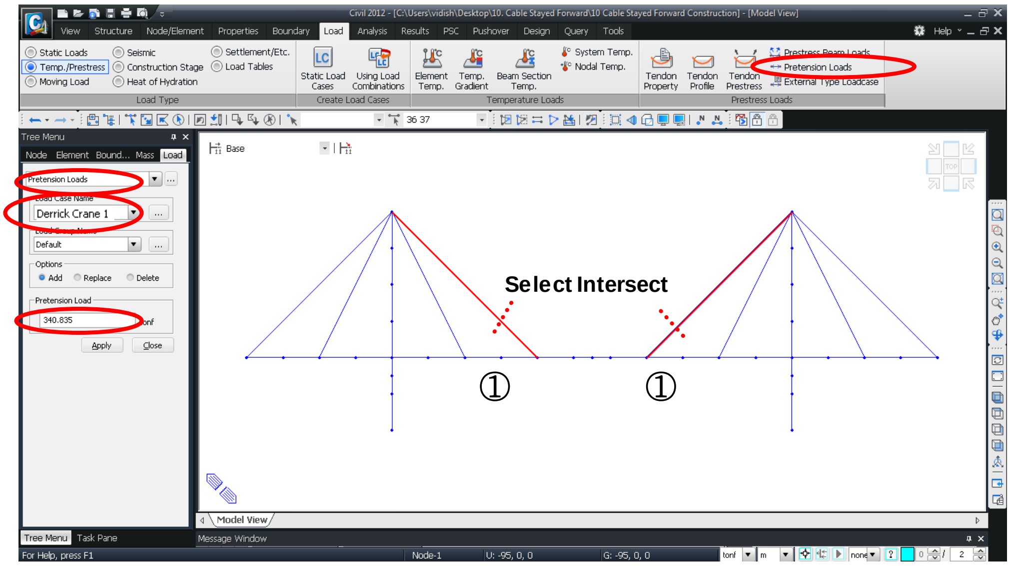


Fig. 59 Entering Cable Pretension obtained from forward analysis

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

Table 6. Cable Pretension (Pretension Loading) calculated by Initial Equilibrium State Analysis

Load Group	Element No.	Pretension Loading	Load Group	Element No.	Pretension Loading
Pretension Load 1	36, 37	340.835	Pretension Load 3	35, 38	193.011
Pretension Load 2	33, 40	333.808	Pretension Load 4	34, 39	254.370

Assign Construction Stage

We now assign the predefined Structure Group, Boundary Group and Load Group to each corresponding construction stage. First, we assign Stage 1 to Construction Stage as the 1st stage in forward analysis. Stage 1 is a construction stage, which installs towers.

Load Tab > Construction Stage Option > *Define C.S.*

Stage 1 Modify/Show

Save Result>**Stage** (on)

