Advanced Application 3

Completed State and Construction Stage Analyses of a Suspension Bridge

Civil

CONTENTS

Introduction

Suspension bridges can generally be classified as long span structures. Suspension bridges comprise longitudinal deck (main girders) supported by hangers suspended from cables. The cables are connected to anchors at each end.

The analysis of a suspension bridge is divided into completed state analysis and construction stage analysis.

The ***completed state analysis*** is performed to check the behavior of the completed bridge. At this stage, the structure is in balance under self-weight, and the deflection due to the self-weight has already occurred. This stage is referred to as the initial equilibrium state of the suspension bridge. The initial equilibrium state analysis will provide the coordinates and tension forces in the cables. The completed state analysis of the suspension bridge is performed to check the behavior of the structure under additional loads such as live, seismic and wind loadings. The self weight loading in the initial equilibrium state will also be added to the total loading for the completed state analysis.

Suspension bridges exhibit significant nonlinear behavior during the construction stages. But it can be assumed that the bridge behaves linearly for additional loads (vehicle, wind load, etc.) in the completed state analysis. This is due to the fact that sufficient tension forces are induced into the main cables and hangers under the initial equilibrium state loading. It is thus possible to perform a linearized analysis for the additional static loads at the completed state by converting the tension forces in the main cables and hangers resulting from the initial equilibrium state loading into increased geometric stiffness of those components. This linearized analytical procedure to convert section forces to geometric stiffness is referred to as the ***linearized finite displacement method***. This procedure is adopted because a solution can be found with relative ease within acceptable error limits in the completed state analysis.

Construction stage analysis is performed to check the structural stability and to calculate section forces during erection. In carrying out the construction stage analysis, large displacement theory (geometric nonlinear theory) is applied in which equilibrium equations are formulated to represent the deformed shape. The effect of large displacements cannot be ignored during the construction stage analysis. The construction stage analysis is performed in a backward sequence from the state of equilibrium as defined by the initial equilibrium state analysis.

This tutorial explains the overall modeling and result analyzing capabilities for the completed state and construction stage analyses of a suspension bridge.

Procedure for Completed State Analysis

User may directly model the structure by fixing the towers & sag points without using the Wizard.

Initial equilibrium state analysis using Suspension Bridge Wizard
(Calculation of 1st cable coordinates)

↓

Modify the model & boundary conditions to reflect the true structure.

↓

Calculate cable coordinates and initial member forces satisfying the initial equilibrium state.

Accurate shape analysis (use Suspension Bridge Analysis Control)

↓

Eliminate Suspension Bridge Analysis Control Data in order to complete the initial equilibrium state analysis.

Eliminate Suspension Bridge Analysis Control Data

↓

Apply loadings to the completed structure for which initial member forces have been reflected into the geometric stiffness: Linearized finite displacement theory applied

Fig. 1 Analytical Model

Bridge Dimensions

The example model is a suspension bridge having a total length of 650m as shown in Fig. 1. Detailed bridge dimensions are shown in Fig. 2.

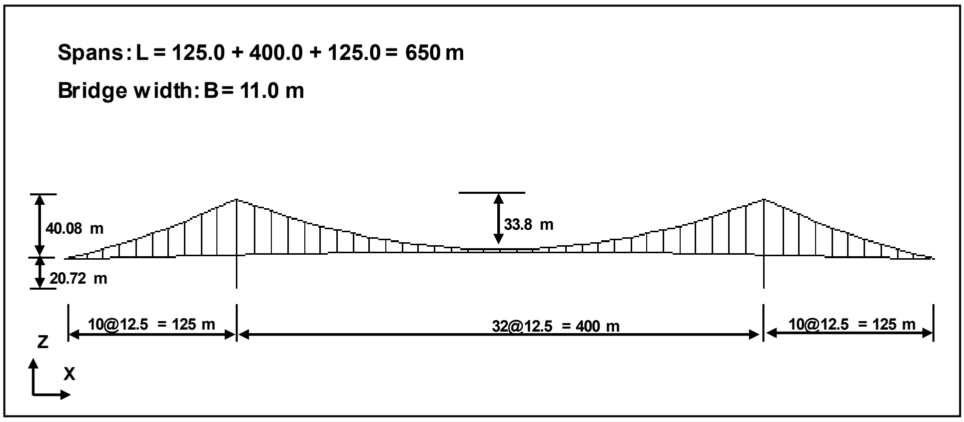


Fig. 2 General Profile

Completed State Analysis Modeling

Structural Modeling

In this tutorial, the suspension bridge modeling sequence is as follows. First, create the model for the completed state analysis, perform completed state analysis, and then create the construction stage analysis model under a different name.

The suspension bridge modeling procedure for the completed state analysis is as follows:

1. Define material and section properties
2. Analyze initial equilibrium state (using Suspension Bridge Wizard)
3. Create a model and enter boundary conditions
   - Divide pylon (tower) members to generate pylon transverse beams
   - Create & remove pylon transverse beams
   - Enter boundary conditions
4. Accurate initial equilibrium state analysis
   - Define structure groups
   - Enter self weight
   - Perform analysis
5. Input static loads & modify boundary conditions
6. Perform completed state analysis

Assign Working Environment

Open a new file (New Project), save as "Suspension Bridge .mcb" (***Save***) and assign a unit system.

/ ***New Project*** / ***Save*** **(Suspension Bridge)**

Tools / Unit System **Unit System** (alternatively select from the status bar at the bottom of the screen)

Length>**m**; Force> **tonf** ↵

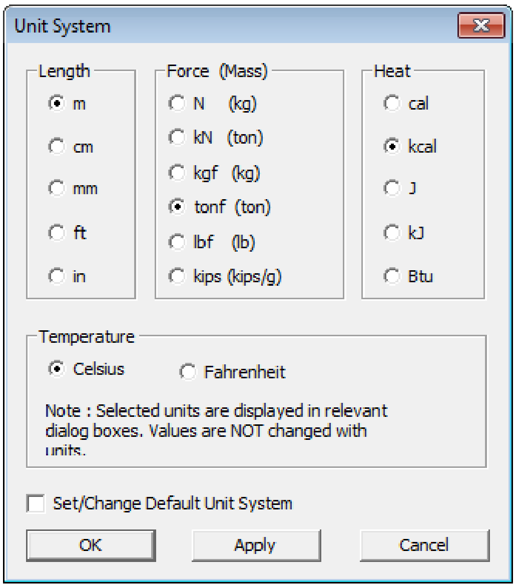
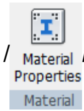


Fig. 3 Assign unit system

In this tutorial, 3-dimensional analysis will be performed.

Define Material Properties

Input material properties for cable, hanger, deck (main girder) and pylon.

Properties / 

Material

Name>**Cable**
Type>**User Defined**
Modulus of Elasticity **(2.0e+7)**
Weight Density **(8.267)** ↵

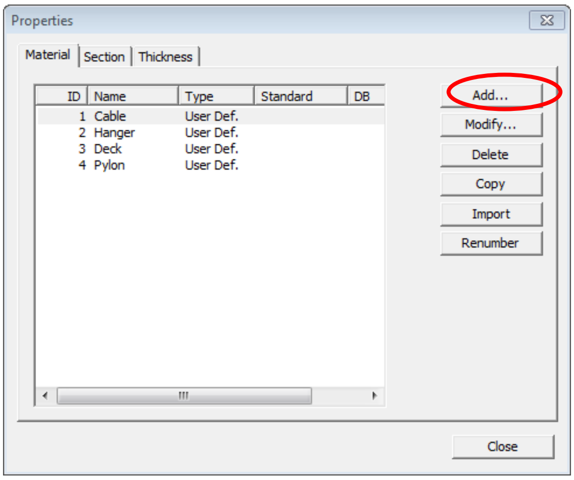


Fig. 4 Define material properties

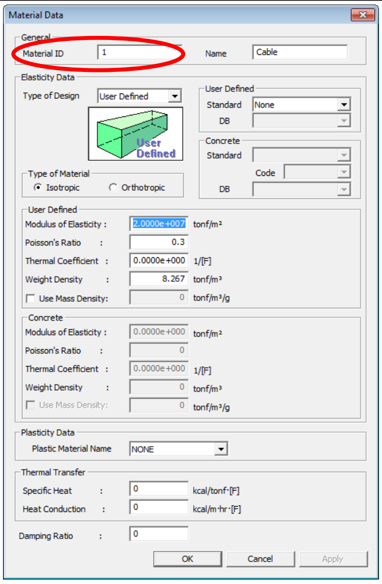


Fig. 5 Material data

By the same method as above, input material properties for hangers, deck (main girders) and pylons using Table 1.

Table 1 Element material properties [unit: tonf, m]

Classification	Cable	Hanger	Deck	Pylon
Type	User Defined	User Defined	User Defined	User Defined
Modulus of Elasticity	2.0×10^7	1.4×10^7	2.1×10^7	2.1×10^7
Poisson's Ratio	0.3	0.3	0.3	0.3
Weight Density	8.267	7.85	0.00 🎧	7.85

🎧 **Because the self weight of Deck is directly entered as nodal loads by the user, Weight Density is assigned 0.**

Define Section Properties

Input the section properties using Fig. 6 and Table 2 as follows:

Properties / 

Section

Value>Section ID **(1)** ; Name **(Cable)**
Size>D **(0.23)** ; Stiffness>Area **(0.04178)** ↵

[unit: m]

Table 2 Section properties

Classification	Cable	Hanger	Deck	Pylon	Pylon-trans
Area	0.04178	0.00209	0.5395	0.16906	0.1046
Ixx	0	0	0.4399	0.1540	0.1540
Iyy	0	0	0.1316	0.1450	0.1080
Izz	0	0	3.2667	0.1143	0.0913

Note that D=0.23 is used for graphical representation only, and the numerical properties in Section Properties (ie, A=0.04178) are used for analysis. They do not have to necessarily correspond. After entering the Size and clicking on Calc. Section Properties produces the numerical properties, which can be subsequently changed.

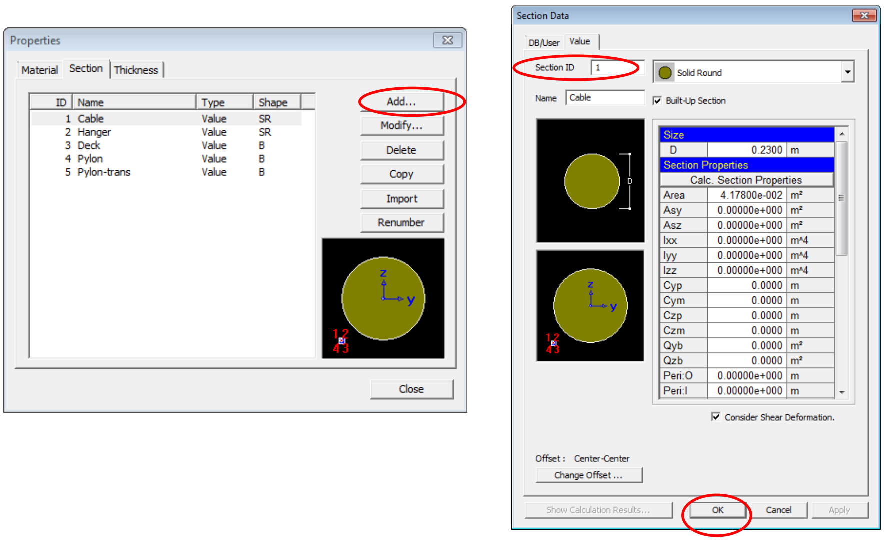


Fig. 6 Input section properties (cable)

Input section properties for other elements using Fig. 7 and 8.

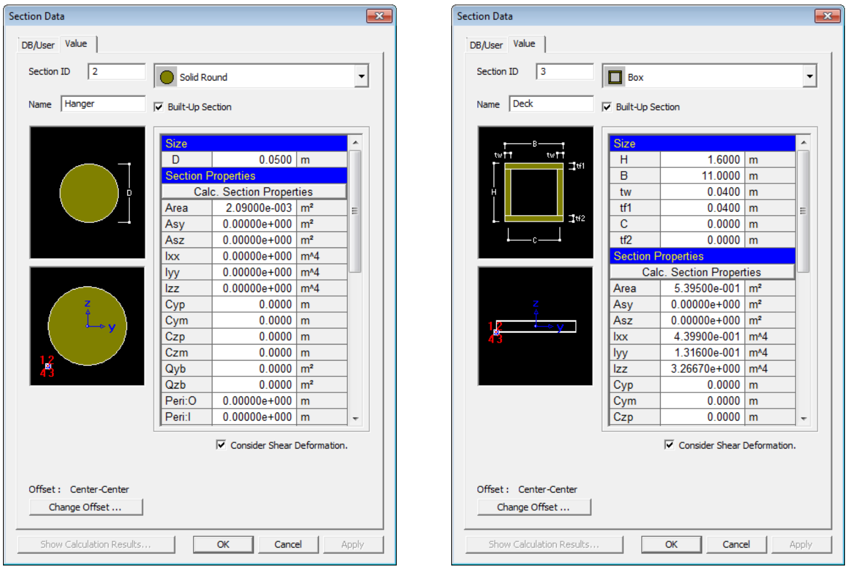


Fig. 7 Input section properties for Hanger & Deck (main girder)

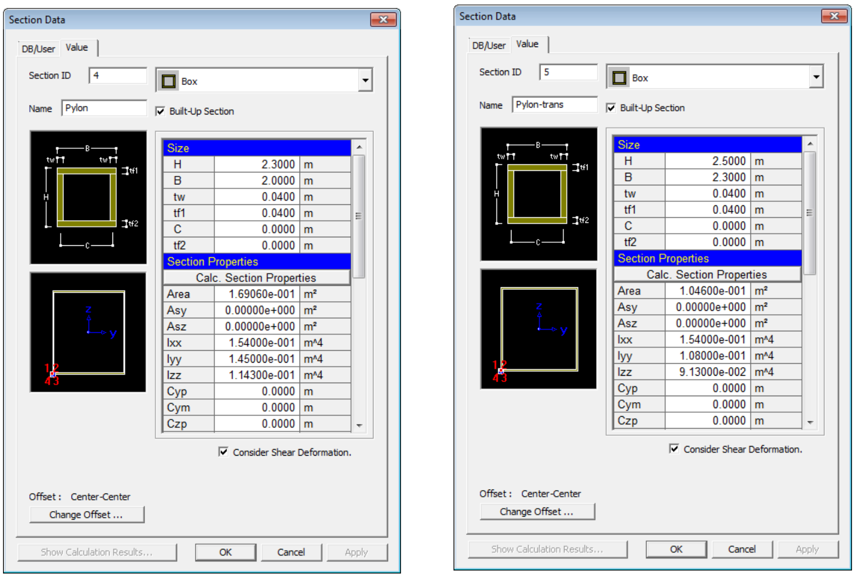


Fig. 8 Input section properties for Pylon & Pylon-trans beam

Initial Equilibrium State Analysis

In the completed state analysis of the suspension bridge, the deflections due to self-weight have already occurred, and the structure has come to an equilibrium state. In this initial equilibrium state, the cable coordinates and tension forces are not simply assumed by the designer, but rather they are automatically determined by using equilibrium equations within the program.

Using the ***Suspension Bridge Wizard*** function, the coordinates of the cables and the initial tension forces within the cables and hangers and the forces in the pylons can be calculated automatically. The initial equilibrium state is determined by inputting the basic dimensions of cable sag, hanger spacing and the self-weight applied to each hanger. The cable and hanger tension forces determined by the Suspension Bridge Wizard are automatically converted into increased geometric stiffness using the ***Initial Force for Geometric Stiffness*** function within the program.

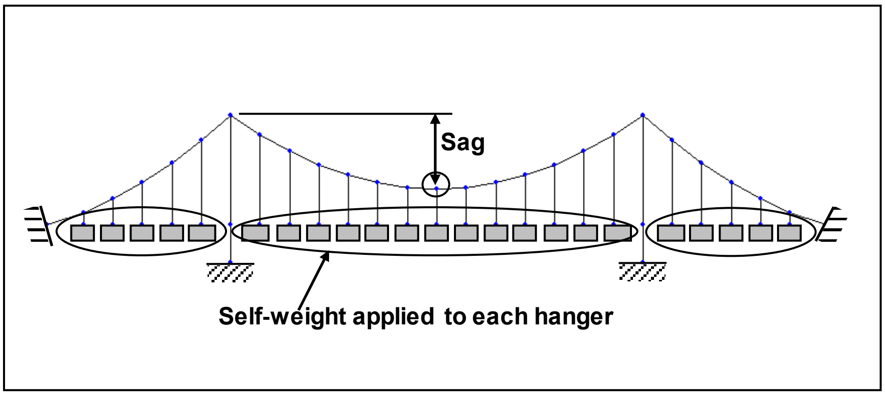


Fig. 9 2-dimensional basic shape for suspension bridge

To obtain the initial tension forces and basic shape, input appropriate data into ***Suspension Bridge Wizard*** as per Fig. 10.

Structure / Suspension Bridge ***Suspension Bridge***
  Node Coordinates & heights > 3-Dimensional **(on)**
    A **(0)**, **(0)**, **(20.48)** ; A1 **(3.6)**, **(0)**, **(20.72)** ;
    B **(128.6)**, **(0)**, **(60.8)** ; C **(328.6)**, **(0)**, **(27)**
  Height **(60.8)**
  Hanger Distance (m)
    Left **(10@12.5)**
    Center **(32@12.5)** ↵
  Material> Main Cable **(1: Cable)** ; Side Cable **(1: Cable)**
    Typical Hanger **(2: Hanger)** ; End Hanger **(2: Hanger)**
    Deck **(3: Deck)** ; Pylon **(4: Pylon)**
  Section> Main Cable **(1: Cable)** ; Side Cable **(1: Cable)**
Typical Hanger **(2: Hanger)** ; End Hanger **(2: Hanger)**
Deck **(3: Deck)** ; Pylon **(4: Pylon)**
  Deck System
    Width **(11)**
    Shape of Deck **(on)** ; Left Slope **(2.77)** ; Arc Length **(650)**
    Advanced…**(on)**
    Advanced unit weight of deck system
    Load Type > **Point Load (on)**
    Left **(9@52.9375)**
    Center **(31@52.9375)** ↵

The program automatically calculates the self weight of the cables. Only the self weight of the Deck needs to be entered.

W_d (Weight of Deck per unit length) : 4.235 tonf/m (assumed)
L_d (Longitudinal spacing of hanger) : 12.5 m
Ignore hanger self-weight

As explained earlier, the geometric shape of the suspension bridge, especially the cable coordinates cannot be arbitrarily determined by the designer. Rather they will be determined by the catenary equation satisfying the equilibrium condition within the program. Using the Suspension Bridge Wizard function, the geometric shape and initial tension forces can be calculated. As shown in Fig. 10, all coordinates of the suspension bridge, including the coordinates of the cables can be determined automatically by entering the coordinates of the pylons, sag (B-C), slope of deck, hanger spacing and self weight applied to the hangers.

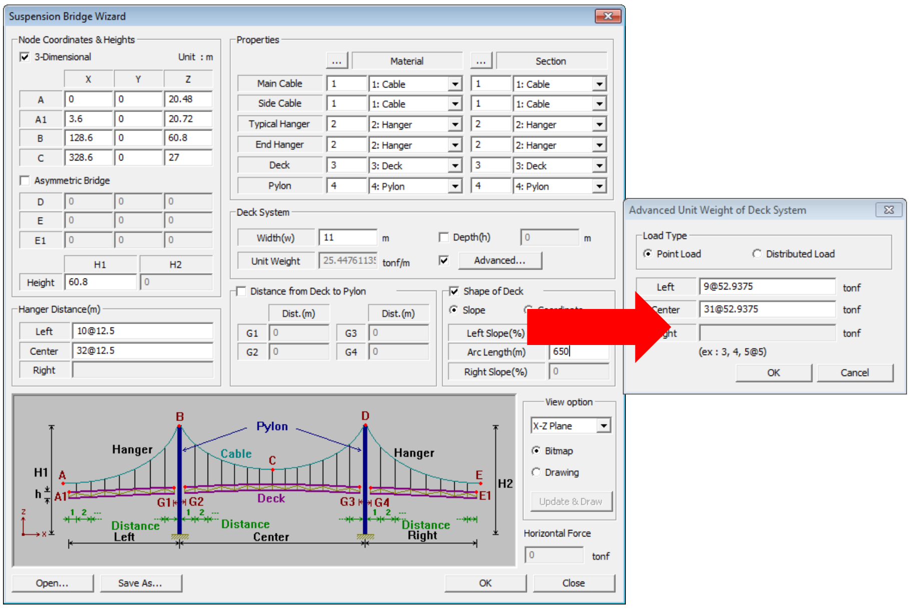


Fig. 10 Suspension Bridge Wizard Input Window

Fig. 11 is the 3D shape generated by the ***Suspension Bridge Wizard*** function. The main cables and hangers are generated as cable elements, and the deck and pylons are generated as beam elements.

