Advanced Application 1

Construction Stage Analysis of MSS using the Wizard

Civil

CONTENTS

Summary

Two construction methods, MSS (Movable Scaffolding System) and FSM (Full Staging Method) are used to construct PSC box bridges span by span. In the MSS method, concrete is poured onto the formwork that is supported by the MSS. Hence, false work and shoring are not necessary. In addition when using this method, the space under the new bridge being constructed can be used without being obstructed by an existing river or roadway.

The structural system of the PSC Box Bridge, which is constructed using the MSS or FSM method, changes at each construction stage. Hence, structural analysis should be performed for each stage, and the sectional stability for each stage must be checked. To consider time dependent characteristics for the concrete and relaxation of the prestressing strand precisely, accumulated analysis results for the preceding construction stages are required for each subsequent construction stage.

In this tutorial, the procedure for performing construction stage analysis of a PSC Bridge using the MSS method, will be discussed. Analytical results, such as stresses, prestress losses, deflections and section forces for each construction stage, will be reviewed.

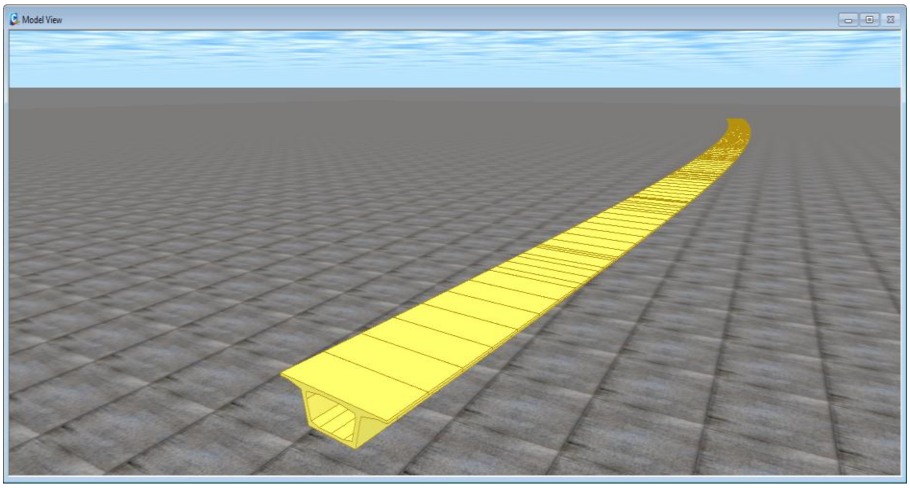


Fig. 1 Analytical model (at completion stage)

Bridge Dimensions and General Section

The general description of the analytical model is as follows:

Bridge type: 11 span continuous PSC box bridge (MSS)
Bridge length: L = 10@50.000 = 500.000 *m*
Bridge Width: B = 12.600 *m* (2 lanes)
Skew: none
Horizontal Radius: R = 2380.000 *m*