Element tab>Group List > **Stage 1**

Activation> Add

Boundary tab>Group List > **Fixed Support (Tower)**

Boundary tab>Group List > **Temporary Support (Tower)**

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

Activation> Add

Load tab> Group List>**Self Weight**

Activation> Add ↵

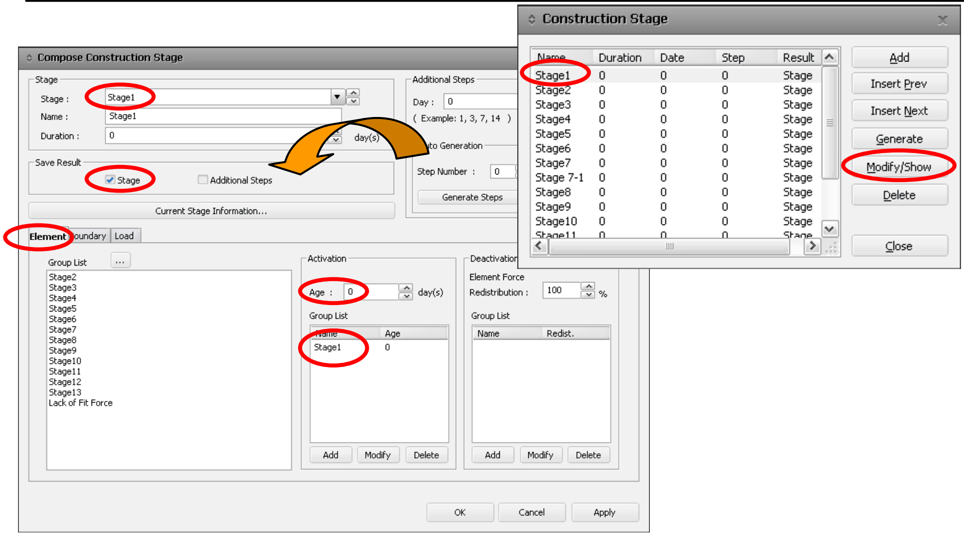


Fig. 60 Defining Elements for Stage 1

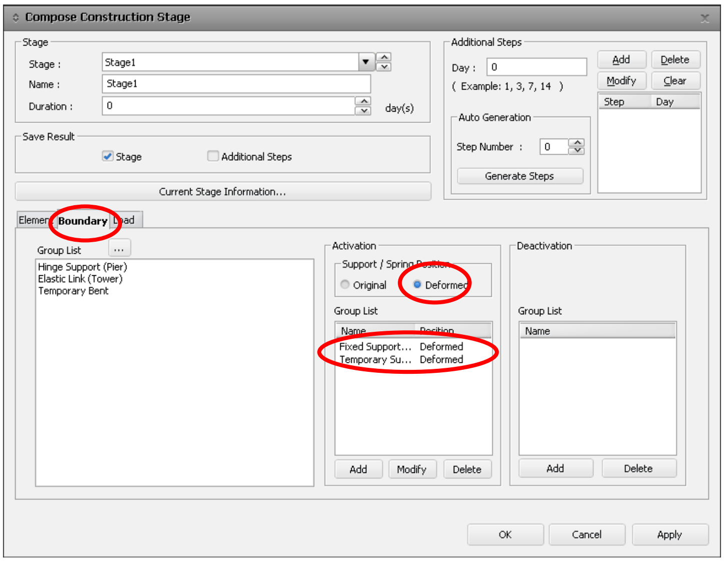


Fig. 61 Defining Boundary Conditions for Stage 1

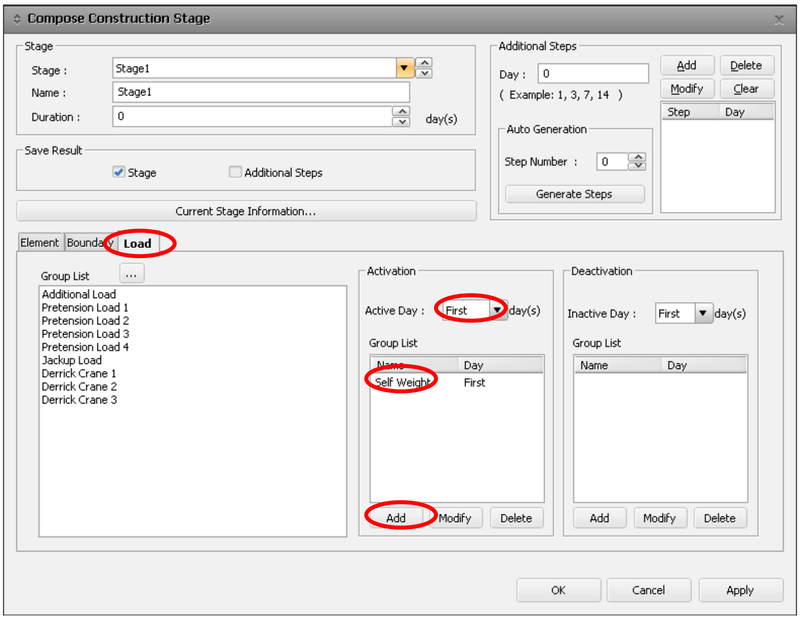


Fig. 62 Defining Loads for Stage 1

Define Construction Stage for each construction stage from Stage 2 to Stage 13 using Table 7 Forward Construction Stage Category.

Table 7. Forward Construction Stage Category

	Structure		**Boundary**		**Load Group**	
	Activation	**Deactivation**	**Activation**	**Deactivation**	**Activation**	**Deactivation**
Stage 1	Stage1		Fixed Support (Tower) (Deformed) Temporary Support… (Deformed)		Self Weight	
Stage 2	Stage2		Hinge Support (Pier) (Deformed) Temporary Bent (Deformed)			
Stage 3					Derrick Crane1	
Stage 4	Stage4			Temporary Bent	Pretension Load 4	
Stage 5	Stage5					
Stage 6	Stage6				Pretension Load 3	
Stage 7					Derrick Crane2	Derrick Crane1
Stage 7-1	Stage7				Pretension Load 2	
Stage 8	Stage8					
Stage 9	Stage9				Pretension Load 1	
Stage 10					Derrick Crane3	Derrick Crane2
Stage 11						Derrick Crane3
Stage 11-1	Stage11					
Stage 12			Elastic Link (Tower) (Deformed)	Temporary Support (Tower)	Jack Up Load	
Stage 13					Additional Load	