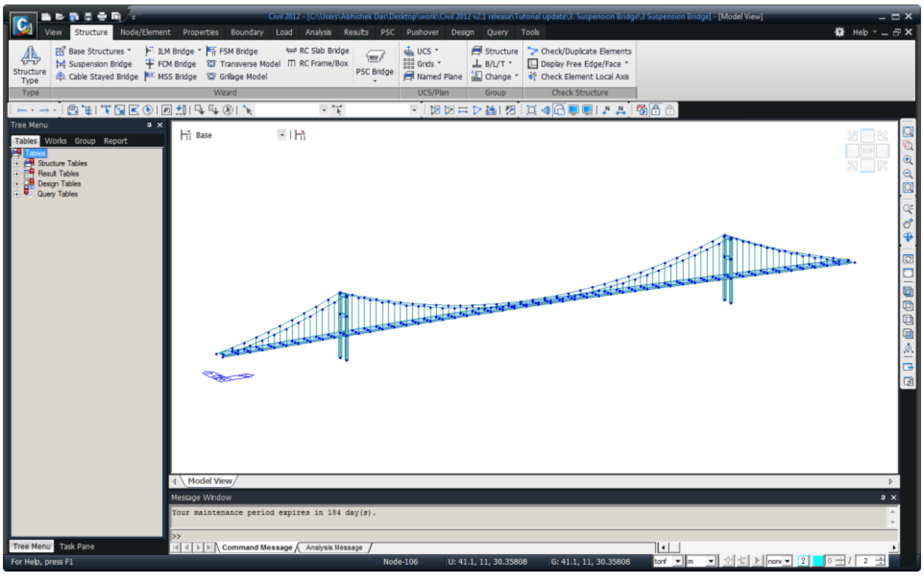


Fig. 11 Initial equilibrium state analysis using Suspension Bridge Wizard

Divide pylon elements to create pylon transverse beams

Align nodes 258 & 260 in line with node 215 and align nodes 262 & 264 in line with node 247.

Node/Element / Translate ***Translate…***

Select Nodes 258, 260, 262, 264

Mode > Move (on)

Translation > Equal Distance (on) ; dx, dy, dz : **0, 0, 2.796635**

(z coordinate of nodes 258, 260, 262, 264 = 20.72 and

z coordinate of nodes 215 & 247 = 23.516635)

Number of times : **1** ↵

To create the pylon transverse beams, divide the pylons as shown in Fig. 12.

Node/Element/ Divide ***Divide…***

View / / Intersect Line ***Select Intersect*** (Elements : **255, 258, 260, 263**)

Divide>Element Type>**Frame**

Unequal Distance **(1.25, 18.75)** ↵

View / ***Shrink Elements***

Input the distances to locate the pylon transverse beams from the top of the pylons.

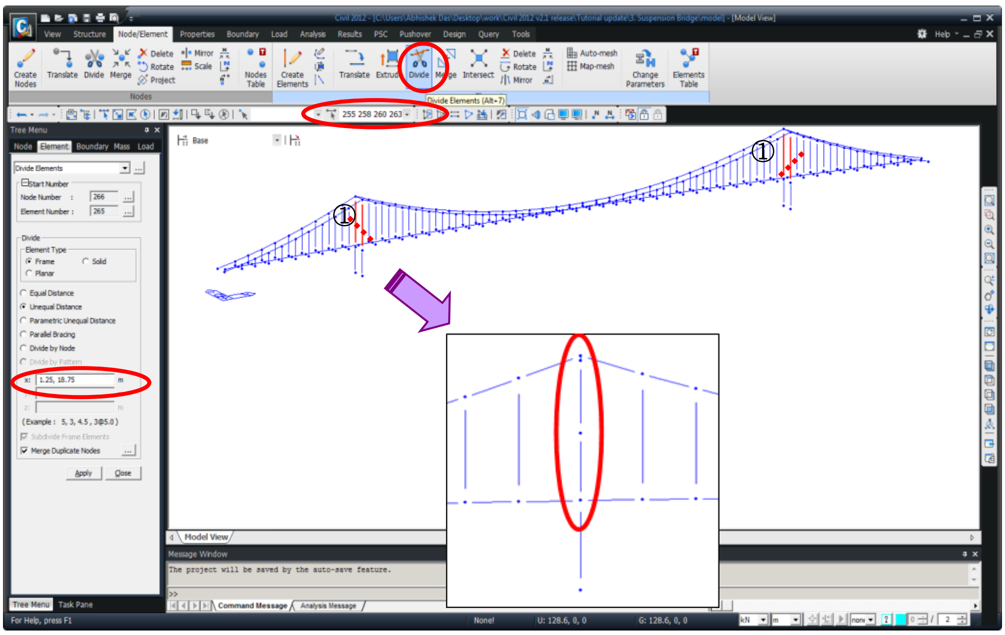


Fig. 12 Pylon element division

Create pylon transverse beams

Generate the pylon transverse beams as follows:

Zoom (***Window*** Magnify the left pylon as Fig. 13)

Node/Elements / Create Elements ***Create…***

Element Type>General beam/Tabered beam

Material>**4: Pylon** ; Section>**5: Pylon-trans**

Intersect>**Node (on)** ; **Elem (on)**

Nodal Connectivity **(260, 258)** ; **(269, 267)** ; **(268,266)** ↵

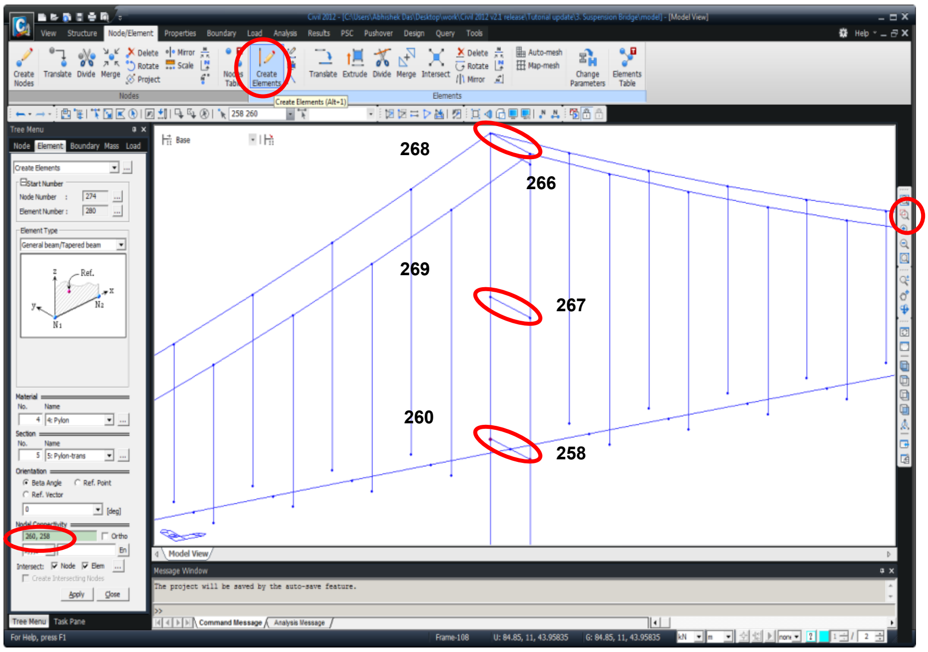


Fig. 13 Generate pylon transverse beams (left pylon)

Generate the pylon transverse beams for the right pylon.

Zoom Fit
Zoom Window (Magnify the right pylon as Fig. 14)
Node/Element / Create Elements ***Create Elements***

Element Type>General beam/Tabered beam
Material>**4: Pylon** ; Section>**5: Pylon-trans**
Intersect>**Node (on)** ; **Elem (on)**
Nodal Connectivity **(264, 262)** ; **(273, 271)** ; **(272, 270)** ↵

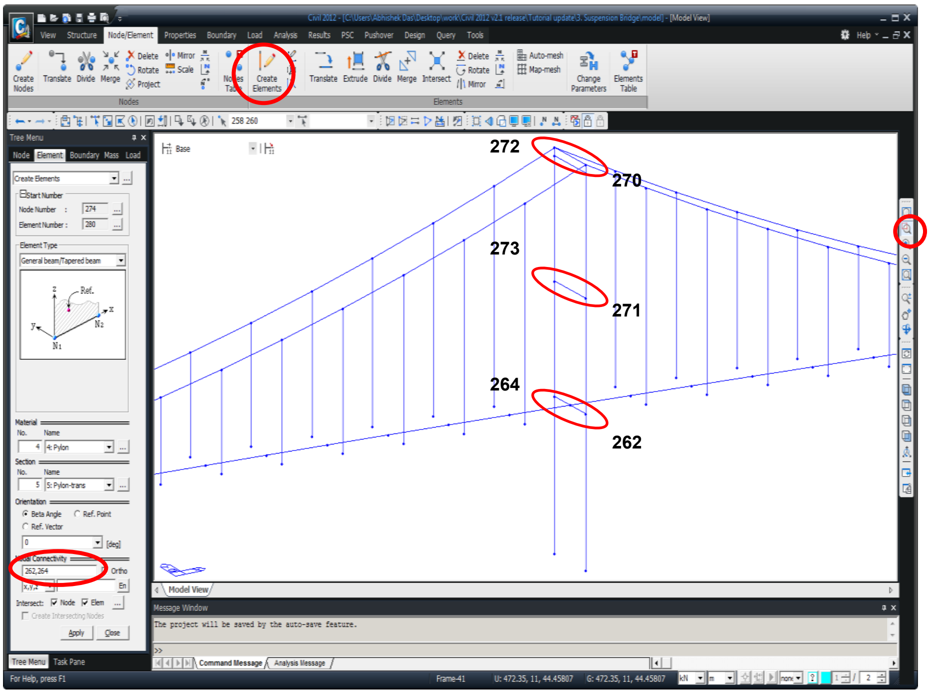


Fig. 14 Generate pylon transverse beams (right pylon)

Remove pylon transverse beams

Remove the very top pylon transverse beams generated by the Wizard.

Node/Element / Delete ***Delete…***
Select Identity-Elements
(257, 262)
Type>Selection **(on)** ↵

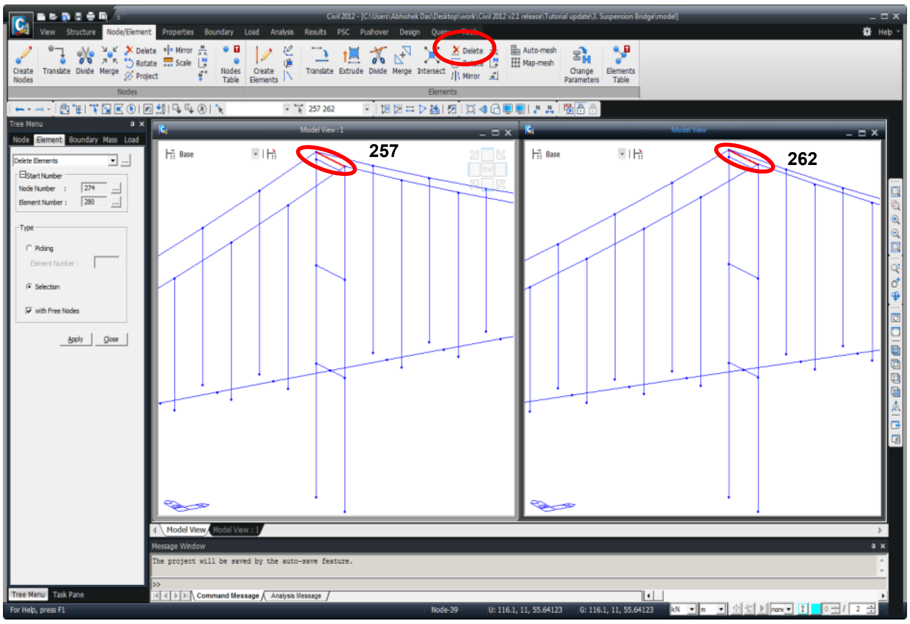


Fig. 15 Remove pylon transverse beams

Input Boundary Conditions

Input boundary conditions for the pylons, cable anchors and the ends of the side spans.

Cable anchors: fix (Nodes: 1, 103, 53, 155)

Pylon base: fix (Nodes: 259, 261, 263, 265)

(Fixed supports are automatically generated and entered upon execution of Wizard.)

Ends of side spans: hinge with rotational restraints (Nodes: 205, 257)

Iso View ; ***Zoom Window***

Boundary / Define Supports ***Supports***

Zoom Fit ; ***Zoom Window***

Select Single (Nodes: **205, 257**)

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

Rotational restraint about the bridge axis is provided at the bearings.

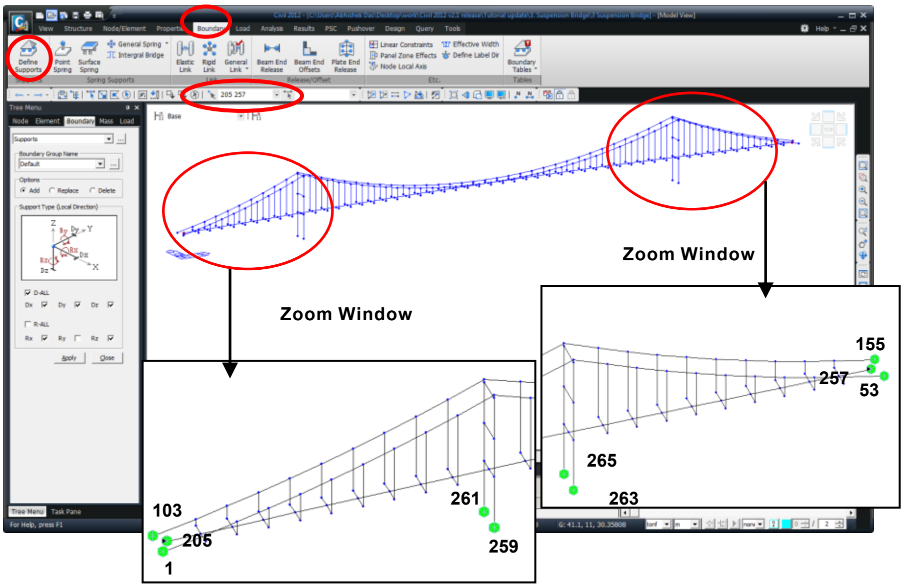


Fig. 16 Input Boundary Condition

In this model, the boundary condition for the deck at the pylons is roller, which is separated as shown in Fig. 17. Assign the boundary condition for the deck at the pylons as a roller condition using the ***Beam End Release*** function.

Zoom Fit ; ***Zoom Window*** (Magnify the left pylon part as shown in Fig. 17)

Boundary / Beam End Release ***Beam End Release***

Boundary Group Name>**Default**

Options>**Add/Replace**

Select Single (Elements: **212**)

General Types and Partial Fixity

My (i-Node) (on) ; **Fx (j-Node)** (on) ; **My (j-Node)** (on) ↵

Select Single (Elements: **213**)

General Types and Partial Fixity

Fx (i-Node) (on) ; **My (i-Node)** (on) ; **My (j-Node)** (on) ↵

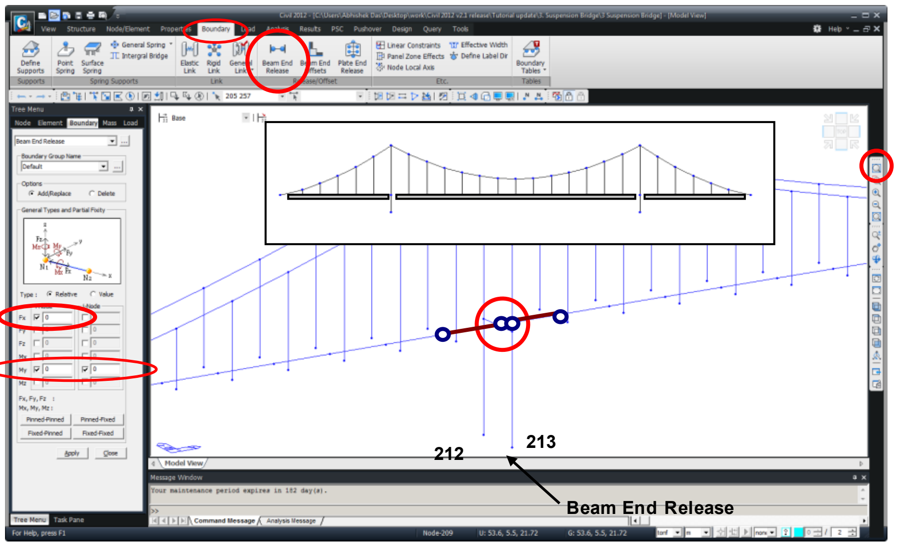


Fig. 17 Input connection condition for the deck at the left pylon

Similarly, assign the boundary condition for the deck at the right pylon.

Zoom Fit
Zoom Window (Magnify the right pylon part as shown in Fig. 18)
Boundary / Beam End Release ***Beam End Release***

Boundary Group Name>**Default**
Options>**Add/Replace**
Select Single (Elements: **244**)
General Types and Partial Fixity
My (i-Node) (on) ; **Fx (j-Node)** (on) **; My (j-Node)** (on) **;** ↵
Select Single (Elements: **245**)
General Types and Partial Fixity
Fx (i-Node) (on); **My (i-Node)** (on) **; My (j-Node)** (on) **;** ↵

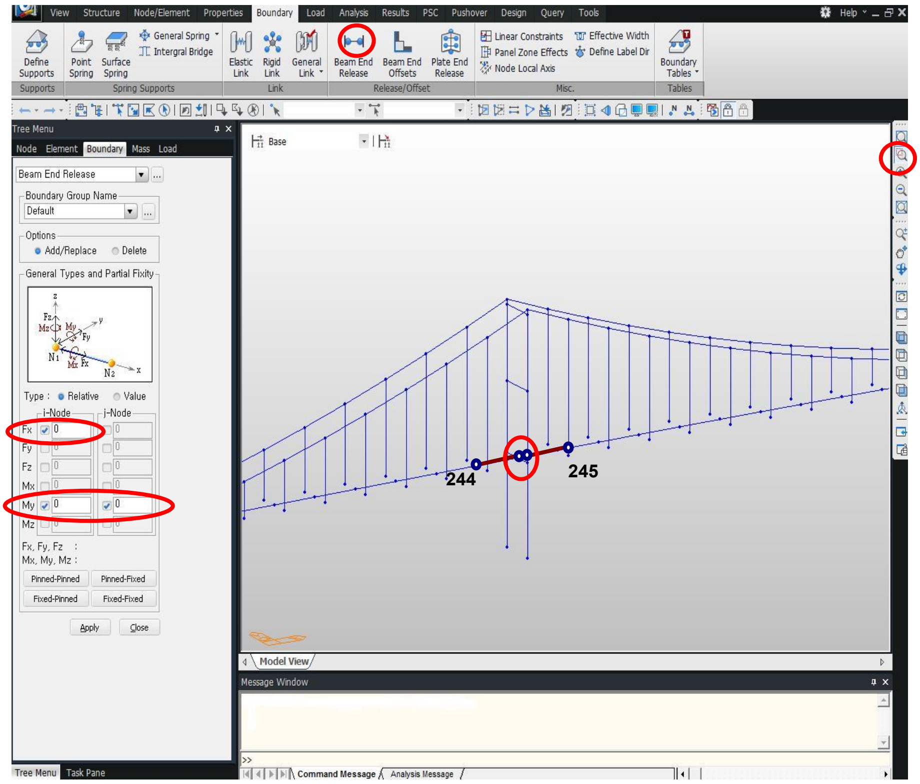


Fig. 18 Input connection condition for the deck at the right pylon

In the case of a suspension bridge with dead anchors for cables in which decks (girders) are initially unconnected with hinges while being hung from the hangers and subsequently connected, the decks are unstressed at the initial equilibrium state. In such hinge construction, the Beam End Release function is used to release moments in the decks prior to carrying out the initial equilibrium state analysis using Suspension Bridge Analysis Control.

When releasing moment about My, only one element at a node is released to avoid instability as shown in Fig. 19.

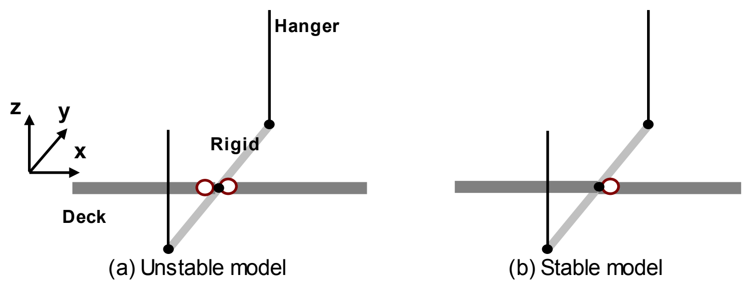


Fig. 19 Pin connection of decks

As shown in Fig, 20, j-end of the decks is ***Beam End Released*** in the part ①, and i-end of the decks is ***Beam End Released*** in the part ②.

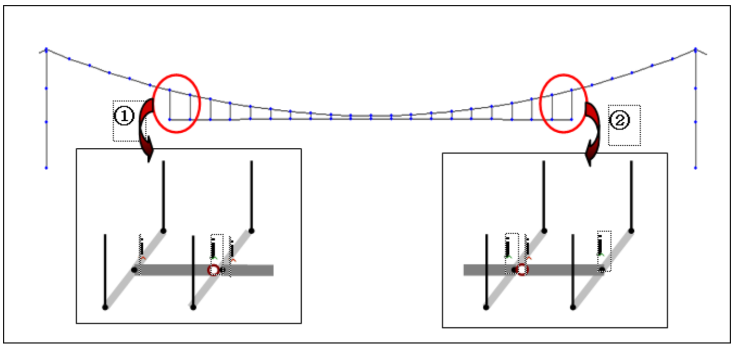


Fig. 20 Pin connection of decks (construction stages)

Define boundary group

Group tab

Group>Boundary Group>New…

Name **(Pin Connection)** Add

The decks in the parts ① and ② in Fig. 21 are ***Beam End Released*** at i-end about M_y .