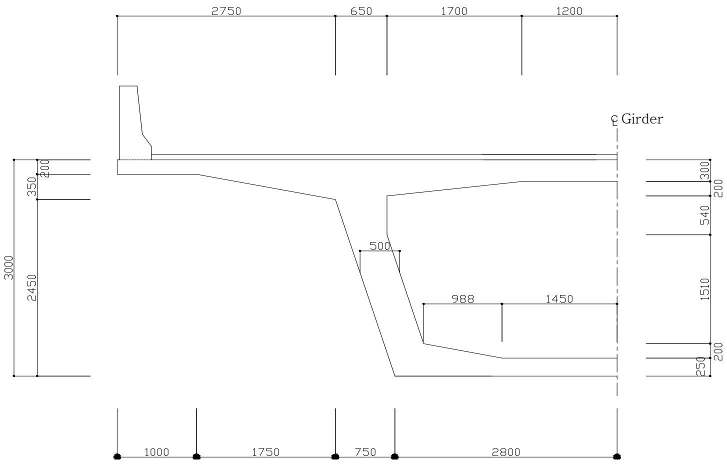


Fig. 2 ***General Section***

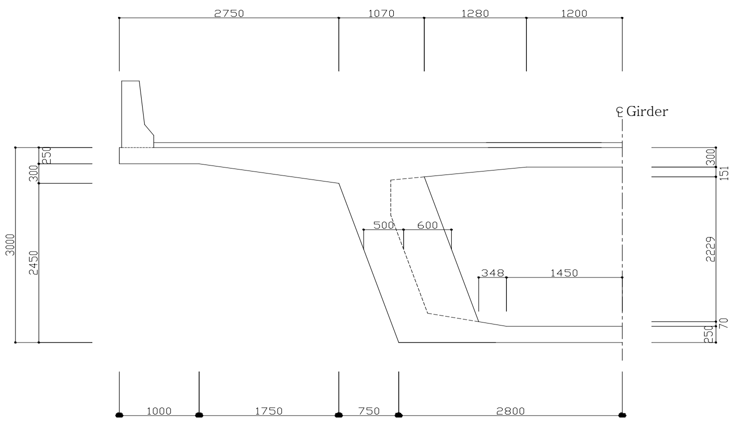


Fig. 3 Construction Joint Section

Construction Sequence for MSS

The construction sequence for the MSS method is as follows:

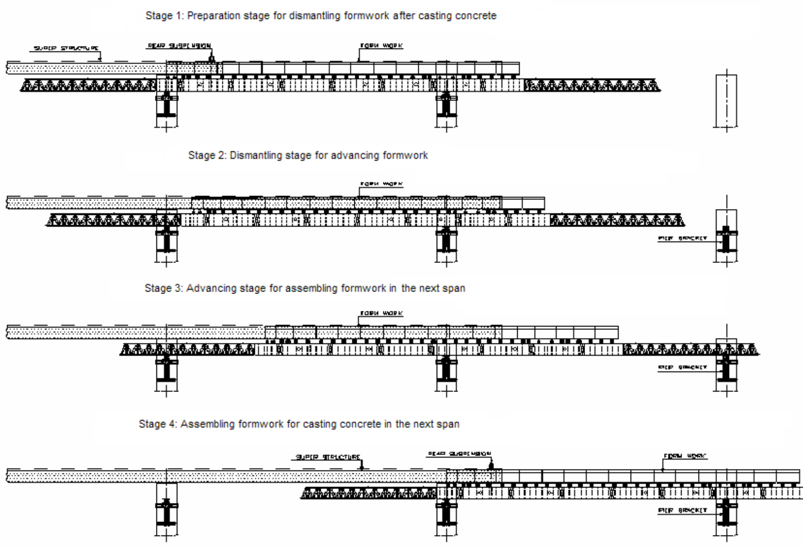


Fig. 4 Construction sequence

In the MSS construction stage analysis, construction sequence shown above should be followed precisely. In construction stage analysis, each construction stage is defined through activation/deactivation of **Structure Groups**, **Boundary Groups** and **Load Groups**. The procedure for performing construction stage analysis using MSS can be summarized as below. Of the steps given below, steps 2 to 8 are performed automatically when using the MSS Bridge Wizard.

---

1. Define material and section properties
2. Structural modeling
3. Define and compose Structure Group
4. Define and compose Boundary Group
5. Define Load Group
6. Load Input
7. Tendon Placement
8. Prestressing
9. Define and link Time Dependent Material Properties
10. Perform Structural Analysis
11. Review Output

Material Property and Allowable Stress

- **Superstructure Concrete**
  ASTM Grade C6000

- **Prestressing Strand** -Φ15.2 mm (0.6˝strand)
  Yield Strength: f_{py} = 1600000 kN/m^2
  Ultimate Strength: f_{pu} = 1900000 kN/m^2
  Elastic Modulus: E = 2.0 x 10^8 kN/m^2
  Initial Stress: f_{pi}=0.7f_{pu}=1330000 kN/m^2
  Anchorage Slip: $\Delta s = 6$ mm
  Friction Coefficient: $\mu = 0.30$ / rad
  Wobble Coefficient: $k = 0.006$ / m

Loads

- **Dead Load**
  Self-weight
  Input Self-weight
  Superimposed dead load
  w=38.00 kN/m