Construction Stage Models

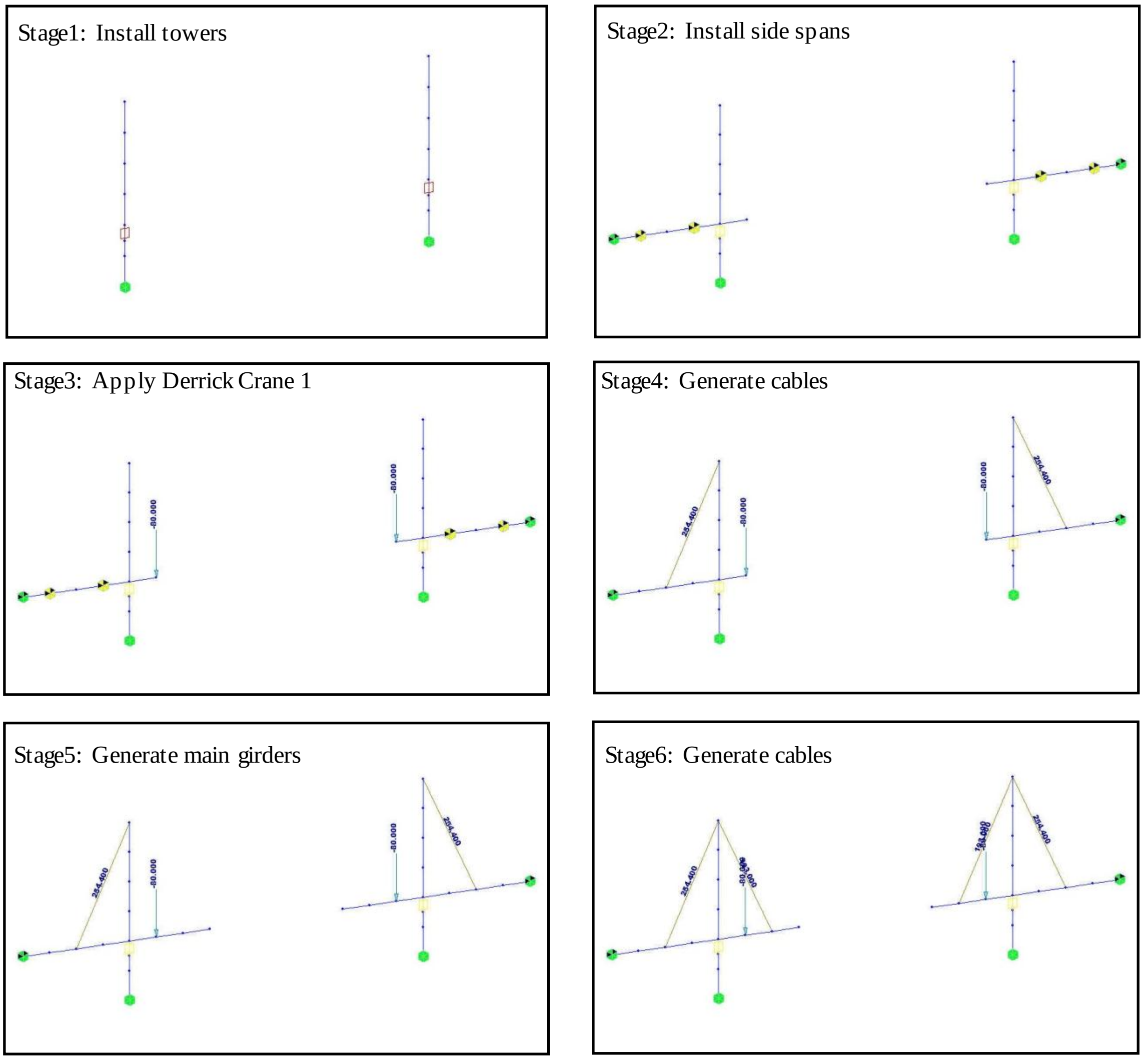


Fig. 63 Modeling Construction Stages (Stage1~Stage6)