Boundary / Beam End Release ***Beam End Release***

Options>**Add/Replace**

Select Window (Elements: ①, ② in Fig. 21)

(Elements: **204to211 230to243**)

Boundary group name > **Pin Connection**

General Types and Partial Fixity>**My (i-Node)** (on) ↵

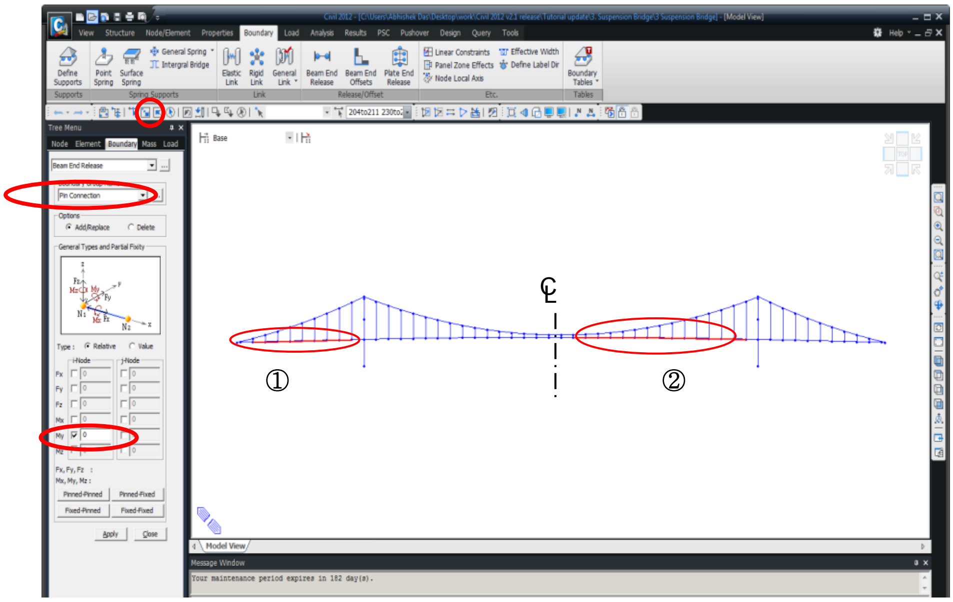


Fig. 21 Definition of pin connections of decks

The decks in the parts ① and ② in Fig. 22 are ***Beam End Released*** at j-end about M_y .

Boundary / Beam End Release ***Beam End Release***

Options > **Add/Replace**

Select Window (Elements : ①, ② in Fig. 22)

(Elements : **214to228 246to253**)

Boundary group name > **Pin Connection**

General Types and Partial Fixity > **My (j-Node)** (on) ↵

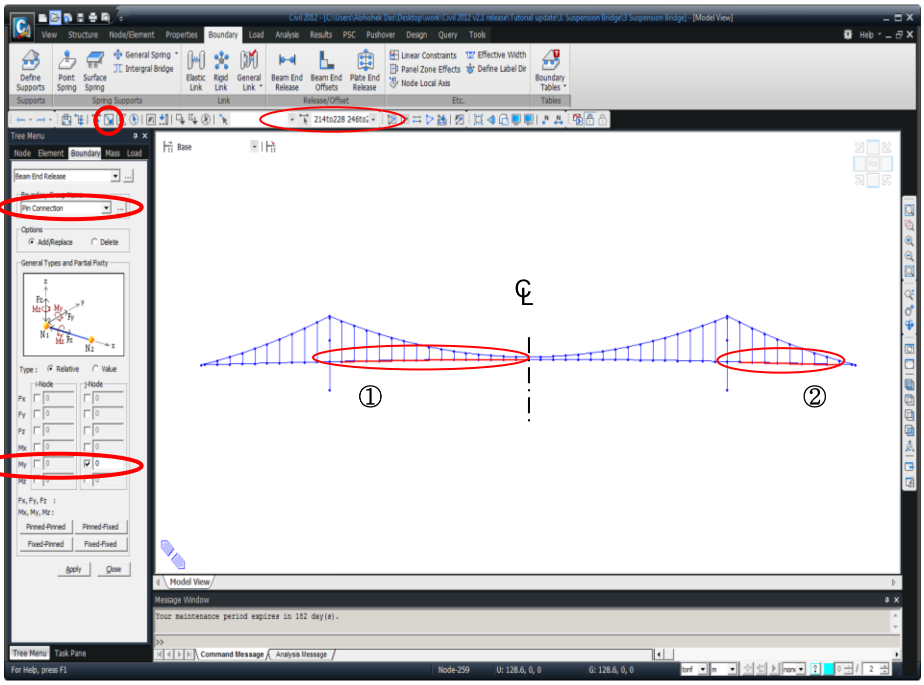


Fig. 22 Definition of pin connections of decks

Define Structure Groups

In order to carry out the analysis for cable initial shape for the total structural system, which contains the pylons and decks, using Suspension Bridge Analysis Control, we need to define Structure Groups for Sag Points, whose coordinates are unchanged, and Nodes, which need to be updated.

Group tab

Group>Structure Group>New…

Name **(Nodes to be updated)** Add

Name **(Sag Points)** Add

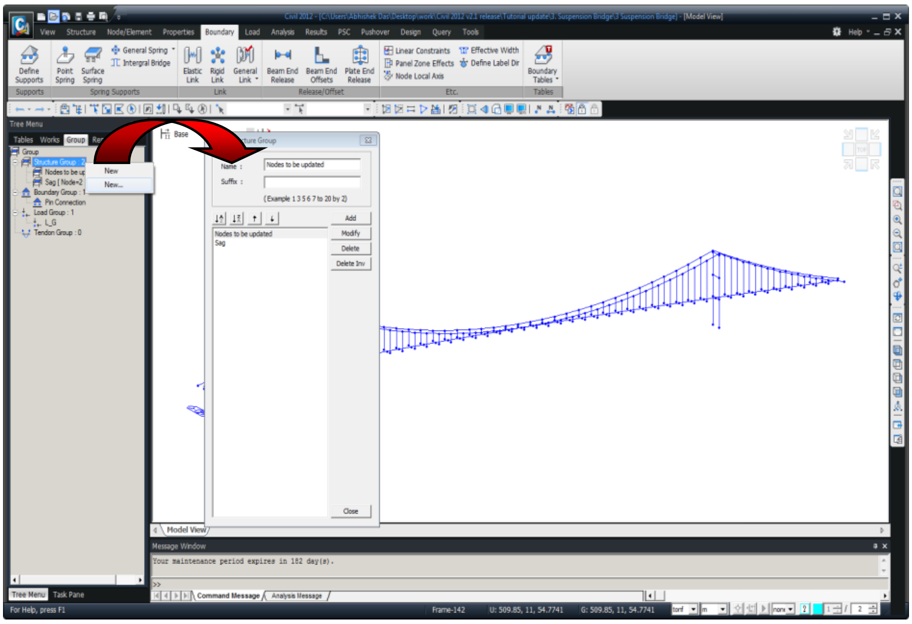


Fig. 23 Define Structure Groups

In order to execute Suspension Bridge Analysis Control, we define Structure Groups for the nodes joining the cables and hangers and the nodes corresponding to Sag Points of cables at the center span.

Group > Structure Group

Select Identity-Nodes

(2to10, 12to42, 44to52, 104to112, 114to144, 146to154)

Nodes to be updated *(Drag & Drop*)

Select Identity-Nodes (27, 129)

Sag Points (*Drag & Drop)*

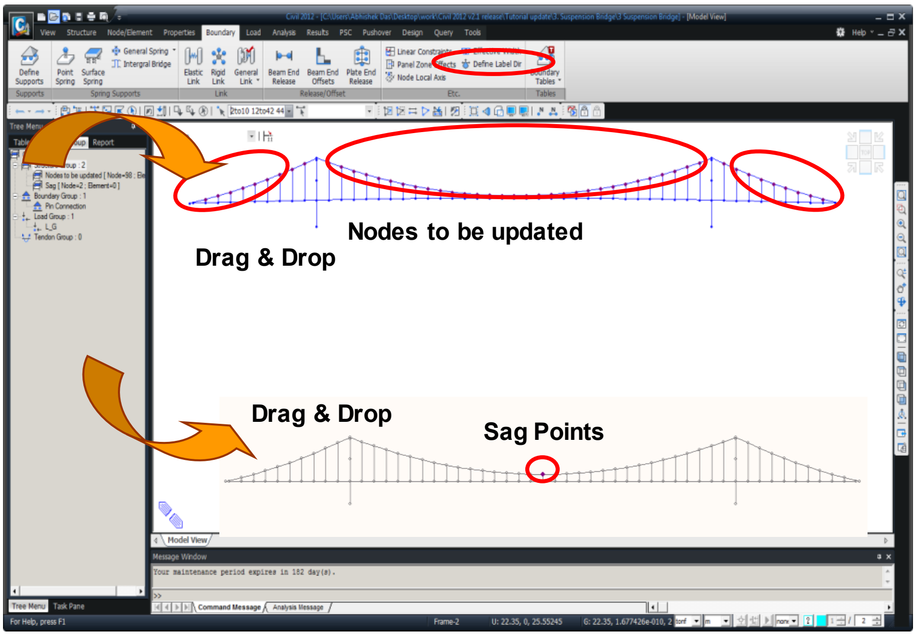


Fig. 24 Define Structure Groups

Input loadings

The **Static Load Case**, ***Self Weight***, is automatically generated and entered upon execution of Wizard. Define a Load Group for Self Weight and modify the Load Group of Self Weight already created.

Load / Static Loads / Self Weight ***Self Weight***

Load Case Name > **Self Weight (Select)**

Load Group Name > [...] ↵

Define Load Group > Name > **L_G** ; [Add] ↵

[Close] ↵

Load Group Name > **L_G**

Self weight factor > **Z = -1**

[Add] ↵

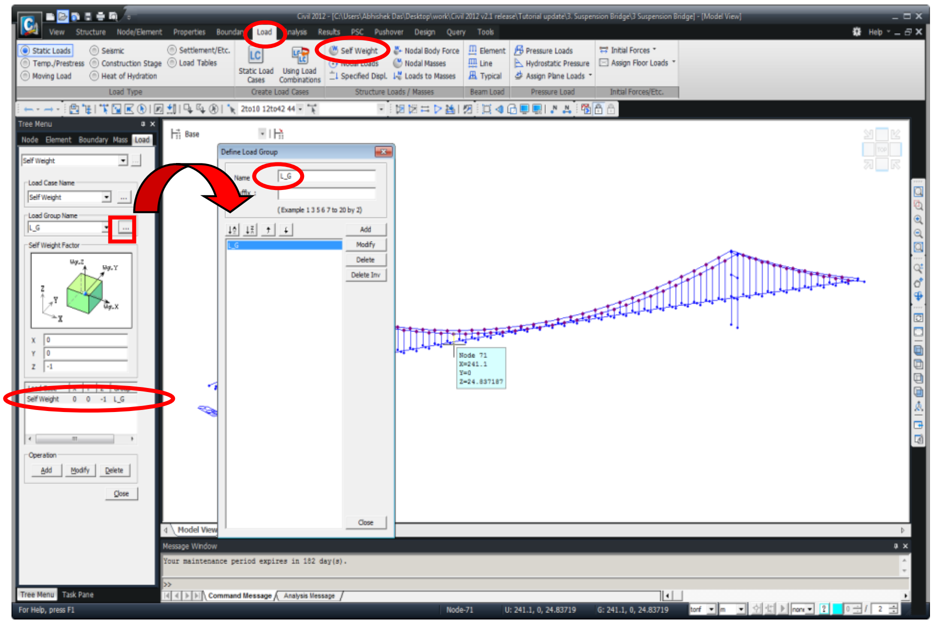


Fig 25. Entry of self weight excluding the decks

🎧 Since the Weight Density of the decks is 0, the self weight of the decks cannot be considered by the Self Weight function.

Because the weight of the decks was entered 0, we specify the self weight of the decks. 🎧

W_d (Weight of Deck per unit length) : 4.235 tonf/m (assumed)

L_d (Longitudinal spacing of hanger) : 12.5 m

Ignore hanger self-weight

Self weight of the decks acting on the hangers

Deck : $W_d/2$ x L_d =4.235 / 2 x 12.5 = 26.469 tonf

Load / Static Loads / Nodal Loads ***Nodal Loads***

Select identity – Nodes (54to102 156to204)

Load Case Name > **Self Weight**

Load Group Name > **L_G** ; Options **> Add**

Nodal Loads

Fz : **-26.469** ↵

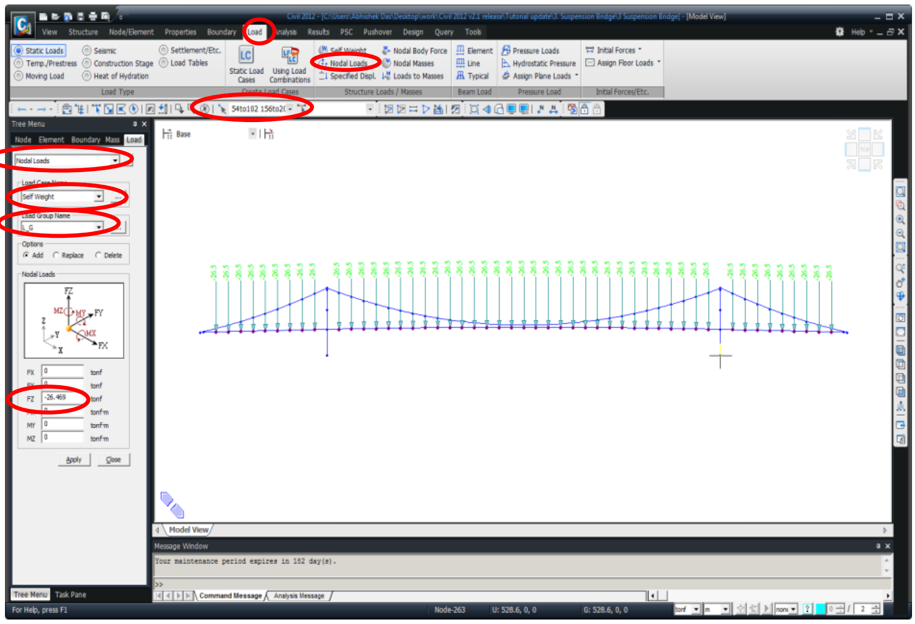


Fig. 26 Self weight of decks

Suspension Bridge Analysis Control

Suspension Bridge Analysis Control executes accurate initial equilibrium state analysis for the total structural system, which reflects modified pylons and decks, based on the cable coordinates generated from Suspension Bridge Wizard, unstressed length and horizontal tensions.

Analysis / Suspension Bridge ***Suspension Bridge Analysis Control***

Control Parameters> Number of Iterations ; **(10)**
Node Group to be Updated ; **(Nodes to be updated)**
Convergence Tolerance ; **(1e-005)**
Sag Point Group ; **(Sag Points)**
Constant Horizontal Force of Cable>(off)
Load Case to be Considered; **(Self Weight)**
Scale Factor; **1** [Add] ↵
[OK] ↵

Accurate initial equilibrium state analysis is performed for the self weight of the bridge.

Analysis / Perform Analysis ***Perform Analysis***

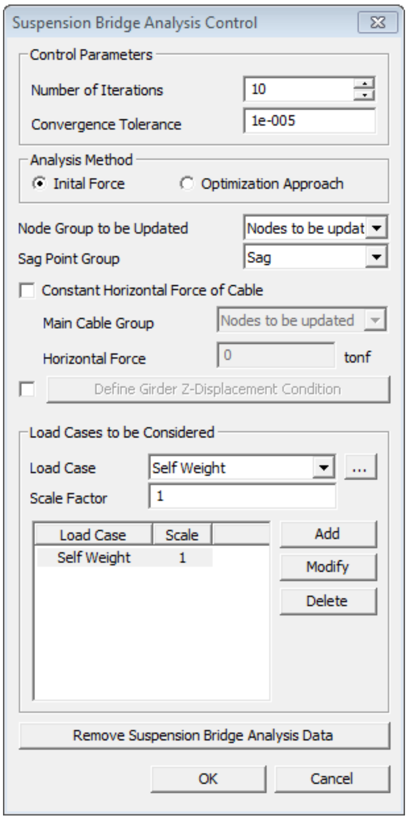


Fig. 27 Suspension Bridge Analysis Control

Upon execution of Suspension Bridge Analysis Control, Initial Forces (Large Displacement) are calculated, which are used to represent the initial equilibrium state in large displacement analysis and construction stage large displacement analysis. Initial Forces (Large Displacement) includes Initial Forces for Geometric Stiffness and Equilibrium Element Nodal Force. Initial Forces (Small Displacement) are calculated, which are used to represent initial equilibrium state in linear analysis. Initial Forces (Small Displacement) includes Initial Element Forces. The calculated values can be checked in tables.

The values of Initial Forces calculated by Suspension Bridge Analysis Control can be readily checked in tables by right-clicking in Works Tree.

Load / Load Tables / Initial Forces Geometric Stiff. ***Initial Forces for Geometric Stiffness*** ↵

Load / Static Loads / Initial Forces Initial Forces / Large Displacement / ***Equilibrium Element Nodal Force*** ↵

Load / Static Loads / Initial Forces Initial Forces / Small Displacement / ***Initial Element Forces*** ↵

Initial Forces (Large Displacement)

Initial Forces for Geometric Stiffness

This is used to represent initial equilibrium state in construction stage large displacement analysis and large displacement analysis. The program internally generates external forces, which are in equilibrium with the entered member forces as well as the initial forces. Once the initial forces are considered for formulating geometric stiffness, the data is ignored in linear analyses such as completed state analysis.

Equilibrium Element Nodal Force (used in construction stage analysis)

Equilibrium Element Nodal Forces are used specifically for backward construction stage large displacement analysis. Without loads, which are in equilibrium with these nodal forces, the nodal forces cause deformation. The nodal forces are ignored in large displacement analysis having no construction stages.

Initial Forces (Small Displacement)

Initial Element Forces

Initial element forces are considered in formulating geometric stiffness in completed state linear analysis. This data is ignored if large displacement analysis is carried out.

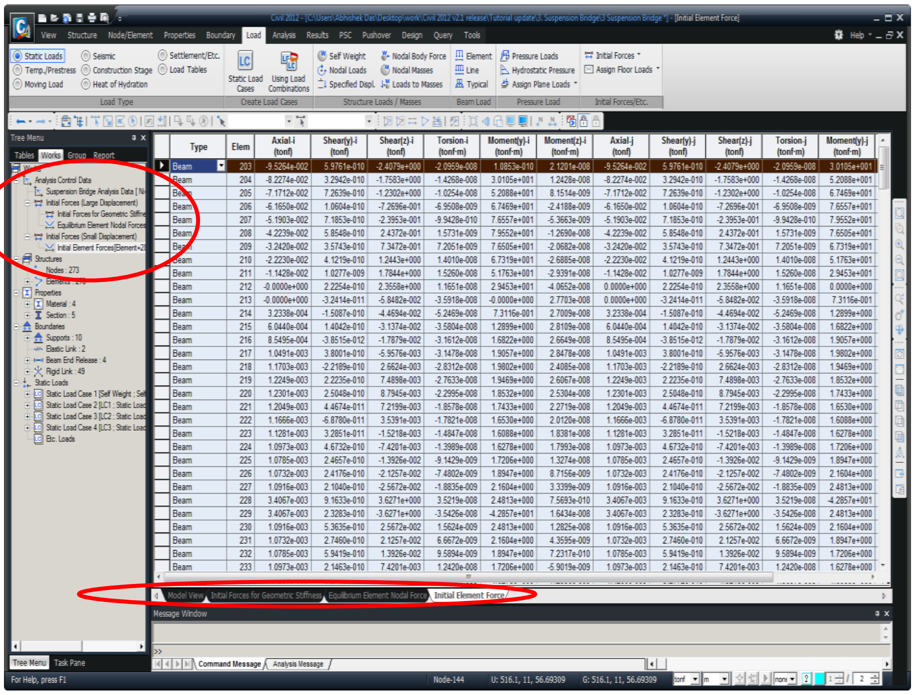


Fig. 28 Initial Forces Tables

Remove Nonlinear Analysis Control Data and Suspension Bridge Analysis Data

Linearized finite displacement analysis is sufficient for the completed state analysis, so it is carried out as such. However, because initial equilibrium state analysis is carried out by nonlinear analysis when Suspension Bridge Wizard is executed, Nonlinear Analysis Control Data is generated. We now need to remove Nonlinear Analysis Control Data to perform linear analysis. Also, once we obtain member forces to formulate geometric stiffness through Suspension Bridge Analysis Control, we can then remove Suspension Bridge Analysis Data to perform completed state analysis.