- **Prestress**
  Strand (φ15.2 mm×22 (φ0.6˝- 22))
  Area: A_u = 1.387 × 22 = 30.514 cm^2
  Duct Size: 110/113 mm
  Prestressing force: Jacking 70% of tensile strength
  f_{pj} = 0.7 f_{pu} = 1330000 kN/m^2
  P_i = A_u × f_{pj} = 4058.362 kN
  Prestressing losses after the initial loss are automatically calculated by the program

  Friction loss: $P_{(X)} = P_0 \cdot e^{-(\mu\alpha + kL)}$
  $\mu = 0.30$, $k = 0.006$
  Anchorage Slip Loss: $\Delta I_c = 6$ mm
  Elastic Shrinkage Loss: $\Delta P_E = \Delta f_P \cdot A_{SP}$
  Final Loss (automatically calculated within the Program)
  Relaxation
  Creep and Shrinkage Loss

- **Creep and Shrinkage**
  Cement: Normal Cement
  Concrete age when long term load is acting: t_c = 5 days
  Concrete age when concrete is exposed to air: t_c = 3 days

Relative Humidity: RH = 70 %

Air or Curing temperature: $T = 20\,^{\circ}C$

Applied Code: CEB-FIP

Creep Coefficient: automatically calculated within the Program

Shrinkage Coefficient: automatically calculated within the Program

- **Rear Cross Beam Reaction**

  Assume rear cross beam reaction due to the MSS girder self-weight as follows:

  P = 4000 kN

  Location: 3 m from the construction joint

  Reactions due to self-weight of wet concrete: calculated automatically by the program.

Assign Work Environment

For the construction stage analysis, open a new file (**New Project**) and save (**Save**) as '**MSS**'.

And assign the unit system to '**kN**' and '**m**'. This unit system can be changed arbitrarily for the user's convenience.

Click on - *New Project*

-*Save* **(MSS)**

Tools / ***Unit System*** Unit System

The unit system can be changed by clicking the unit selection button ( ) in the Status bar at the bottom of screen.

Length> **m**; Force (Mass)>**kN (ton)** ↵

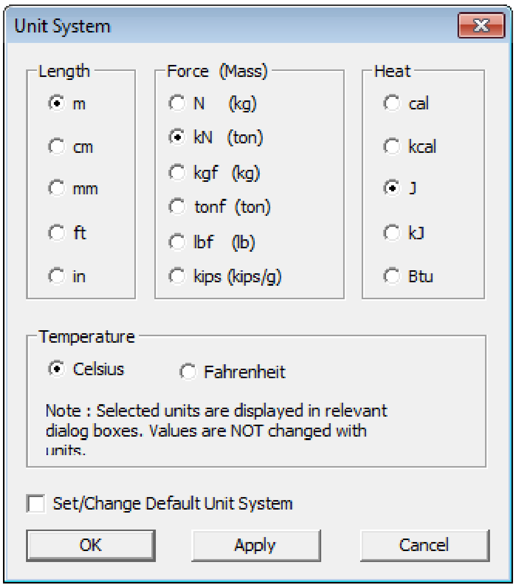


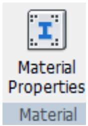
Fig. 6 Assign unit system

Define Material Properties

The PSC box section is defined within the MSS Wizard, and need not be defined separately.

Define material properties for PSC box girder and tendon.

Properties / ***Material Properties***



Type of Design>**Concrete**; Standard>**ASTM(RC)**

DB>**Grade C6000**↵

It is more convenient to use the Apply button when multiple properties are defined at the same time.