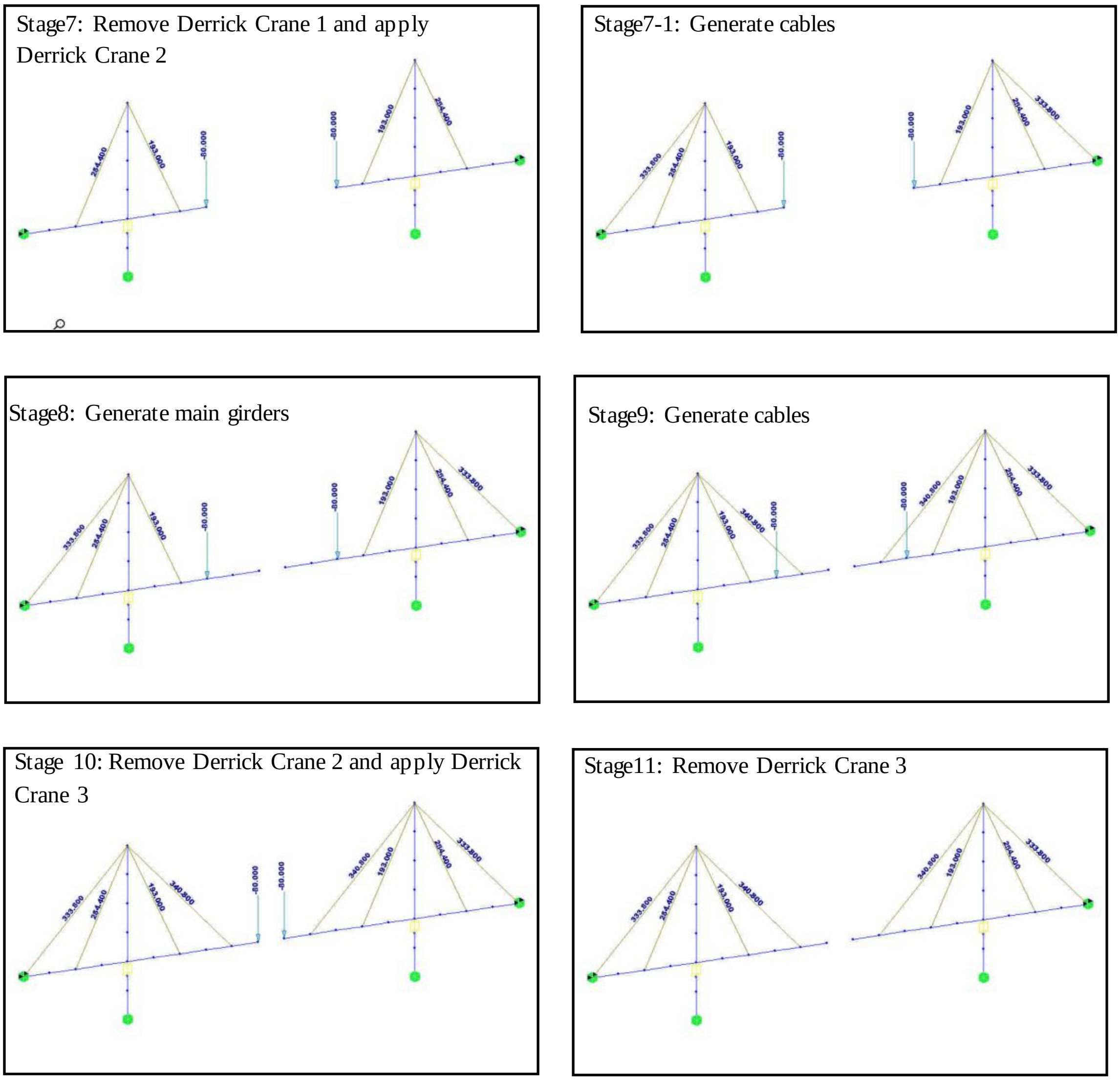


Fig. 64 Modeling Construction Stages (Stage 7~Stage 11)