Analysis / Nonlinear ***Nonlinear Analysis Control...***
Remove Nonlinear Analysis Control Data ↵

Analysis / Suspension Bridge ***Suspension Bridge Analysis Control...***
Remove Suspension Bridge Analysis Data ↵

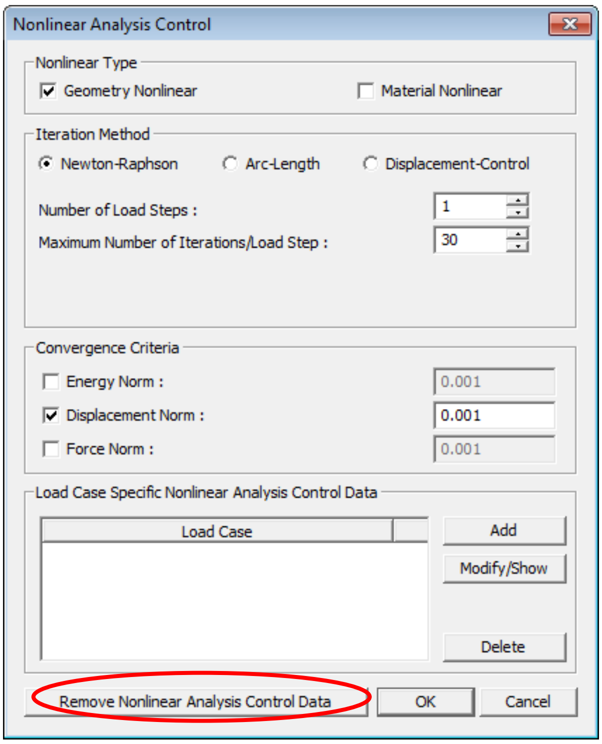


Fig. 29 Remove Nonlinear Analysis Control Data

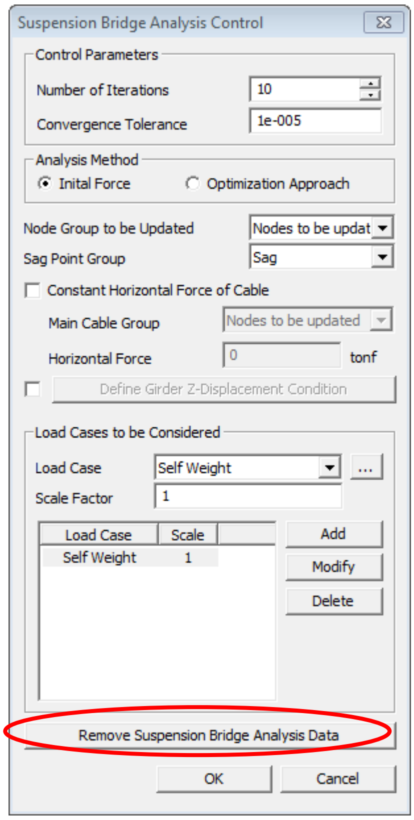


Fig. 30 Remove Suspension Bridge Analysis Data

Remove and Modify Beam End Release Conditions for Deck

After initial equilibrium state analysis, completed state analysis is performed with the decks being connected. As such, we now remove the Beam End Release conditions for the decks.

Tree Menu>**Works tab**

Boundaries>Beam End Release>Type 1 : **Delete** ↵

Boundaries>Beam End Release>Type 4 : **Delete** ↵

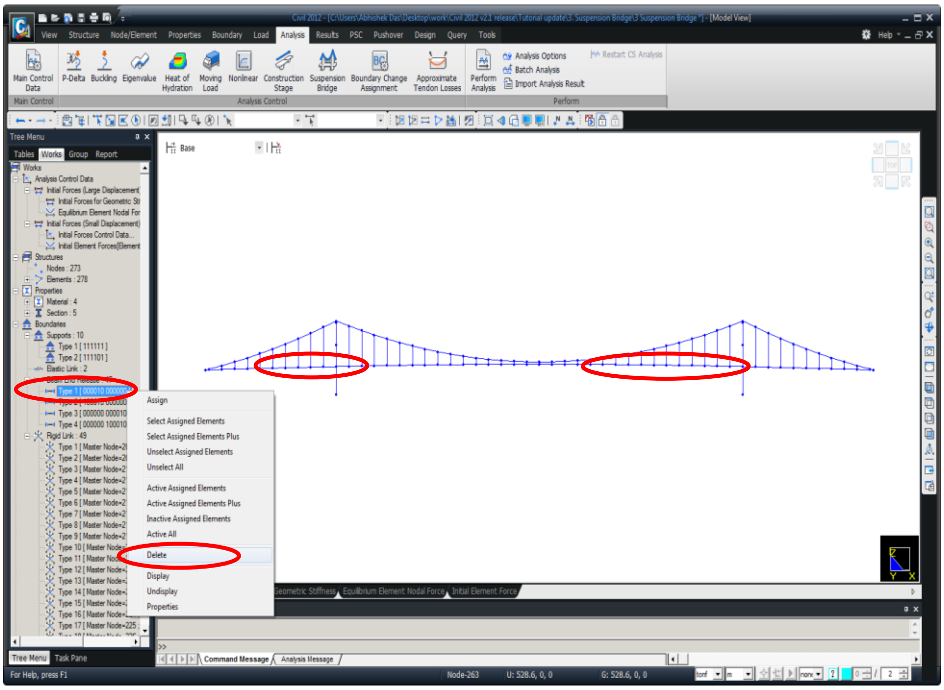


Fig. 31 Delete Beam End Release

The pylons and decks are connected to carry out the completed state analysis. So we remove the Beam End Release conditions for the decks.

Tree Menu>**Works tab**

Boundaries>Beam End Release>Type 1 : **Properties** ↵

My (i-Node) (off) ↵

Boundaries>Beam End Release>Type 2 : **Properties** ↵

My (j-Node) (off) ↵

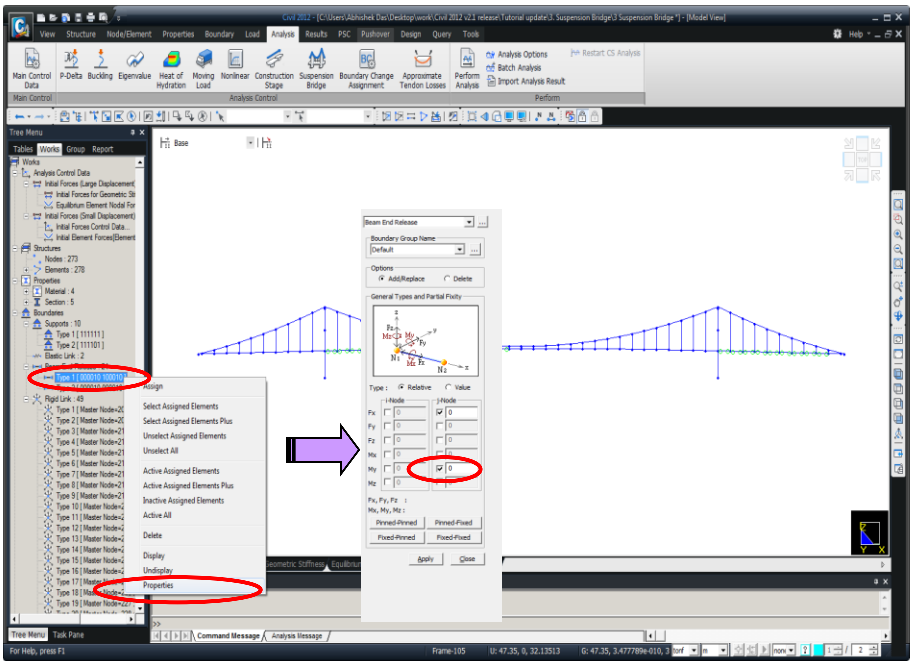


Fig. 32 Modify Beam End Release

Input center span stay

At the center part of the center span, we model the center stay, which will equalize the movement of the girders and the main cable in the axis of the bridge. The structural type of the center stay is normally a center diagonal stay type or a linking type. In this tutorial, we will model the center stay that connects the girders and cables using the Elastic Link function.

Zoom Fit ; ***Zoom Window***
Node Number (on)
Boundary / Elastic Link ***Elastic Link***

Boundary Group Name>**Default**
Options>**Add**
Link Type>General Type>SDz **(1e11)**
2Nodes **(27, 78)**
Copy Elastic Link>Distance>Axis>**y** (on)>Distances **(11)** ↵

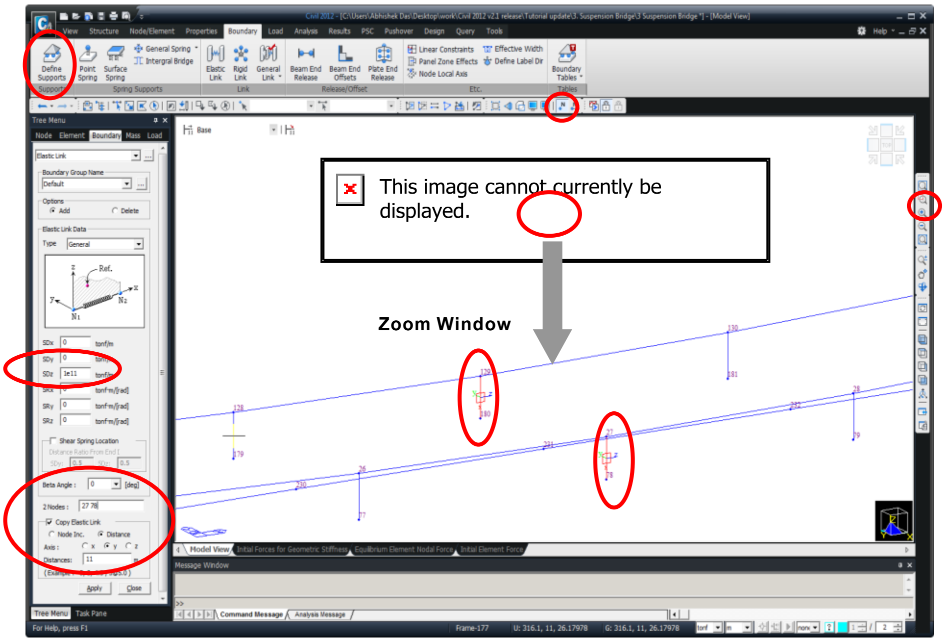


Fig. 33 Connection of Deck (main girders) and cables

Input Load Cases and Static Loads

In order to examine the behavior of the suspension bridge at the stage of the completed state, we assume static vehicle test loading and input the static loads as shown in Fig. 35. We first generate static load cases as shown in Fig. 34.

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

Name **(LC1)** ; Type > **User Defined**
Description **(Static Load Test)** ↵
Name **(LC2)** ; Type >**User Defined**
Description **(Static Load Test)** ↵
Name **(LC3)** ; Type > **User Defined**
Description **(Static Load Test)** ↵

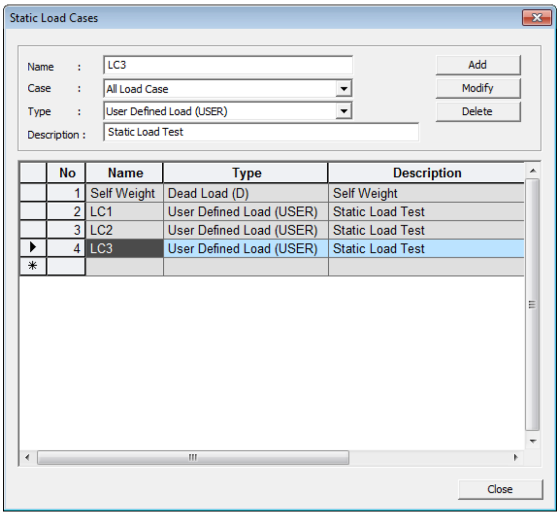


Fig. 34 Define Static Load Cases

Assume the vehicle weight as 46 tonf, and apply the load at three different locations as separate load cases.

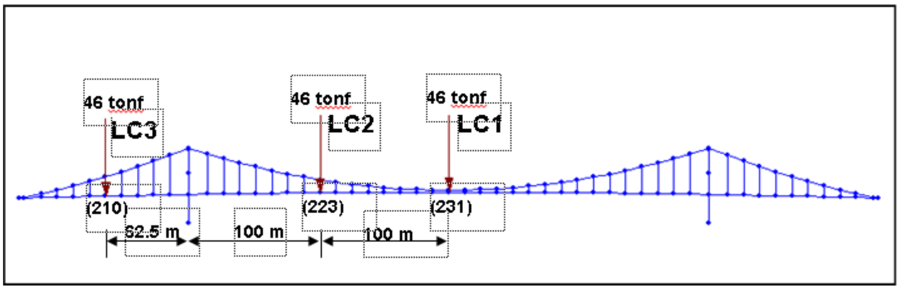


Fig. 35 Static Load cases

Apply static loads to the main girders.

Load / Static Loads / Nodal Loads ***Nodal Loads***

Select Identity-Nodes **(231)**

Load Case Name>**LC1**

Load Group Name>**Default** ; Options>**Add**

Nodal Loads>FZ **(-46)** ↵

Select Identity-Nodes **(223)**

Load Case Name>**LC2** ;

Nodal Loads>FZ **(-46)** ↵

Select Identity-Nodes **(210)**

Load Case Name>**LC3** ;

Nodal Loads>FZ **(-46)** ↵

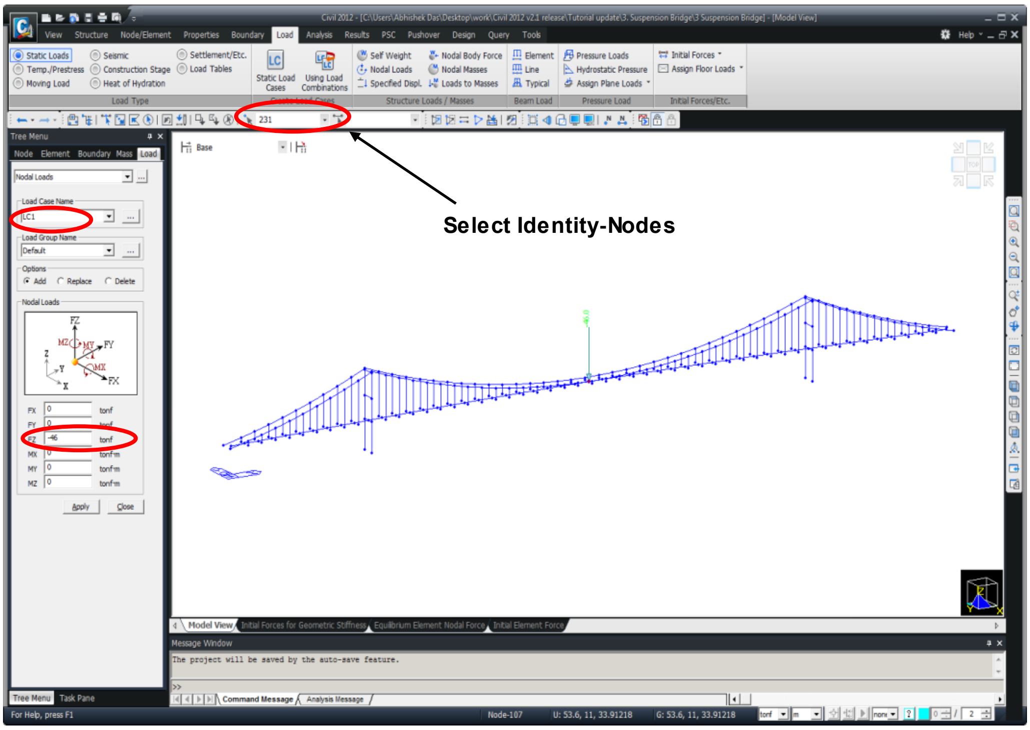
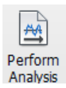


Fig. 36 Input static load (LC1)

Perform Structural Analysis (Completed State Analysis)

We will perform structural analysis as the modeling for the completed state analysis is now completed.

Analysis / 

Perform Analysis

Review Results of Completed State Analysis

Static Analysis Results

Review displacements and member forces for the three static load cases.

Review deformed shape

Front View

Result / Deformations / Deformed Shape...

Load Cases / Combinations > **ST: LC1**

Components> **DXYZ**

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

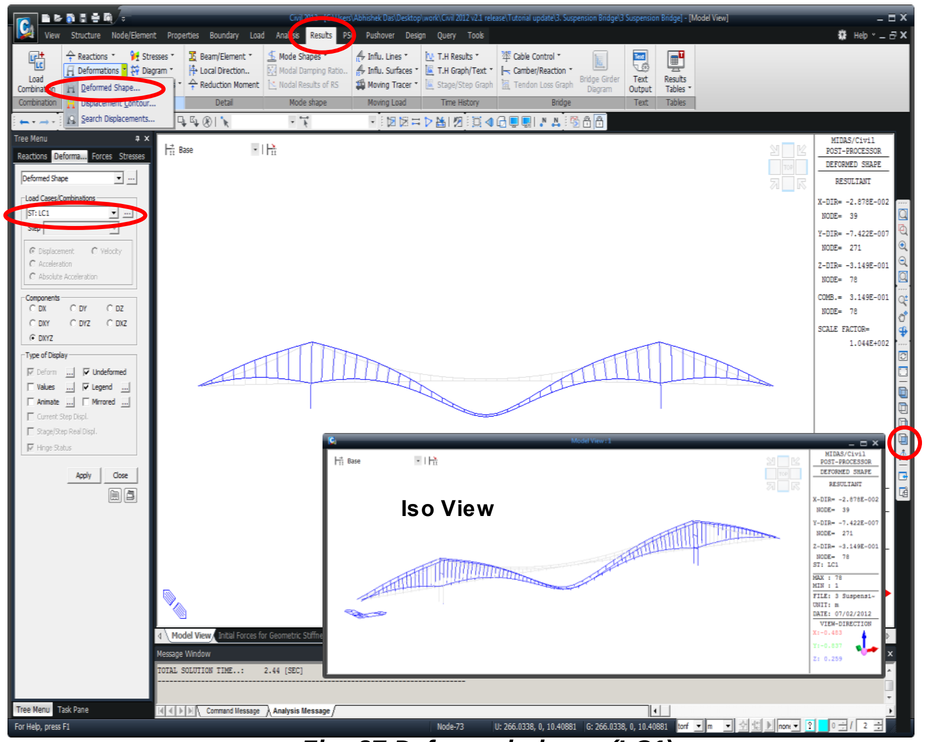


Fig. 37 Deformed shape (LC1)

Review deformed shapes for load cases 2 & 3 using the same procedure.

Result / Deformations / Deformed Shape...

Load Cases / Combinations > **ST: LC2**

Components> **DXYZ**

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

Load Cases / Combinations > **ST: LC3**

Components> **DXYZ**

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

Review displacements in a tabular format at the loading locations.

Results / Results Tables Result Tables / ***Displacements***

Records Activation Dialog>Node or Element>**210 223 231**
Loadcase/Combination> **LC1, LC2, LC3** (on) ↵

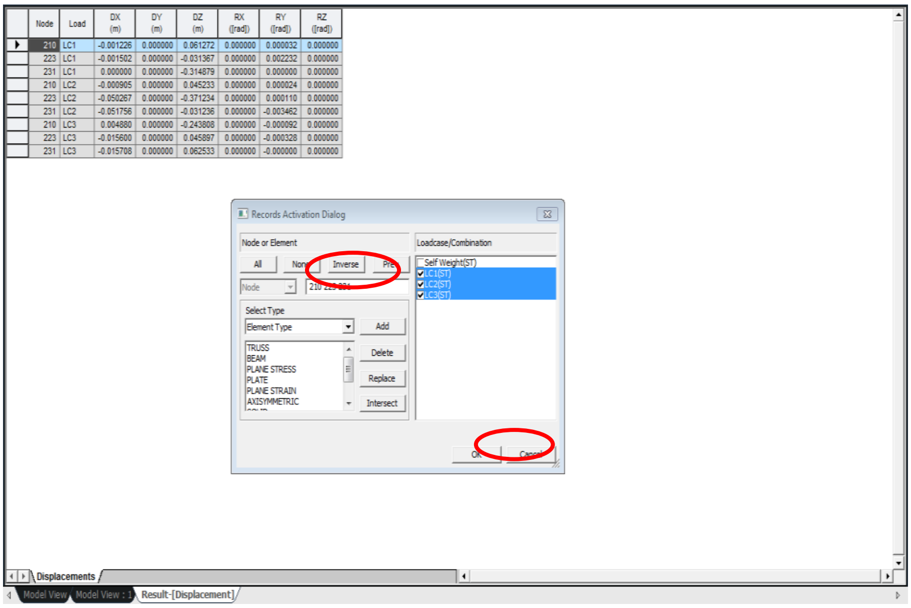


Fig. 38 Displacement table

Review bending moments

Review bending moments in the deck.

Results / Forces / Beam Diagrams...