Name **(Tendon)**; Type of Design>**User Defined**; Standard>**None**

Analysis Data

Modulus of Elasticity **(2.0e8)** ↵

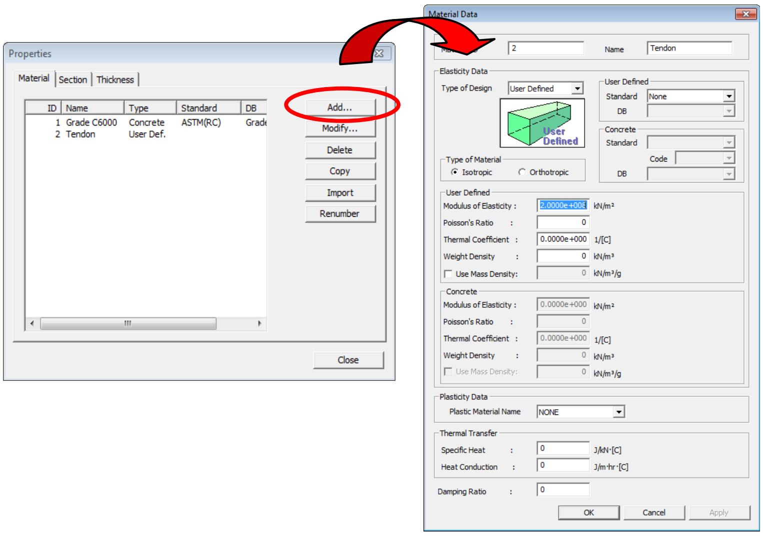


Fig. 7 Material property input dialog box

Modeling using MSS Bridge Wizard

Perform modeling the MSS Bridge using the MSS Bridge Wizard function. The MSS Bridge Wizard consists of three tabs – Model, Section and Tendon.

Input Model Data

The MSS Bridge and FSM Bridge Wizards automatically create models and construction stages for the MSS and FSM Bridges, respectively. The difference in construction stage analysis between MSS and FSM is the method of supporting the wet concrete and self-weight of formwork. In the FSM method, wet concrete and self-weight of formwork are supported by shoring and, hence, the PSC box girders, which have been already constructed, are not affected by them. On the other hand, in the MSS method, the weight of the wet concrete and self-weight of formwork is transmitted to the overhanging parts of the constructed PSC box girders through the rear crossbeam. This prevents uneven deflection across the construction joint. The key difference between the MSS and FSM can be summarized as "the method in which the wet concrete and self-weight of formwork are supported during construction". If the MSS Bridge Wizard is selected and the Movable Scaffolding Reaction is entered, this reaction is automatically calculated and inputted as a construction stage load.

The Movable Scaffolding Reaction is the reaction from the self-weight of the MSS girder that acts on the rear cross beam.

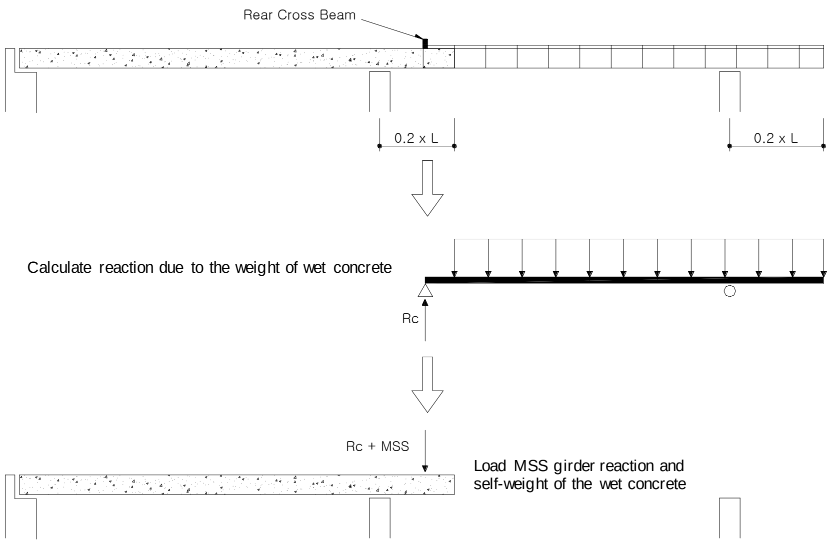


Fig. 8 Reaction acting at Rear Cross Beam

Select the MSS Bridge Wizard and input bridge material, span layout, horizontal curvature radius, location of fixed support, location of construction joint, construction duration for each span (20 days) and initial age for PSC box girder.

In this example, it is assumed that 15 days will be required for installing formwork, rebars and sheaths, and 5 days will be required for pouring and curing concrete. Hence, a total of 20 days per each segment is assumed.

If MSS Bridge Wizard is selected, the difference between the Stage Duration and Initial Member Age is calculated in the program to define Additional Steps, and then the reactions due to self-weight of MSS girder and wet concrete are applied.

In MIDAS/Civil, the load that has a time lag in the same structural system can be defined by using Additional Steps without having to define a different construction stage. More detailed explanations can be found in Functions of Civil > Construction Stage Analysis Data > Define Construction Stage in the Online help.