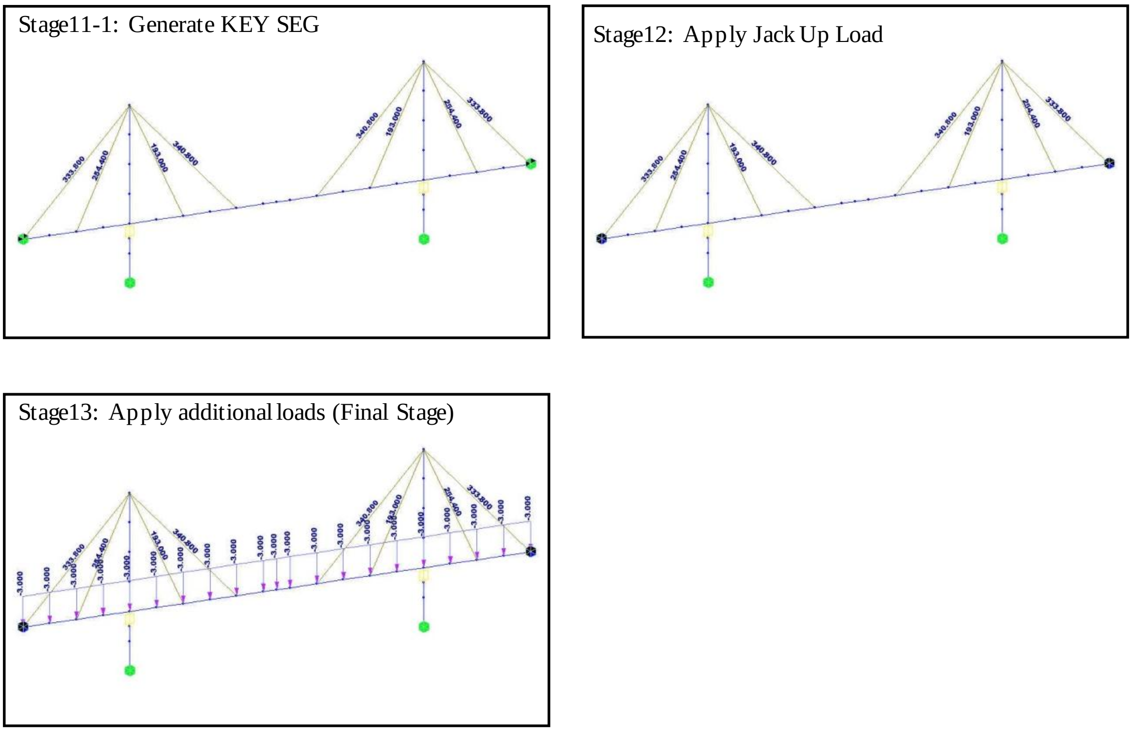


Fig. 65 Modeling Construction Stages (Stage 11-1~Stage 13)

Input Construction Stage Analysis Data

Analysis Tab > ***Construction Stage***

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

Cable-Pretension Force Control > **Internal Force** (on)

Initial Tangent Displacement Erected Structures> **All** (on)

Lack of Fit Force Control (on) > **Lack of Fit Force** (Select) ↵

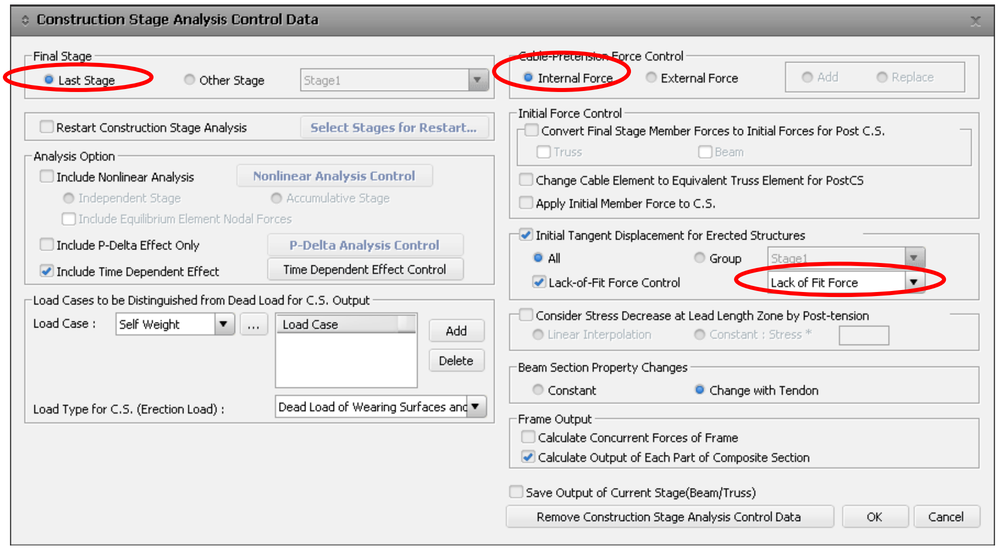


Fig. 65 Construction Stage Analysis Control Data Dialog Box

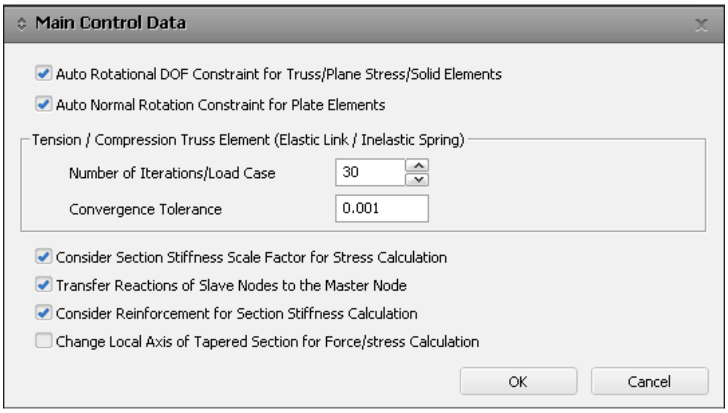
Adjusting Cable-Pretension using Cable-Pretension Force Control and Lack of Fit Force Control