Load Cases/Combinations>**ST: LC1** ; Components>**My**

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

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

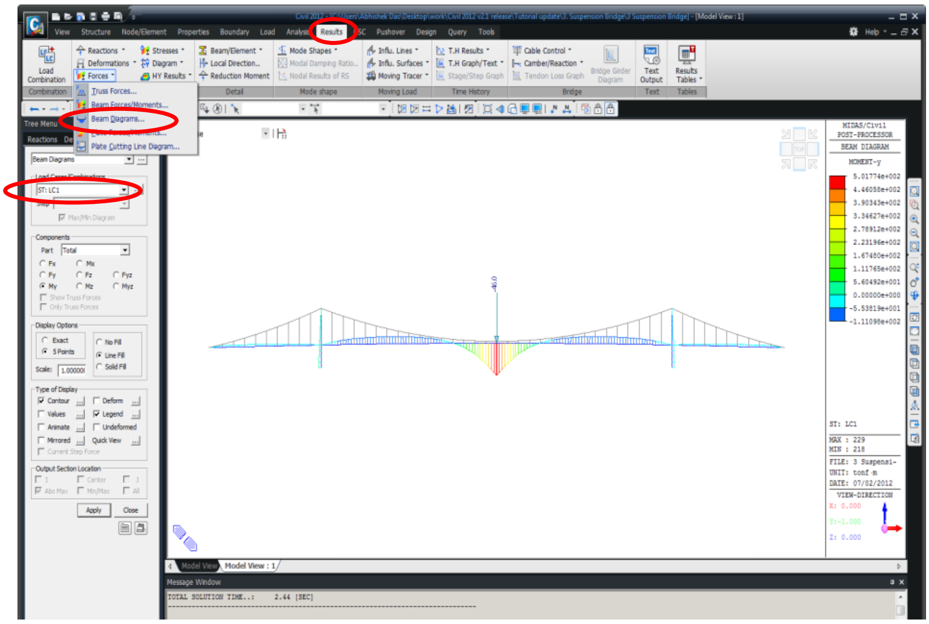


Fig. 39 Bending moment diagram for Deck (LC1)

Review axial forces

Review axial forces in the main cables.

Result / Forces / Truss Forces

Load Cases/Combinations>**ST:LC1**

Force Filter>**All**

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

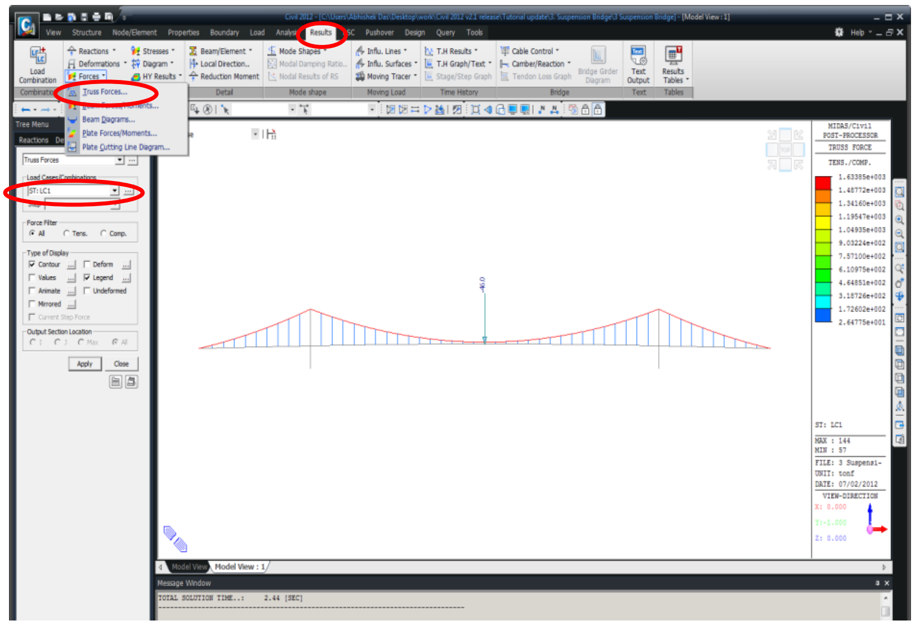


Fig. 40 Max tension forces in the cables (LC1)

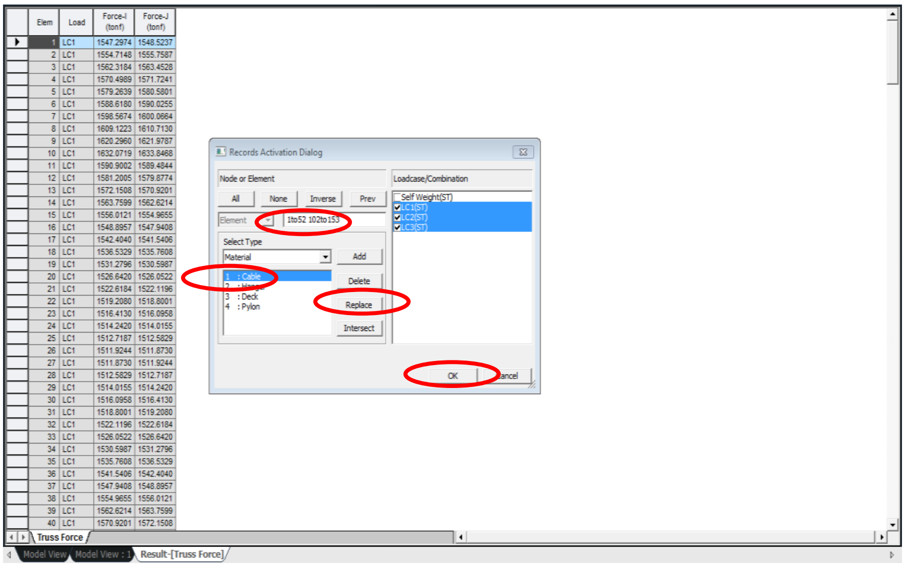
Review the cable axial forces in tabular format.

Results / Results Tables / Result Tables / Truss / ***Force***

Records Activation Dialog>Node or Element>

Select Type>Material>**1: Cable** Replace

Loadcase/Combination> **LC1, LC2, LC3** (on) ↵



Elem	Load	Force-I (tonf)	Force-J (tonf)
1	LC1	1547.2974	1548.5237
2	LC1	1554.7148	1555.7587
3	LC1	1562.3184	1563.4528
4	LC1	1570.4989	1571.7241
5	LC1	1579.2639	1580.5801
6	LC1	1588.6180	1590.0255
7	LC1	1598.5674	1600.0664
8	LC1	1609.1223	1610.7130
9	LC1	1620.2960	1621.9787
10	LC1	1632.0719	1633.8468
11	LC1	1590.9002	1589.4844
12	LC1	1581.2005	1579.8774
13	LC1	1572.1508	1570.9201
14	LC1	1563.7599	1562.6214
15	LC1	1556.0121	1554.9655
16	LC1	1548.8957	1547.9408
17	LC1	1542.4040	1541.5406
18	LC1	1536.5329	1535.7608
19	LC1	1531.2796	1530.5987
20	LC1	1526.6420	1526.0522
21	LC1	1522.6184	1522.1196
22	LC1	1519.2080	1518.8001
23	LC1	1516.4130	1516.0958
24	LC1	1514.2420	1514.0155
25	LC1	1512.7187	1512.5829
26	LC1	1511.9244	1511.8730
27	LC1	1511.8730	1511.9244
28	LC1	1512.5829	1512.7187
29	LC1	1514.0155	1514.2420
30	LC1	1516.0958	1516.4130
31	LC1	1518.8001	1519.2080
32	LC1	1522.1196	1522.6184
33	LC1	1526.0522	1526.6420
34	LC1	1530.5987	1531.2796
35	LC1	1535.7608	1536.5329
36	LC1	1541.5406	1542.4040
37	LC1	1547.9408	1548.8957
38	LC1	1554.9655	1556.0121
39	LC1	1562.6214	1563.7599
40	LC1	1570.9201	1572.1508

Fig. 41 Table of tension forces in the main cables

* The above output of axial forces shows the additional axial force in the cables. At the initial equilibrium state, tension forces due to the self weight have already occurred. Therefore, the total member forces in the cables and hangers then become the summation of the above axial forces and the ***Initial Force for Geometric Stiffness*** introduced during preprocessing.

The following procedure will generate the total axial forces, which include both the initial forces and additional forces determined previously.

Load / Initial Forces /Small Displacement/ Initial Forces Control Data

Add Initial Force to Element Force (on)

Load Case > **LC1** ↵

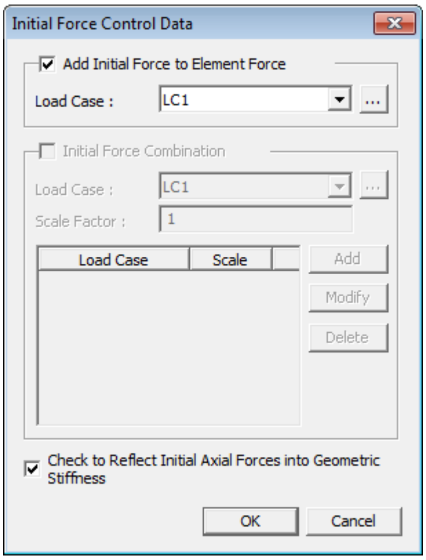
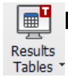


Fig. 42. Initial Force Control Data

Analysis / Perform Analysis ***Perform Analysis***

Review the cable axial forces in the tabular format.

Results / 

Result Tables / Truss / ***Force***

Records Activation Dialog>Node or Element>

Select Type>Material>**1: Cable** [Replace]

Loadcase/Combination> **LC1** (on) ↵

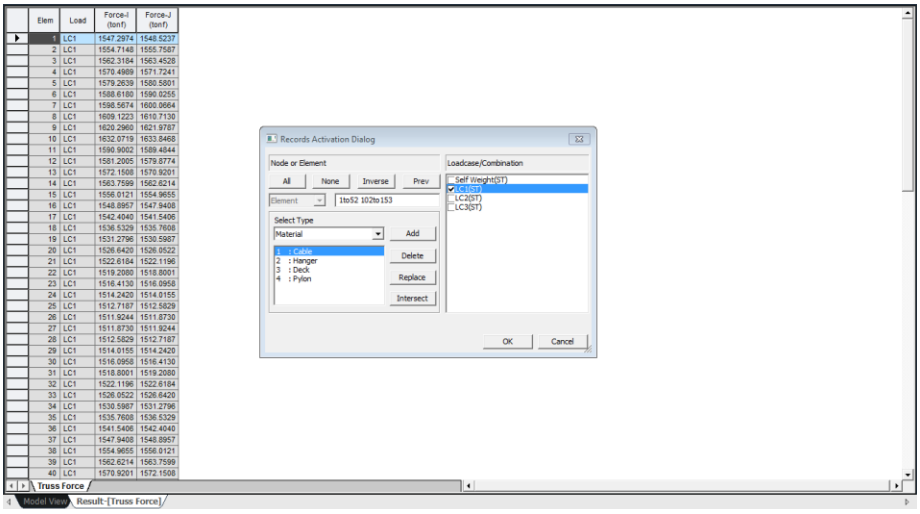


Fig. 43 Sum of initial forces and additional forces in cable

Modeling for Construction Stage Analysis

A suspension bridge is relatively unstable during construction compared with the completed state. Therefore, geometric nonlinear analysis (large displacement analysis) must be performed instead of linearized finite displacement analysis or P-Delta analysis. Moreover, construction sequence analysis is warranted to reflect the forces and displacements of previous stages in the subsequent stages.

In this chapter, we will perform a backward construction stage analysis for the construction of a suspension bridge starting from the completed state analysis model that was created earlier. The backward analysis sequence used in this tutorial is shown in Fig. 44.

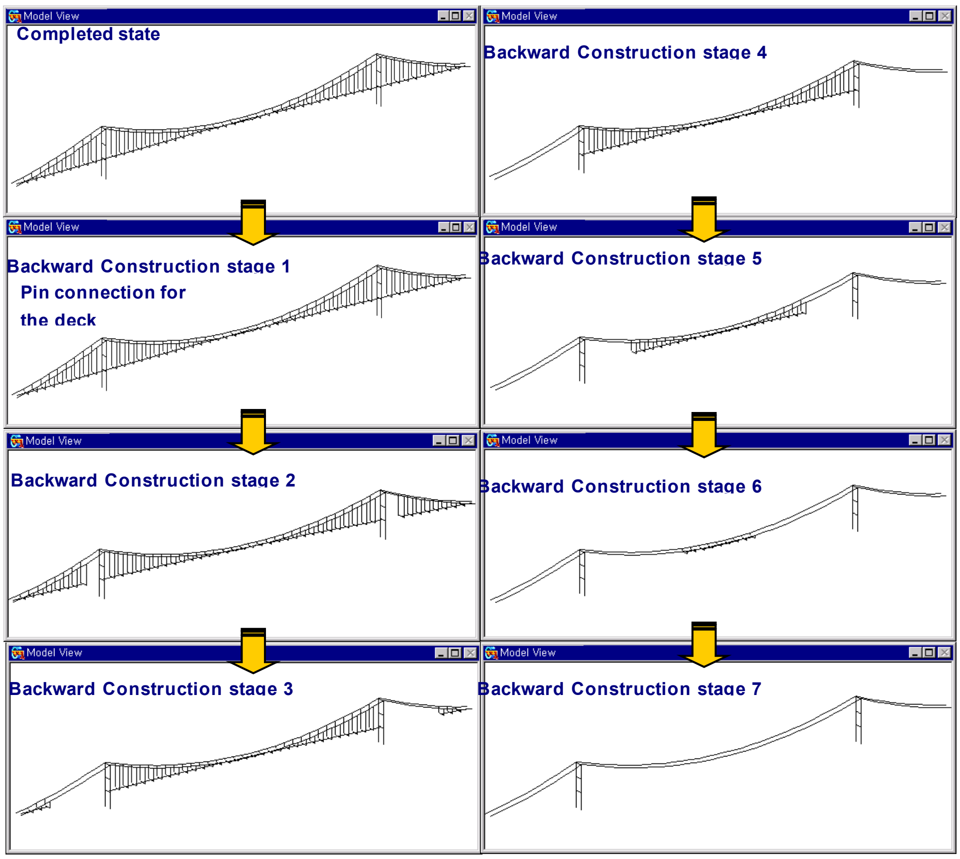


Fig. 44 Sequence of backward construction stage analysis

Assign Working Environment

To generate a construction stage analysis model using the final stage analysis model, we first save the completed state analysis model data under a different file name.

Save As **(Suspension Bridge Construction.mcb)**

To generate a construction stage analysis model, the following should be added to the completed state analysis model.

Modeling

- Define construction stages
  Define elements, boundary conditions and loadings pertaining to each construction stage.
- Define Structure groups
  Group elements that are added / deleted at each construction stage.
- Define Boundary groups
  Group boundary conditions that are added / deleted at each construction stage.
- Define Load groups
  Group loads that are added / deleted at each construction stage.

Analysis

- Nonlinear Analysis (geometric nonlinear analysis)
- Construction Stage Analysis

Define Construction Stage Names

Define construction stages for backward construction stage analysis. First, define all the names to be used for the construction stages by using the Construction Stage dialog box. Then, define Structure Groups, Boundary Groups and Load Groups pertaining to each construction stage, and assign each group to a corresponding construction stage.

In this tutorial, there are eight construction stages defined including the completed state as shown in Fig. 45.

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

Construction Stage> Generate

Stage>Name **(CS)** ; Suffix **(0to7)**

Save Result>**Stage** (on) ↵

Define the names of the construction stages using the same prefix and different suffix numbers.

When generating output for each construction stage, the output for each construction stage is saved and produced.

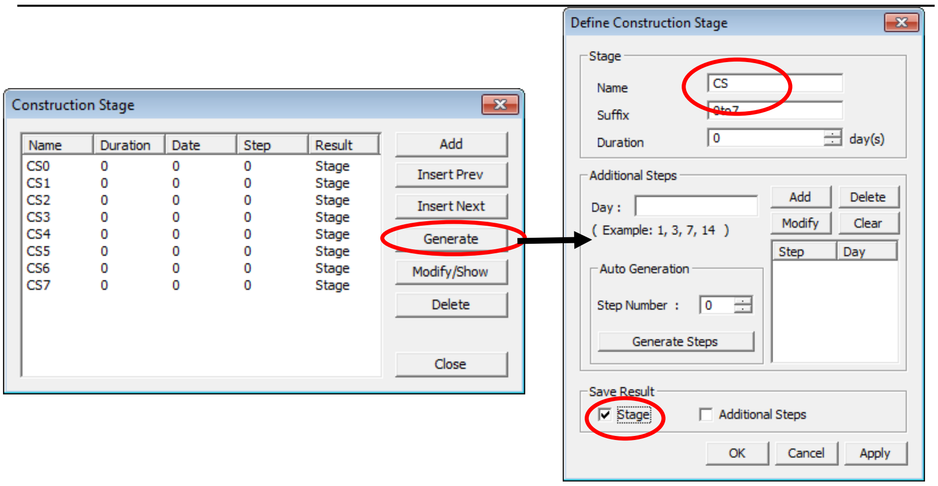


Fig. 45 Define the construction stage names using the Construction Stage dialog box

Assign Structure Groups

Assign elements, which are added or deleted in each construction stage, to the Structure Groups. First, create the name of each Structure Group, and then assign the corresponding elements.

Tree Menu>**Group tab**

Group>Structure Group> **New…** ***(right-click on Structure Group)***

Name **(S_G)** ; Suffix **(0, 2to7)** ↵

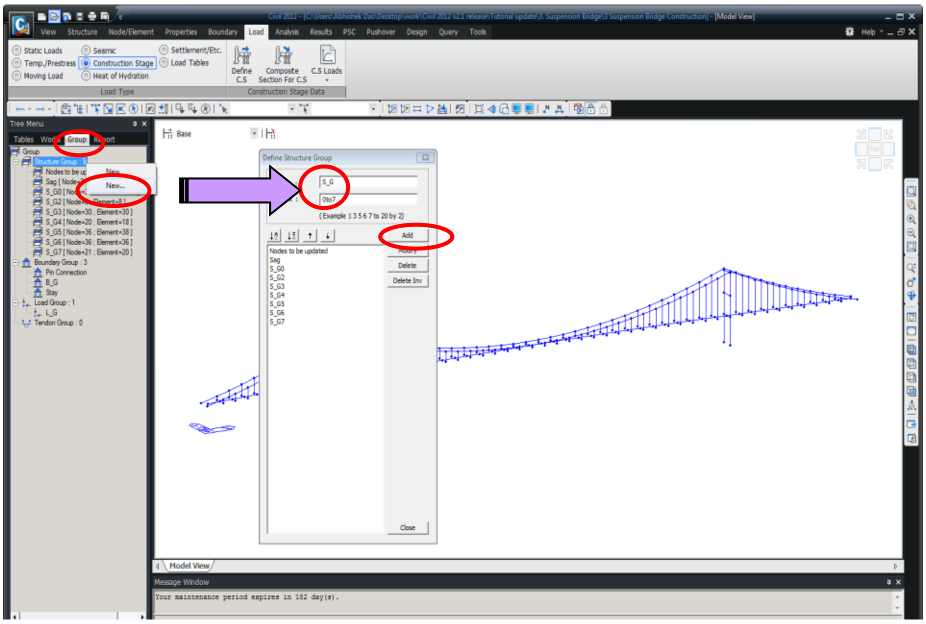


Fig. 46 Define Structure Groups

Assign elements, which are added/deleted in each construction stage to a corresponding Structure Group. At the completed state - final stage (CS0) and the stage in which the deck is pin connected (CS1), the corresponding elements are identical, and only the boundary condition is changed. Therefore, we will define the construction stage as Structure Group S_G0.

Tree Menu>**Group tab**

Select All

Group>Structure Group>**S_G0 (Drag & Drop)**

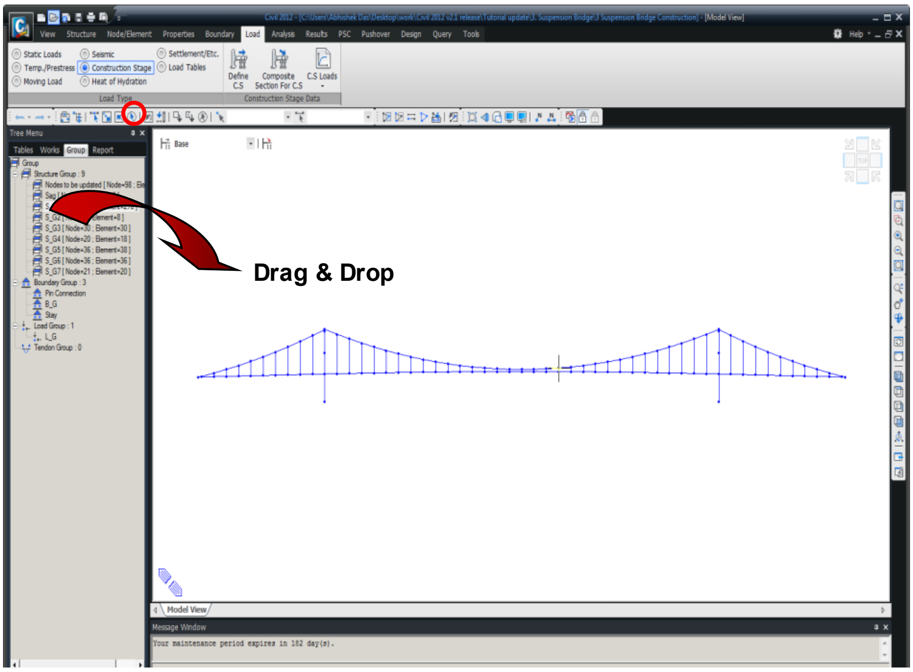


Fig. 47 Define Structure Group (S_G0)

Define the deck and hangers, which are deleted in the backward construction stage CS2, as Structure Group S_G2.

When selecting elements, all elements intersected by the selection window can be selected if the selection window is created from right to left.

To define the structure group precisely, inactivate the previously defined element group to prevent it from being selected in another element group.

Tree Menu>**Group tab**

Select Window (Elements: **Fig. 48** ①, ②)

Group>Structure Group>**S_G2** (**_Drag & Drop_**)

S_G2>**Inactivate**

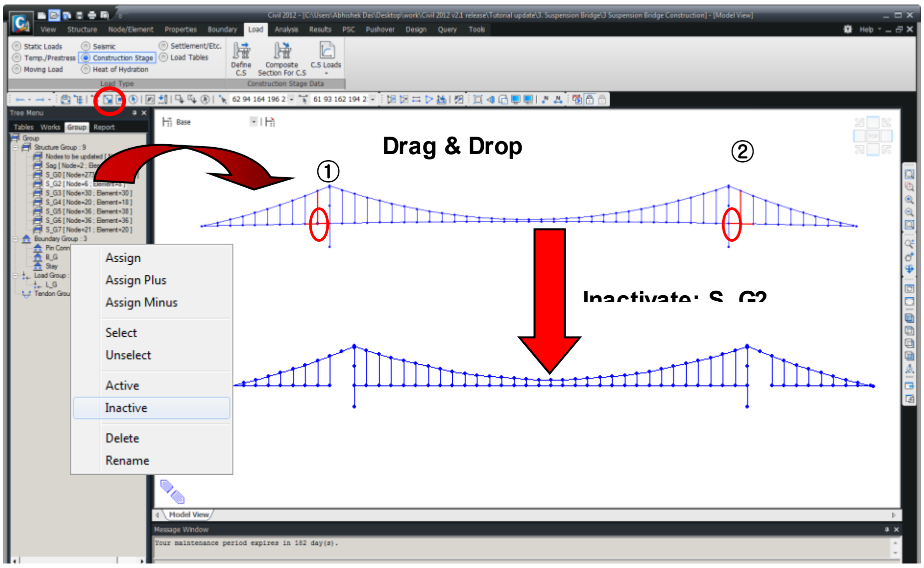


Fig. 48 Define Structure Group (S_G2)

Define the deck and hangers, which are deleted in the backward construction stage CS3, as Structure Group S_G3.

Tree Menu>**Group tab**

Select Window (Elements: **Fig. 49** ①, ②)

Group>Structure Group>**S_G3** (***Drag & Drop***)

S_G3>**Inactivate**

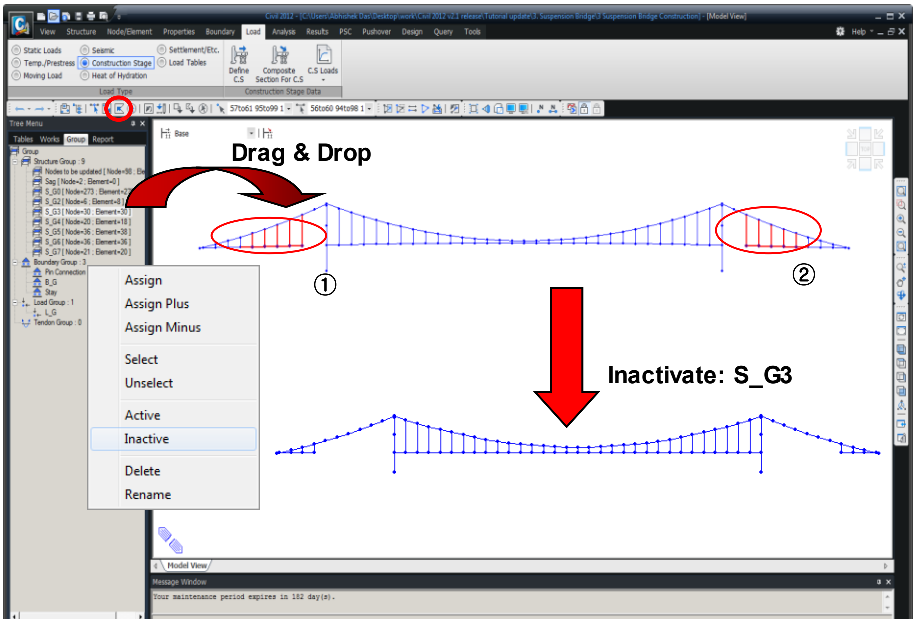


Fig. 49 Define Structure Group (S_G3)

Define the deck and hangers, which are deleted in the backward construction stage CS4, as Structure Group S_G4.

Tree Menu>**Group tab**

Zoom Window (**Fig. 50** ①)

Select Window (Elements: **Fig. 50** ②)

Zoom Fit

Zoom Window (**Fig. 50** ③)

Select Window (Elements: **Fig. 50** ④)

Group>Structure Group>**S_G4** (***Drag & Drop***)

S_G4>**Inactivate**

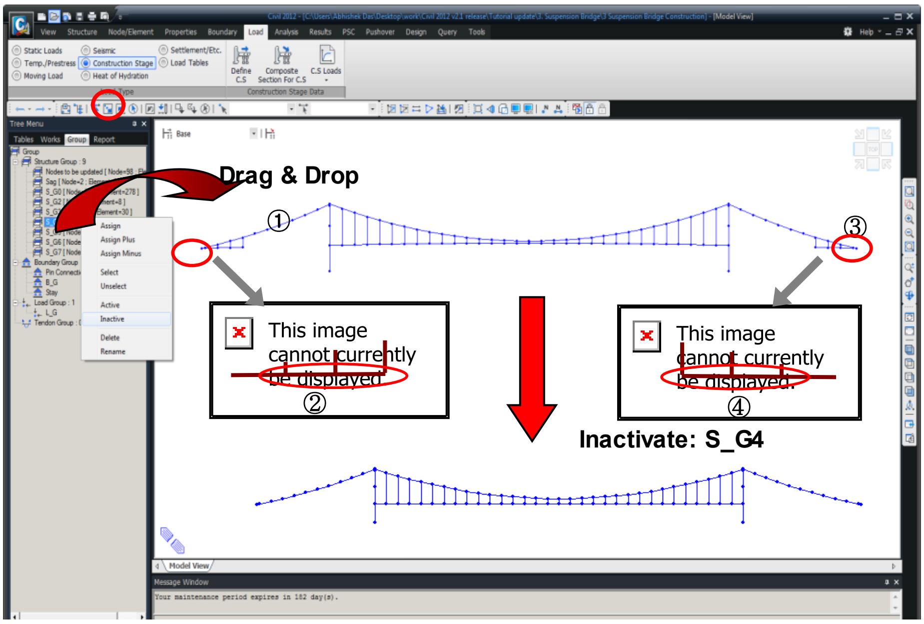


Fig. 50 Define Structure Group (S_G4)

Define the deck and hangers, which are deleted in the backward construction stage CS5, as Structure Group S_G5.

Tree Menu>**Group tab**

Select Window (Elements: **Fig. 51** ①, ②)

Group>Structure Group>**S_G5** (***Drag & Drop***)

S_G5>**Inactivate**

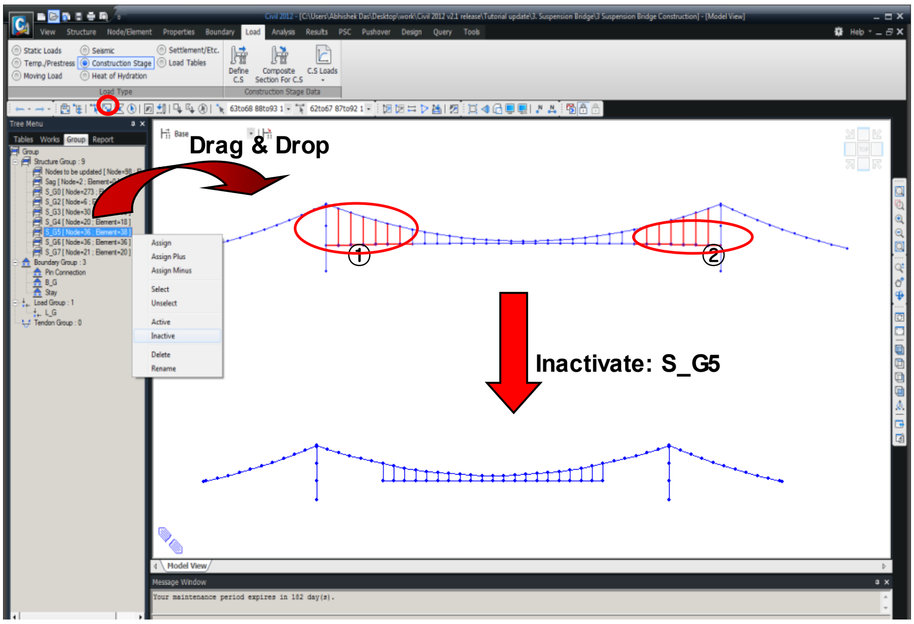


Fig. 51 Define Structure Group (S_G5)

Define the deck and hangers, which are deleted in the backward construction stage CS6, as Structure Group S_G6.

Tree Menu>**Group tab**

Select Window (Elements: **Fig. 52** ①, ②)

Group>Structure Group>**S_G6** (***Drag & Drop)***

S_G6>**Inactivate**

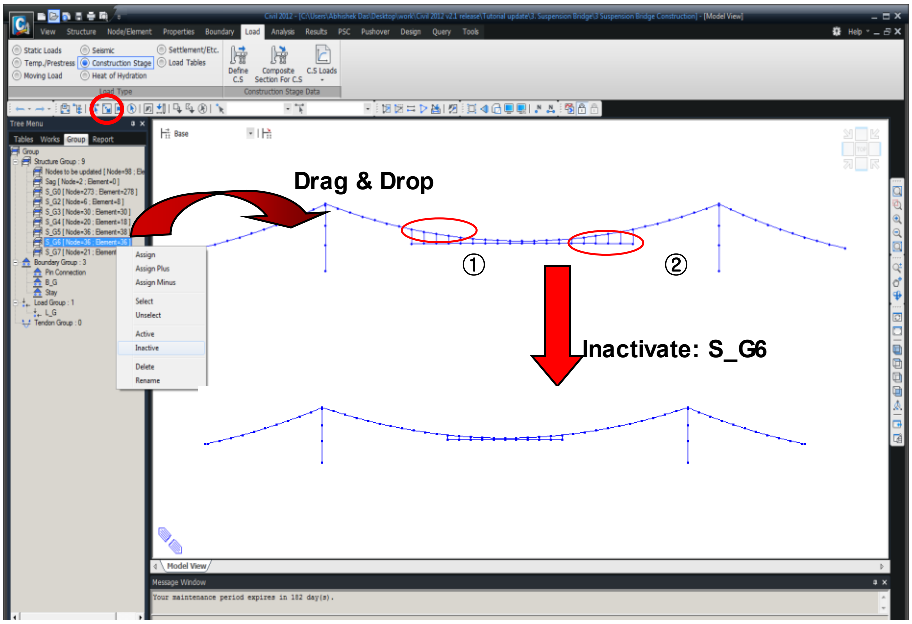


Fig. 52 Define Structure Group (S_G6)

Define the deck and hangers, which are deleted in the backward construction stage CS7, as Structure Group S_G7.

Tree Menu>**Group tab**

Zoom Window (Fig. **53** ①)

Select Window (Elements: **Fig. 53** ①)

Group>Structure Group>**S_G7** (***Drag & Drop***)

S_G7>**Inactivate**

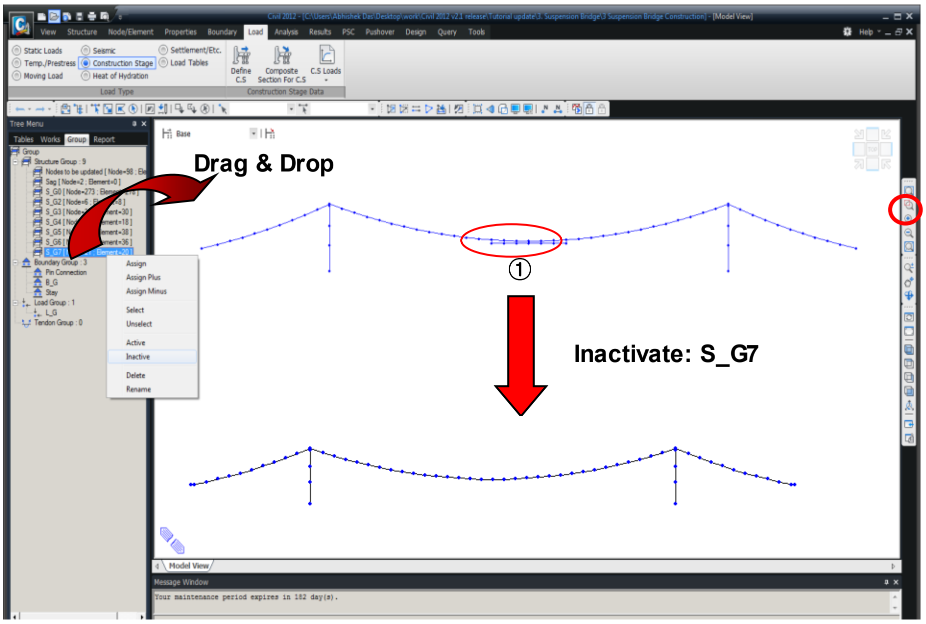


Fig. 53 Define Structure Group (S_G7)

Assign Boundary Groups

Assign boundary conditions for each construction stage to Boundary Groups. First, we generate the name of each Boundary Group, and assign boundary conditions for each construction stage to a corresponding Boundary Group already generated.

Activate All

Group tab>Boundary Group>**New…** ***(right-click on Boundary Group)***

Name **(B_G)** ↵

Name **(Stay)** ↵

Name > **(Pin Connection)** already exists ↵

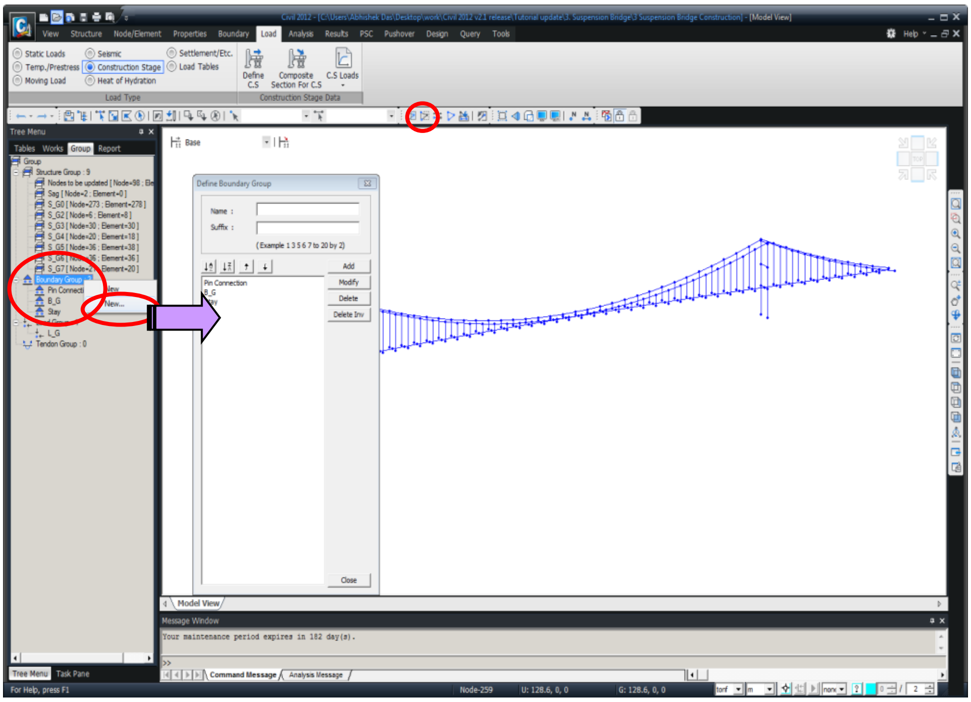


Fig. 54 Create Boundary Group Names

We now group boundary conditions for the pylons, cable anchorages and deck ends. Using the Drag & Drop function, we change the boundary condition group name (Default) already defined to B_G.

Select All

Tree Menu>**Group tab**

Group>Boundary Group>B_G (***Drag & Drop***)

Select Boundary Type ↵

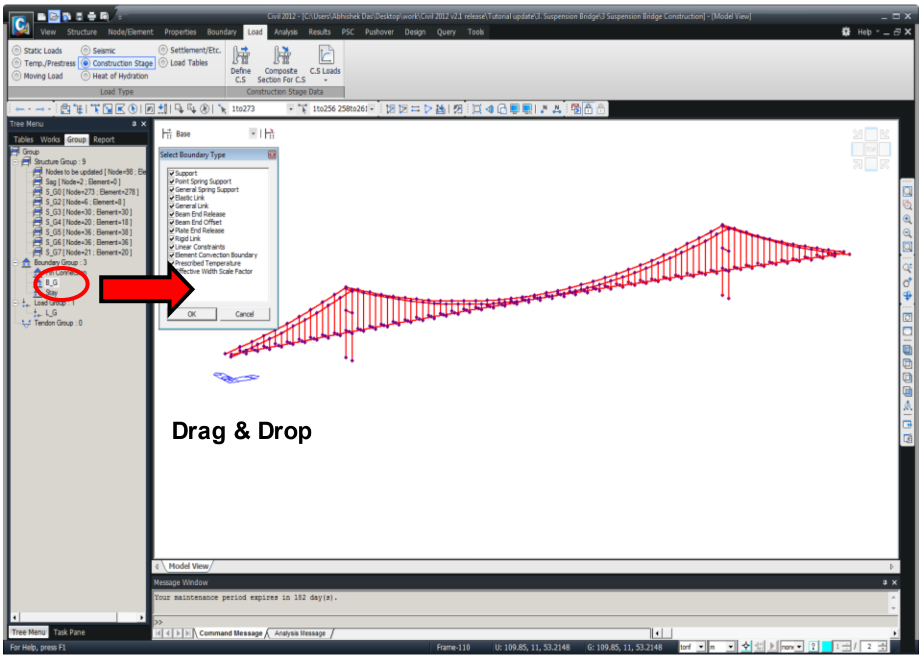


Fig. 55 Changing boundary condition group name

Grouping center span stay

Assign the center span stay, modeled by Elastic Link, as a Boundary Group named "**Stay**".

.

Tree Menu>**Group tab**

Select All

Boundary Group>**Stay** (***Drag & Drop***)

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

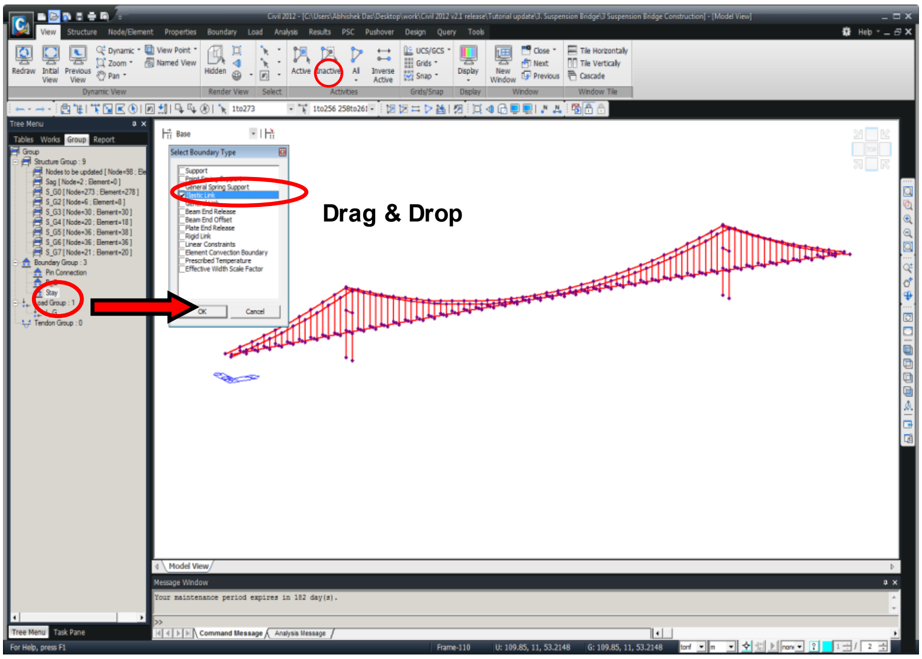


Fig. 56 Grouping center span stay

Pin connections at deck

We assign hinge conditions to the deck in the same way as we did for the completed state analysis. We specify ***Beam End Release*** about moment My at the i-end of the deck in the parts ① & ② of Fig. 57 and assign them to the boundary group, "Pin Connection", at the same time.