Cable-Pretension Force Control function is used to control Cable-Pretension for construction stage analysis of a cable stayed bridge. In general, due to force redistribution we cannot expect the resultant cable forces whose magnitude is the same as the Pretension Loads entered. External Force type is selected when the user directly enters cable pretension. In this tutorial, since we use pretension obtained from the completed state, we select Internal Force type and Lack-of-Fit Force Control to automatically calculate pretension for each stage.

Main Control Data

For convergence, we change the default for Number of Iterations/Load Case (20) to 30.

Analysis Tab > ***Main Control Data***
Tension / Compression Truss Element (Elastic Link / Inelastic Spring)
Number of Iterations/Load Case > **30**↵



Perform Structural Analysis

Perform construction stage analysis for self-weight, superimposed dead load, Jack Up load and Derrick Crane load.

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.

Stage Tab **Stage 6**

Result Tab > Deformations > ***Deformed Shape***

Load Cases/Combinations>**CS:Dead Load**

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

Deform > Deformation Scale Factor **(0.5)** ↵

Values > Value Output Details

Decimal Points > **(4)** ; **MinMax Only** (on) ; **Min & Max** (on)

Stage/Step Real Displ. (on)

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

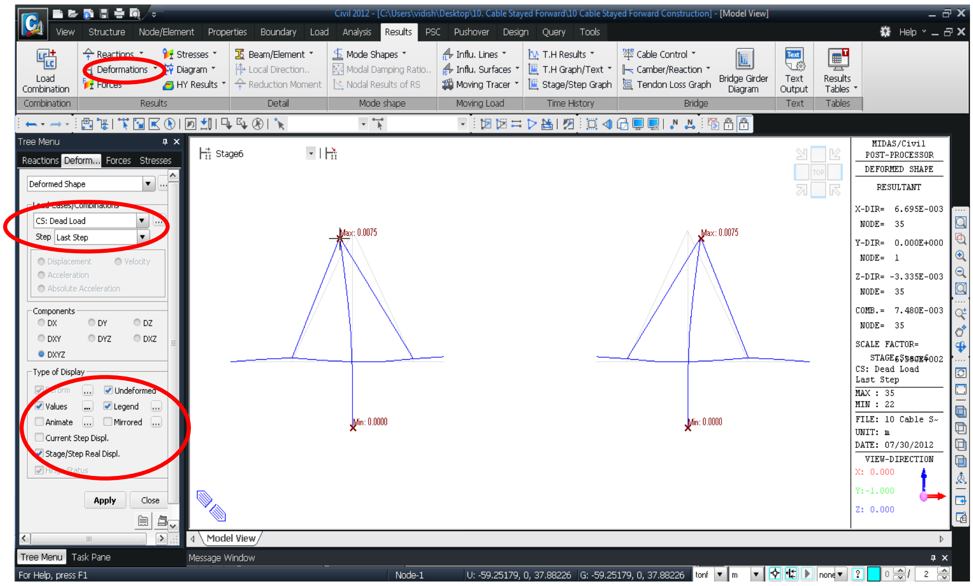


Fig. 66 Deformed Shape for Each Construction Stage from Forward Analysis

Review Bending Moments

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

Result Tab > Forces > ***Beam Diagrams***

Stage Toolbar>**Stage 10**

Load Cases>Combinations>**CS:Summation**

Components>**My**

Display Options>**5Points**; **Solid Fill**

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

Values > Value Output Details

Decimal Points > **(3)** ; **MinMax Only** (on) ; **Min & Max** (on)