Boundary / Beam End Release ***Beam End Release***

Boundary Group Name>**Pin Connection**

Options>**Add/Replace**

Select Window (Elements: **Fig. 57** ①, ②)

(Elements: **204to211 230to243**)

General Types and Partial Fixity>**My (i-Node)** (on) ↵

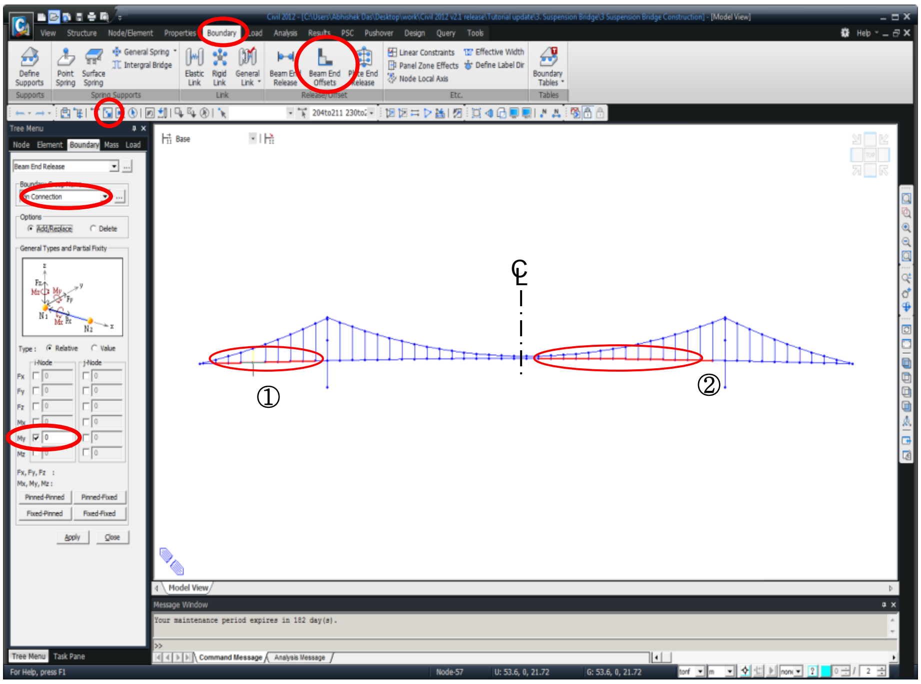


Fig 57 Define Pin Connections at deck

We specify ***Beam End Release*** about moment My at the j-end of the deck in the parts ① & ② of Fig. 58 and assign them to the boundary group, “Pin Connection”, at the same time.

Boundary / Beam End Release ***Beam End Release***

Boundary Group Name> **Pin Connection**

Options> **Add/Replace**

Select Window (Elements: **Fig. 58** ①, ②)

(Elements: **214to228 246to253**)

General Types and Partial Fixity> **My (j-Node)** (on) ↵

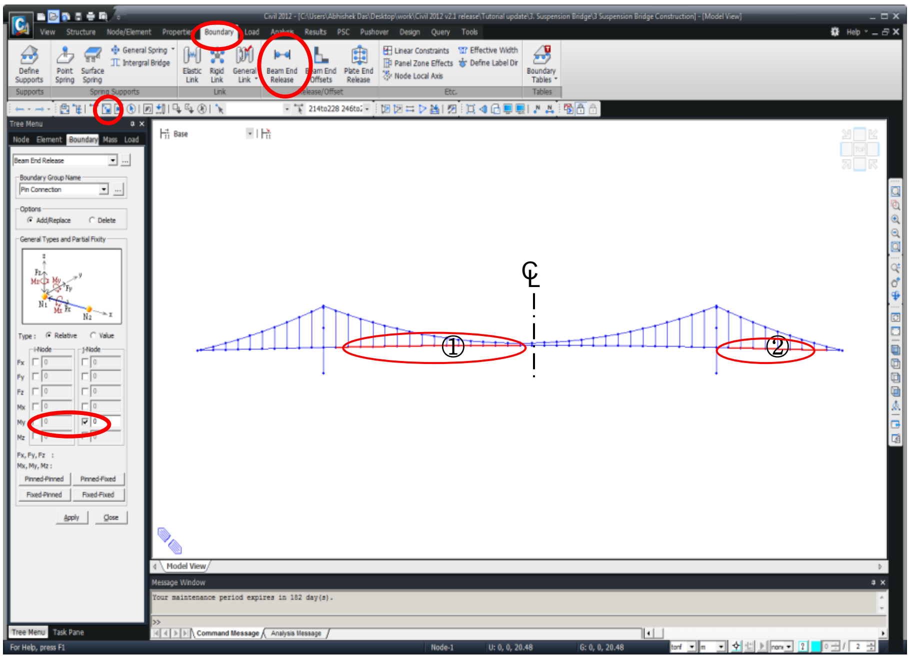


Fig 58 Define Pin Connections at deck

Define Construction Stage Loads and Load Groups

We will remove the loads used in the completed state analysis since they are not used in construction stage analysis. Since the loads in construction stage analysis were not defined in the completed state analysis, we will define the loads for the construction stages and define the Load Groups simultaneously.

The load type that is applied to construction stage analysis must be selected as Construction Stage Load.

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

No 2~4 **(LC1, LC2, LC3)** selected Delete

No 1 **(Self Weight)** selected

Type **(Construction Stage Load)** Modify

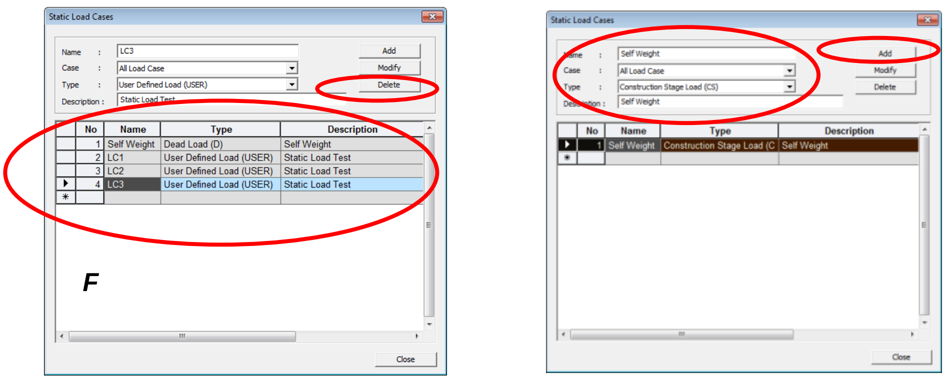


Fig. 59 Define construction stage static load

We will use the Load Group (L_G) already defined for the completed state analysis. When elements are eliminated in construction stages, the self weights of those elements are also eliminated. In construction stage analysis, the Equilibrium Element Nodal Forces calculated in the process of the completed state analysis are applied to the member internal forces. Therefore, the construction stage process is modeled such that only the deck and hanger elements are eliminated in each stage and their internal forces are redistributed 100% to the contiguous elements.

Define Construction Stages

Assign the previously defined structure groups, boundary groups and load groups to the corresponding stages. Table 4 shows the elements, boundary conditions and load groups that are activated or deactivated in each construction stage.

Table 4 Element, boundary condition and load group for each construction stage

Stage	Structure Group		Boundary Group		Load Group	
	Activate	Deactivate	Activate	Deactivate	Activate	Deactivate
CS0	S_G0		B_G, Stay		L_G	
CS1			Pin Connection	Stay		
CS2		S_G2				
CS3		S_G3				
CS4		S_G4				
CS5		S_G5				
CS6		S_G6				
CS7		S_G7				

CS0: Completed state (final stage)
CS1: just before the decks (main girders) are rigidly connected (pin connection stage)
CS2 ~ CS7: construction stages in which the decks (main girders) and hangers are erected (refer to Fig. 43)

Define the construction stage CS0 (Completed state stage)

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

Name>**CS0** Modify/Show

Save Result>**Stage** (on)

Element>Group List>**S_G0** ; Activation> Add

Boundary> Group List> **B_G, Stay** ; Activation> Add

Load>Group List>**L_G** ; Activation> Add ↵

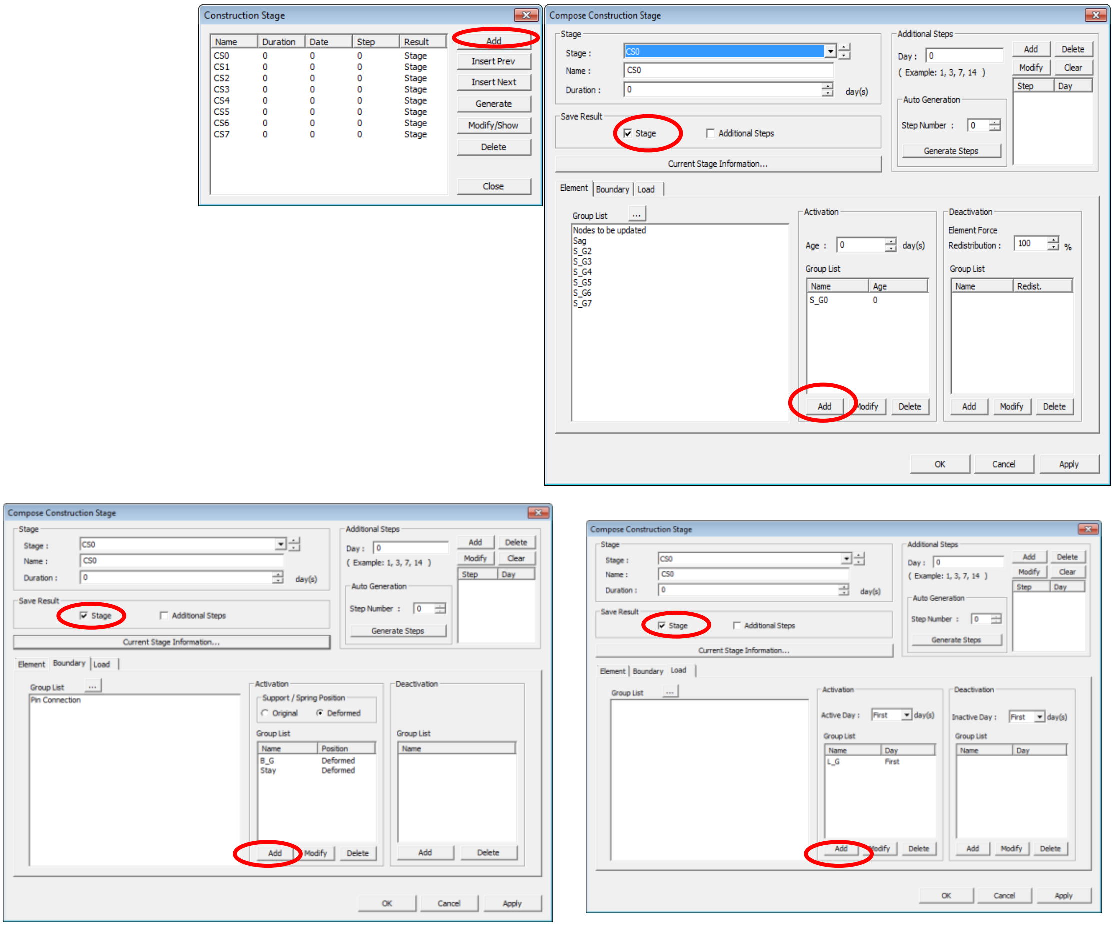


Fig. 60 Define construction stage CS0

Define Construction Stage CS1 (Pin Connection stage).

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

Name>**CS1** Modify/Show

Save Result>**Stage** (on)

Boundary> Group List> **Pin Connection**; Activation> Add

Boundary> Group List> **Stay** ; Deactivation> Add ↵

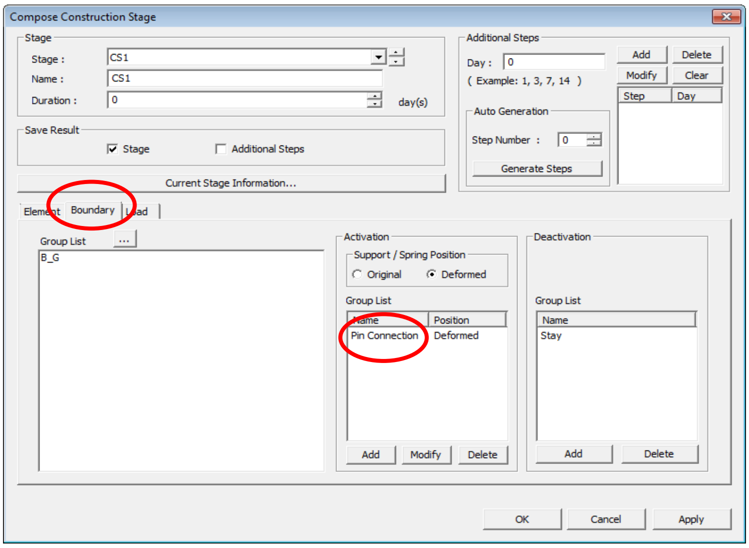


Fig. 61 Define Construction Stage CS1

Define Construction Stage CS2.

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

Name>**CS2** Modify/Show

Save Result>**Stage** (on)

Element> Group List> **S_G2** ;

Element Force Redistribution 100% ;

Deactivation> Add

When elements are deactivated, a percentage of the internal forces of the elements being deactivated is redistributed to contiguous elements.

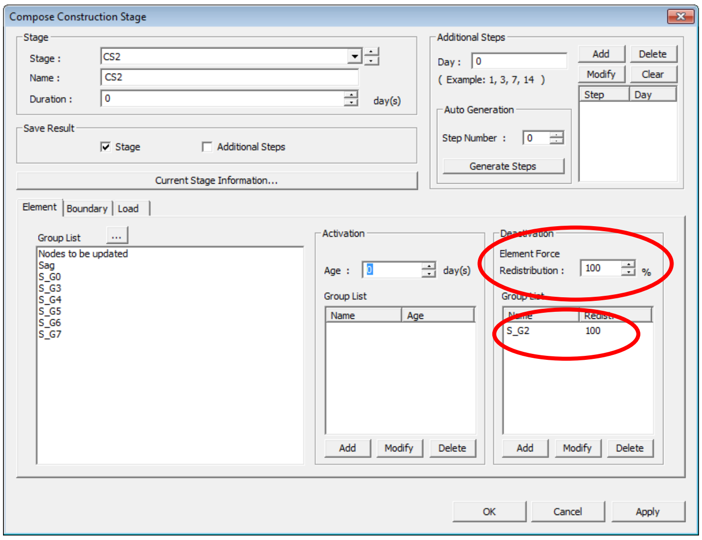
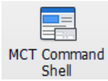


Fig. 62 Define Construction Stage CS2

For efficiency, we will use the ***MCT Command Shell*** even though the remaining construction stages (CS3-CS7) can be defined using the same procedure as above. Repetitive input such as defining the construction stages can be easily input using the ***MCT Command Shell***. The techniques used to input the construction stage information by the ***MCT Command Shell*** is as follows:

Tools> 

Command or Data (***STAGE**) 

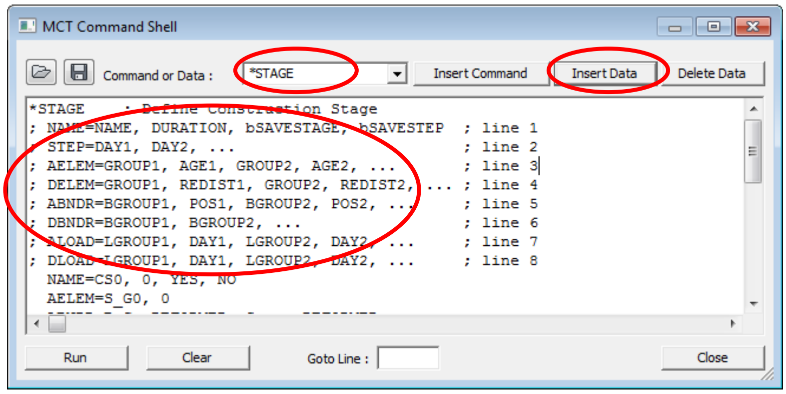


Fig. 63 MCT Command Shell

As shown in Fig. 62, the construction stage information comprises eight lines of commands. Each command is defined below.

NAME: construction stage name, number of days of construction for the stage, flag for saving output
STEP: time Step
AELEM: activated structure group and its initial age
DELEM: deactivated structure group and its internal force redistribution factor for its section forces
ABNDR: activated boundary group and location
DBNDR: deactivated boundary group
ALOAD: activated load group and time step
DLOAD: deactivated load group and time step

Modify the information for the construction stages CS3-CS7 using the ***MCT Command Shell*** as shown in Fig. 64.

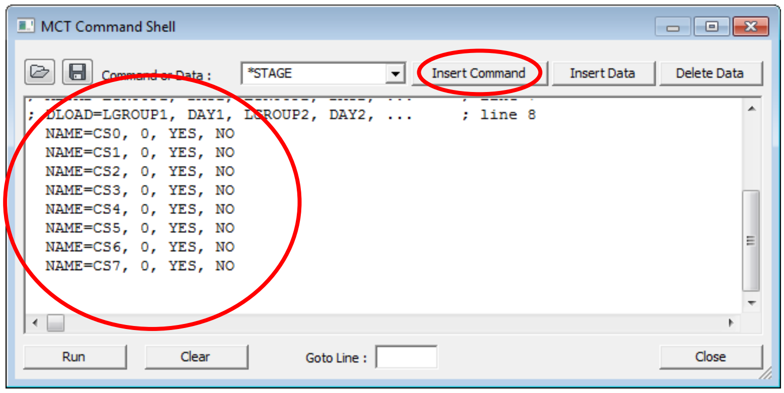
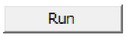


Fig. 64 MCT Command Shell

After input has been completed in the MCT Command Shell, we then simply click the 

button to compose the construction stages with the following messages generated.

Confirm whether the construction stages have been correctly defined or not on the Model View.

Display

Boundary tab

Support (on) ; **Beam End Release Symbol** (on)

Rigid Link (on) ; **Elastic Link** (on)

Load tab

Load Case>**All** (on) ; **Nodal Load** (on) ↵

Stage>**CS3**

Construction stages can be easily viewed on the Model View by simply selecting the construction stages using the direction key on the keyboard, if the Stage Toolbar is activated.

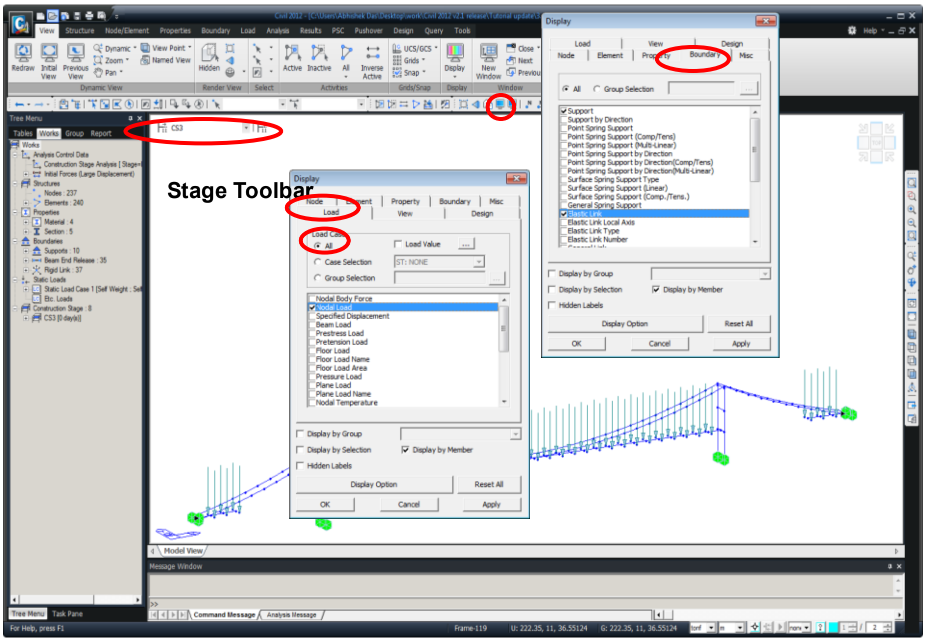


Fig. 65 Check the defined construction stage (CS3) shown on Model view

Input Construction Stage Analysis Data

Select the Last Stage and analysis type for the construction stage analysis. Select the large displacement analysis option, as the effect of large displacements cannot be ignored when calculating forces for construction of a suspension bridge. Nonlinear construction stage analysis is carried out while reflecting the Equilibrium Element Nodal Forces calculated in the completed state analysis.

Stage>**Base**

Analysis / 

Construction Stage Analysis Control

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

Analysis Option> **Include Nonlinear Analysis** (on)

> **Independent Stage** (on)

>**Include Equilibrium Element Nodal Forces** (on)

Convergence Criteria>**Displacement Norm** (on) ↵

Equilibrium Element Nodal Force calculated in the completed state analysis is reflected in the internal member forces.

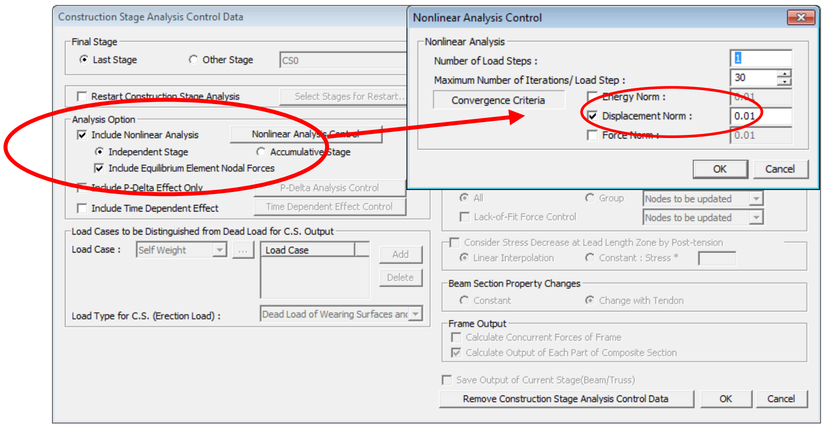
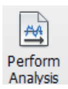


Fig. 66 Construction Stage Analysis Control Data dialog box

Perform Structural Analysis (Construction Stage Analysis)

Now that we have completed the construction stage analysis model, we will perform structural analysis.

Analysis / 

Perform Analysis

Review Construction Stage Analysis Results

Review the change in the deformed shapes and section forces for each construction stage.

Review Deformed Shape

We will examine the global behavior of the structure by checking the deformed shape at each construction stage. The deformed shape at the construction stage CS7, which represents the completed pylons and main cables, is shown in Fig. 66.

If the basic Deformation Scale Factor is too large, adjust the factor to view the deformed shape.

Review the deformed shapes for different construction stages by changing the construction stages by using the Stage Toolbar. Mouse wheel or up/down keys on the keyboard may be used.

Results / Deformations / Deformed Shape...

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

Components> **DXYZ**

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

Deform >Deformation Scale Factor (**0.2**) ↵

Stage Toolbar> **CS7**

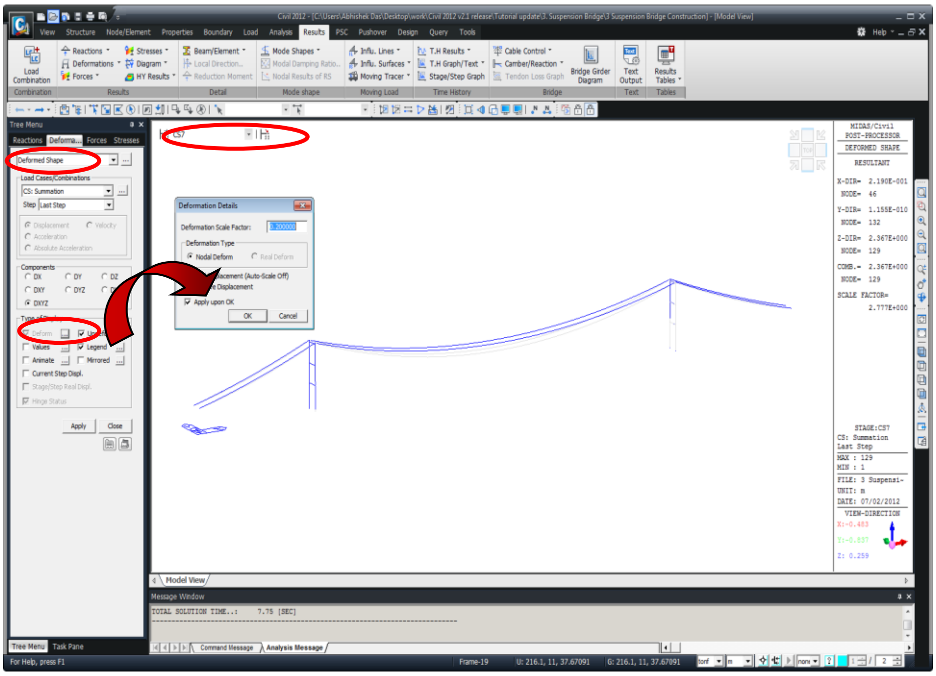


Fig. 67 Check the deformed shapes for different construction stages

Review the change of the center span sag, which is used as a measure of erection precision during construction at each construction stage, by a graph.

Results / Stage/Step Graph ***Stage/Step History Graph***

Define Function>Displacement> Add New Function

Displacement>Name (**Sag**) ; Node Number (**27**)

Components>**DZ** ↵

Mode > **Multi Func.**

Step Option > **Last Step**

X-Axis > **Stage/Step**

Check Functions To Plot > **Sag** (on)

Load Cases/Combinations > **Summation**

Graph Title > **Sag**

Graph ↵

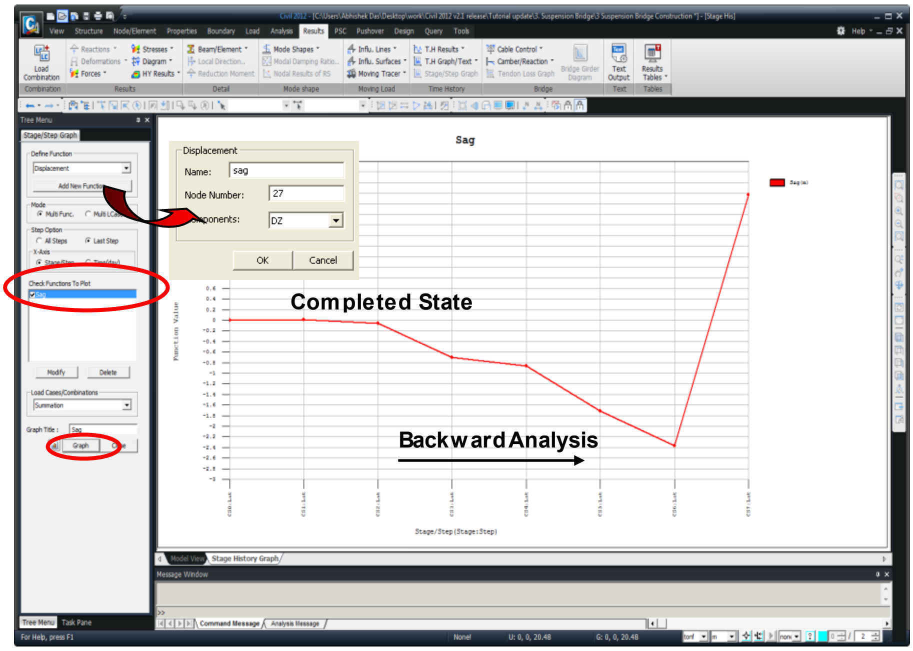


Fig. 68 Changes of the Sag magnitudes through construction stages

Determine a setback

Review the magnitude of a setback for the pylons at the stage when the deck (main girders) and hangers have not been erected.

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

Stage Toolbar>**CS7**

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

Components>**DX**

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

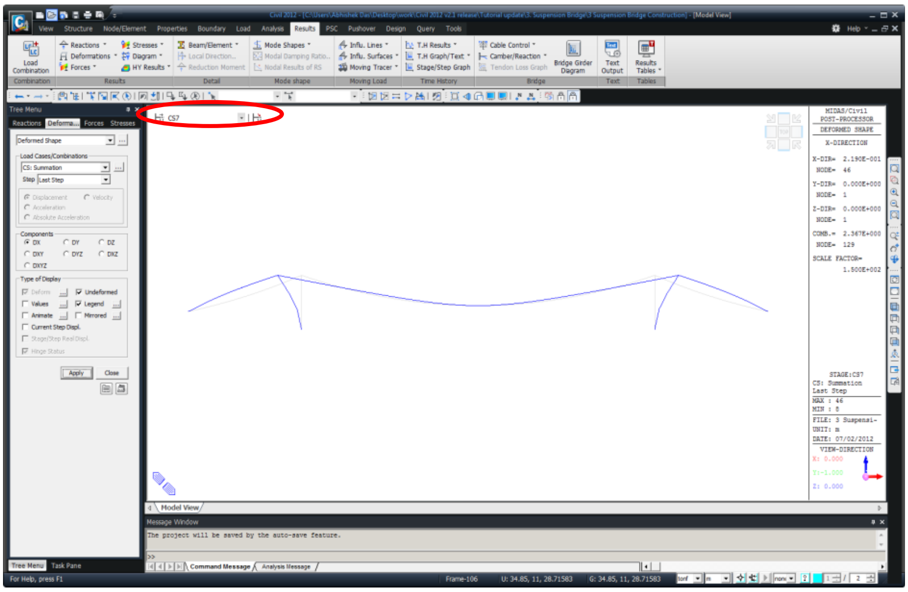


Fig. 69 Review setback value

* Setback value

Suspension bridges are designed to have no bending moments in pylons at the completed state stage by maintaining the applied horizontal forces in equilibrium at the tops of the pylons. However, if the cable is erected with the same center span length of the completed state stage, the resulting horizontal forces at the tops of the pylons are not in an equilibrium condition, and hence, cable slip will likely occur. The tops of the pylons are relocated (a type of horizontal camber) to set the horizontal cable forces balanced left and right. Generally, the tops of the pylons are pulled toward the side spans by wire ropes, and this is called setback.

We will now review the horizontal displacements changing with the construction stages at the top of a pylon by using a graph. As shown in Fig. 69, the horizontal displacement in backward construction stage CS7 becomes the setback value of the pylons.

Results / Stage/Step Graph ***Stage/Step History Graph***

Define Function>Displacement> Add New Function

Displacement>Name **(Right Tower)** ;

Node Number **(43)**

Components>**DX** ↵

Mode > **Multi Func.**

Step Option > **Last Step**

X-Axis > **Stage/Step**

Check Functions To Plot>**Right Tower**

Graph Title **(Right Tower Displacement)** Graph ↵

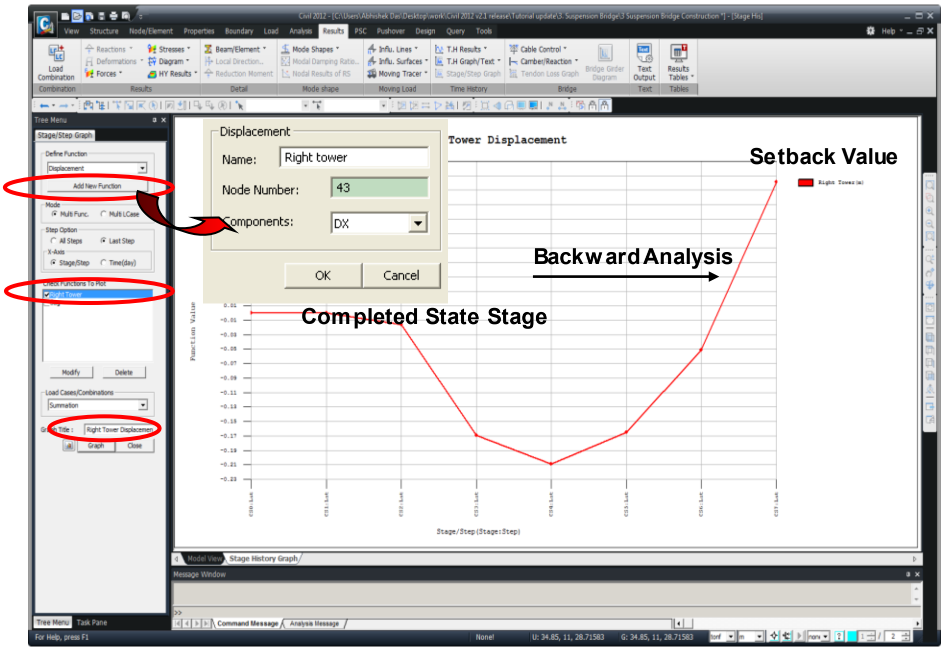


Fig. 70 Horizontal displacements of a pylon with changing construction stages

Review moments

Review the moments in the deck (main girders) and pylons (towers) for each construction stage. It is the characteristic of an earth-anchored suspension bridge that the deck (main girders) are subject to no moments due to dead loads during the construction stages and at the initial equilibrium state. Whereas, the towers are not subject to moments at the initial equilibrium state with the horizontal forces in equilibrium, but moments are developed during construction as shown in Fig. 70.

Results / Forces / Beam Diagrams...

Construction Stage > **CS2**

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

Components>**My**

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

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

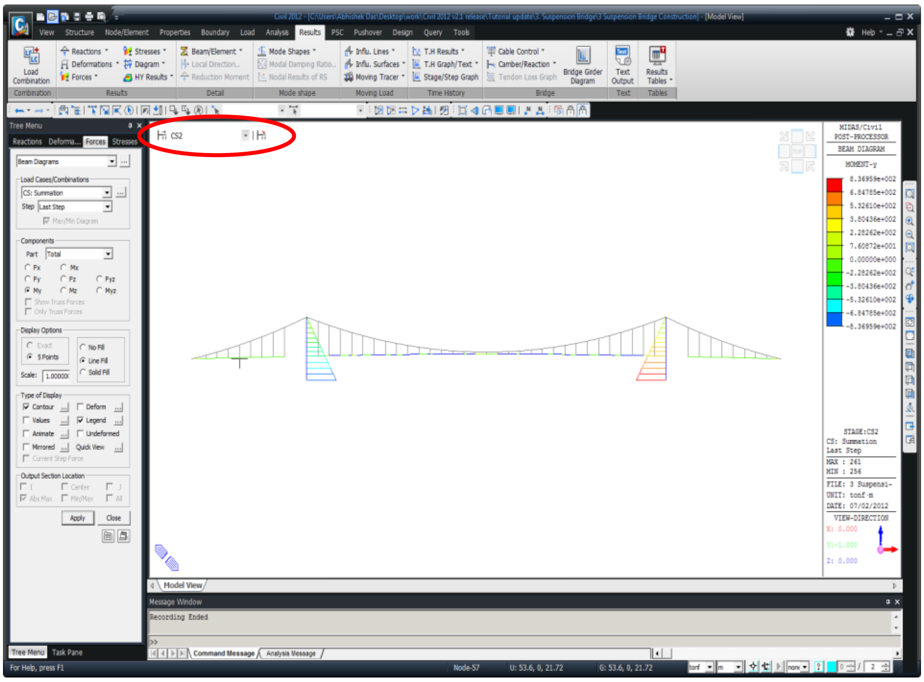


Fig. 71 Review of moments at each construction stage

Review axial forces

Review axial forces in the main cables & hangers for each construction stage.

Results / Forces / Truss Forces...

Construction Stage > **CS2**

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

Force Filter>**All**

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

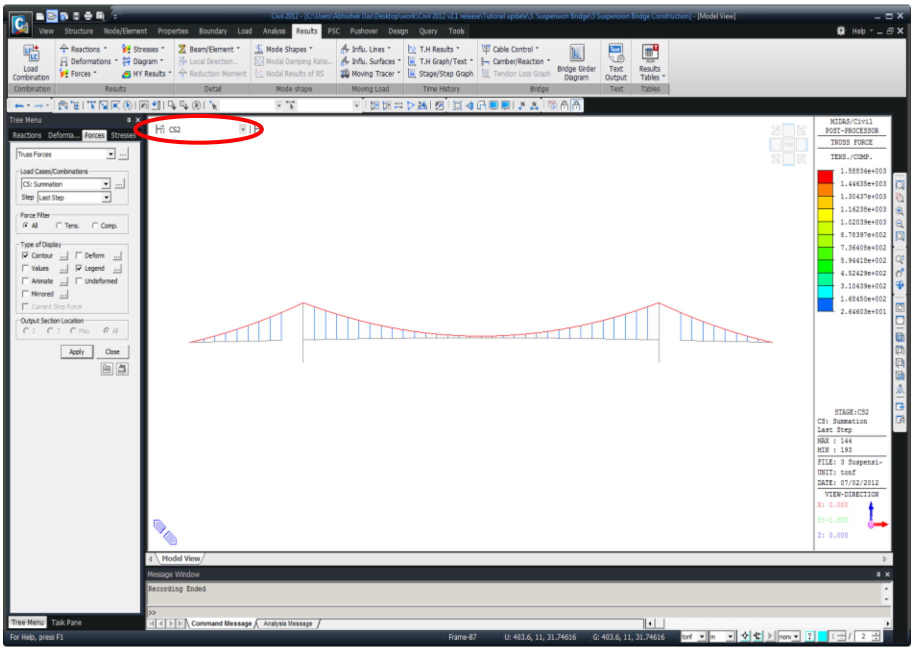


Fig. 72 Review of axial forces in the main cables and hangers

Review the change in tension forces in the cable adjoining the top of the right pylon for each construction stage.

Results / Stage/Step Graph ***Stage/Step History Graph***

Define Function>Truss Force/Stress> Add New Function
Truss Force/Stress >Name **(Cable Force)** ; Element No **(43)**
Force (on) ; Point>**i-Node** ↵

Define Function>Beam Force/Stress> Add New Function
Beam Force/Stress >Name **(Tower Axial Force)** ; Element No **(261)**
Force (on) ; Point>**i-Node** ; Components>**Axial** ↵

Mode > **Multi Func.**
Step Option > **Last Step**
X-Axis > **Stage/Step**
Check Functions To Plot>**Cable Force** ; **Tower Axial Force**
Graph Title > **Cable Force and Tower Axial Force**
Graph ↵

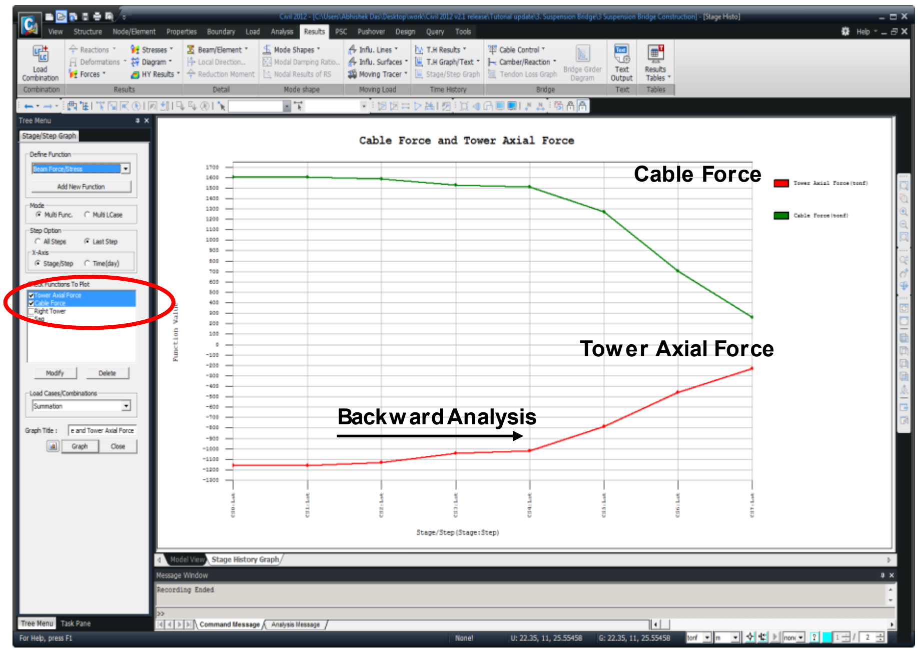


Fig. 73 Graph showing the change in main cable tension force at each construction stage

Review deformed shape using animation

Review the deformed shapes for each construction stage using the Animation function.

View/ Hidden ***Hidden*** (toggle on)

Results / Deformations / Displacement Contour...

Components>**DXYZ**

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

Legend (on) ; **Animate>** ...

Animation Details>**Animate Contour** (on)

Repeat Full Cycle > **(on)**

Frames per Half Cycle (5~50)>**(8)**

Frames per Second (5~50)>**(8)**

Construction Stage Option>**Stage Animation**

From>**CS0** ; To>**CS7** ↵

Record ↵

Close

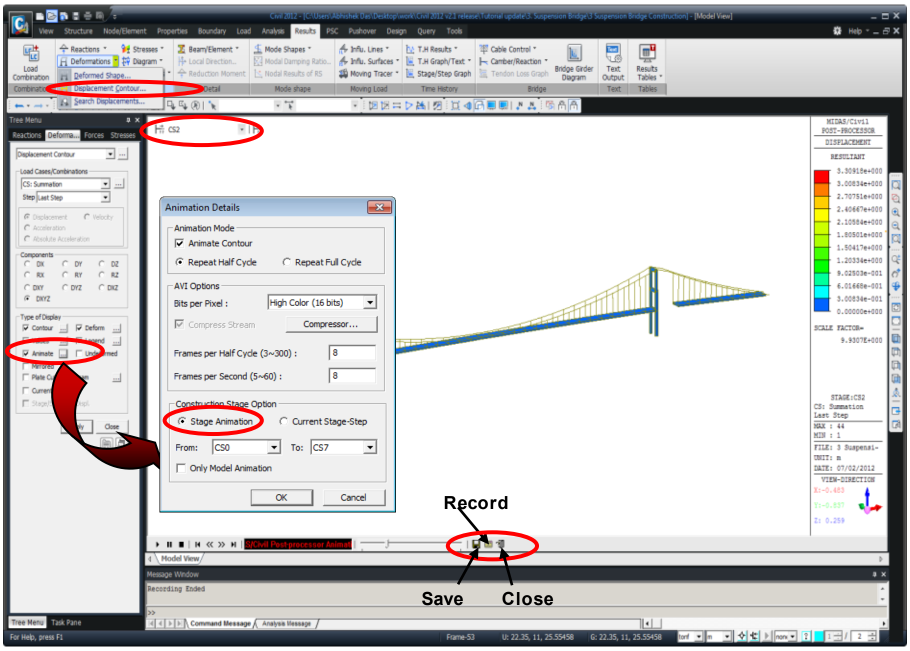


Fig. 74 Review of deformed shapes for each construction stage using the Animation function