↵

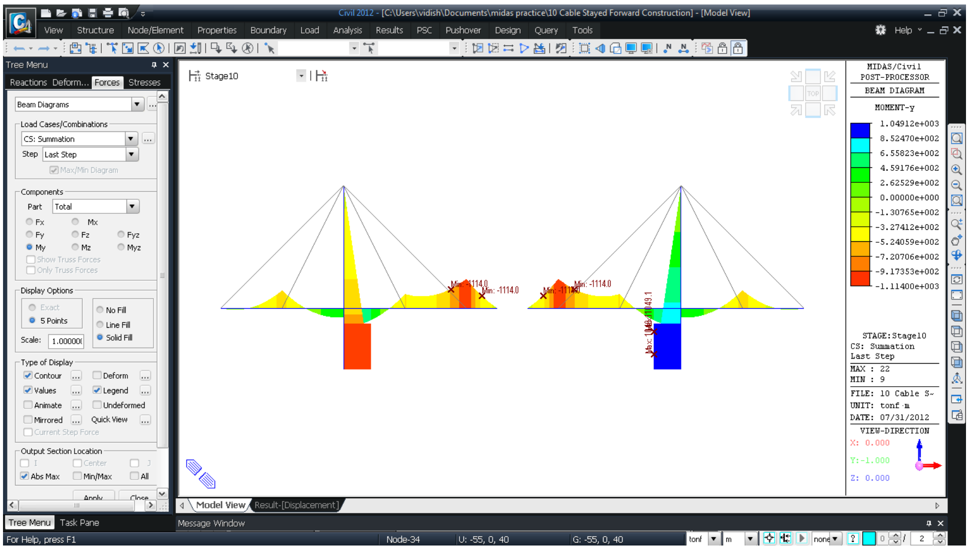


Fig. 67 Bending Moment Diagram for Each Construction Stage from Forward Analysis

Review Axial Forces

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

Results Tab> Forces > ***Truss Forces***

Load Cases/Combinations>**CS:Summation**

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

Values > Value Output Details

Decimal Points > **(3) ; MinMax Only** (off)

Output Section Location > **Max** (on)

Stage Toolbar>**Stage 10** ↵

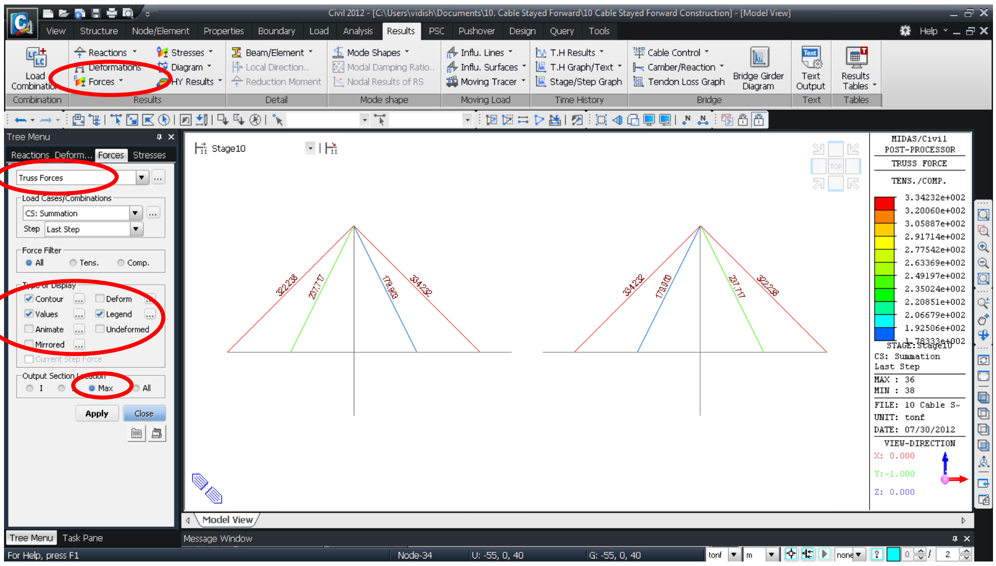


Fig. 68 Changes of Axial Forces for Each Construction Stage from Forward Analysis

Review Nodal Displacements & Member Forces Used for Calculating Lack-of-Fit Forces

For each construction stage, we review nodal displacements and member forces used for calculating Lack-of-Fit Forces.

Results/Result Tables / Construction Stage / Lack of Fit Force (Truss)

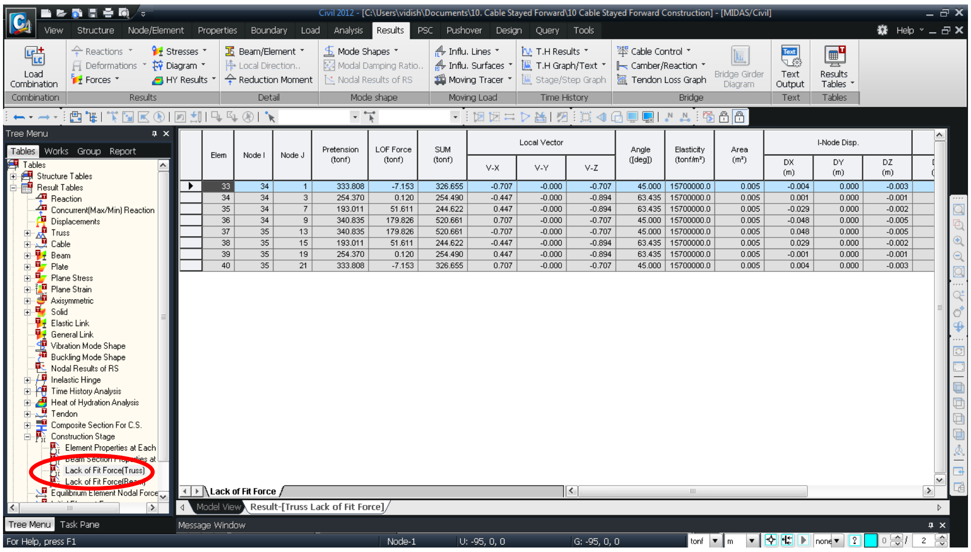


Fig. 69 Nodal Displacements and Member Forces used to calculate Lack-of-Fit Forces for Each Construction Stage

Compare Final Stage Analysis Results with Forward Stage Analysis Results

Compare cable pretension, deflection and moments of girders for final stage analysis with those for forward analysis. The comparison of cable pretension for final stage analysis and that for forward analysis is shown in Table 8.

Table 8. Comparison of Cable Pretension for Final Stage Analysis and that for Forward Stage Analysis

Element	Cable Pretension (Final Stage)		Cable Pretension (Forward Analysis)		Difference (%)	
	I	J	I	J	I	J
33	316.875	315.305	317.833837	316.263837	-0.3	-0.3
34	236.369	234.799	237.019938	235.449938	-0.27	-0.28
35	192.503	190.933	192.255350	190.685350	0.13	0.13
36	344.595	343.025	344.429928	342.859928	0.05	0.05
37	344.595	343.025	344.429928	342.859928	0.05	0.05
38	192.503	190.933	192.255350	190.685350	0.13	0.13
39	236.369	234.799	237.019938	235.449938	-0.27	-0.28
40	316.875	315.305	317.833837	316.263837	-0.3	-0.3

Compare girder deflection for final stage analysis with that for forward analysis.

Table 9. Comparison of Girder Deflection for Final Stage Analysis and that for Forward Stage Analysis

Node	Final Stage (mm)	Forward (mm)	Difference (%)
1	10.000	10.000000	0
2	6.902	6.948079	-0.66
3	4.710	4.781067	-1.49
4	1.948	2.002837	-2.74
5	-0.622	-0.622735	-0.12
6	-3.512	-3.601951	-2.5
7	-5.752	-5.943714	-3.23
8	-7.958	-8.244365	-3.47
9	-9.752	-10.109299	-3.53
10	-11.898	-12.292120	-3.21
11	-12.270	-12.668544	-3.15
12	-11.898	-12.292120	-3.21
13	-9.752	-10.109299	-3.53
14	-7.958	-8.244365	-3.47
15	-5.752	-5.943714	-3.23
16	-3.512	-3.601951	-2.5
17	-0.622	-0.622735	-0.12
18	1.948	2.002837	-2.74
19	4.710	4.781067	-1.49
20	6.902	6.948079	-0.66
21	10.000	10.000000	0

Compare girder moments for final stage analysis with those for forward analysis.

Table 10. Comparison of Girder Moments for Final Stage Analysis and those for Forward Stage Analysis

Node	Final Stage (tonf-m)	Forward (tonf-m)	Difference (%)
1	0.00	263.16	-
2	267.82	-401.67	1.74%
3	-392.35	109.39	-2.38%
4	117.82	-307.55	7.19%
5	-300.00	211.20	-2.53%
6	214.00	-198.06	1.31%
7	-200.00	168.83	0.98%
8	164.00	-392.27	-2.96%
9	-400.00	535.73	1.94%
10	528.00	651.73	-1.47%
11	644.00	535.73	-1.20%
12	528.00	-392.27	-1.47%
13	-400.00	168.83	1.94%
14	164.00	-198.06	-2.96%
15	-200.00	211.20	0.98%
16	214.00	-307.55	1.31%
17	-300.00	109.39	-2.53%
18	117.82	-401.67	7.19%
19	-392.35	263.16	-2.38%
20	267.82	263.16	1.74%
21	0.00	0.00	-