A curved MSS bridge is modeled by checking on Radius and inputting the radius.

Structure / MSS Bridge

Model tab

Bridge Model Data Type>**Type1** ; Bridge Material>**1: Grade C6000**

Span(L)>**10@50** ; Radius (on)>**2380**; Convex (on)

Fixed Support>**250(50)** ; Segment Division per Span>**10**

Cold Joint (S3)>**0.2** ; Anchorage(S4)>**3**

Diaphragm(S5)>**1** ; Stage Duration>**20**

Initial Member Age>**5** ; Movable Scaffolding Reaction>**4000** ↵

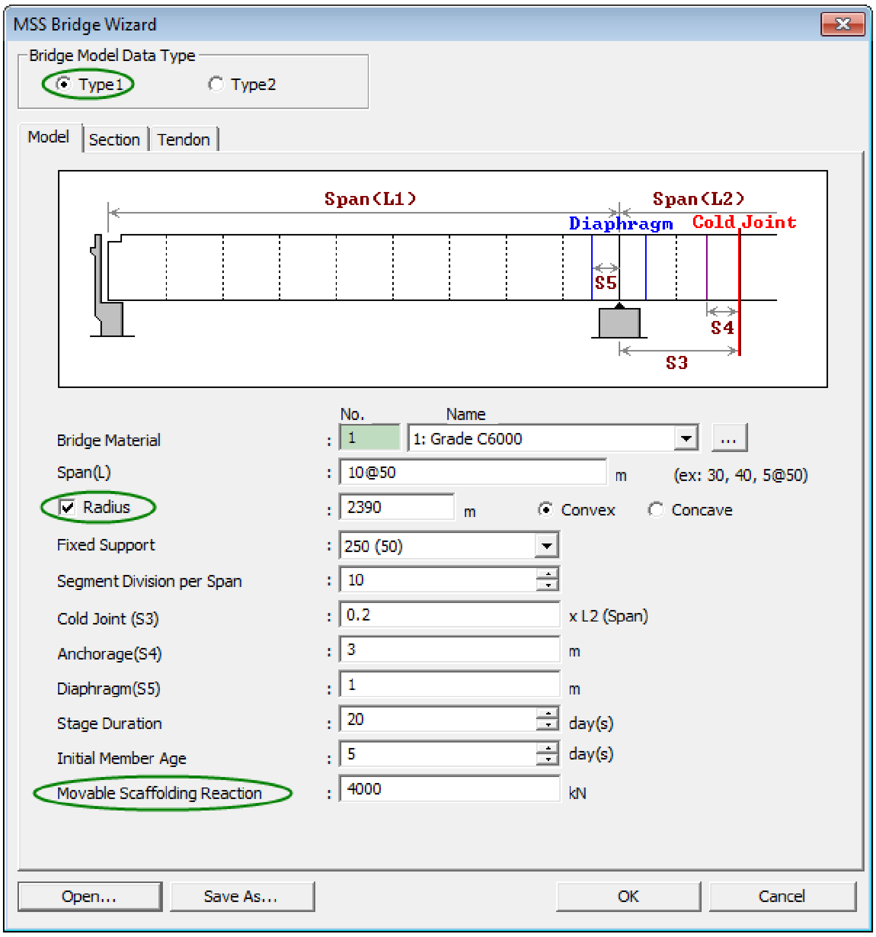


Fig. 9 Model Tab in the MSS Bridge Wizard

Input Section Properties for PSC box

Input the section properties for the general section and construction joint section. Input the section properties for the general section referring to Fig. 11. The defined section shape can be reviewed by selecting Drawing under the View Option.

Section tab

Center tab

H1 (**0.2**); H2 (**2.75**); H3 (**0.3**); H4 (**0.3**)

H5 (**0.2**); H6 (**0.54**); H7 (**0.2**); H8 (**0.25**)

B1 (**2.75**); B2 (**0.75**); B3 (**2.8**); B4 (**1.75**)

B5 (**1.7**); B6 (**1.2);** B7 (**0.988**); B8 (**1.45**)

View Option>**Drawing**

PSC box section for MSS Bridge is modeled with respect to the level of Center-Bottom.

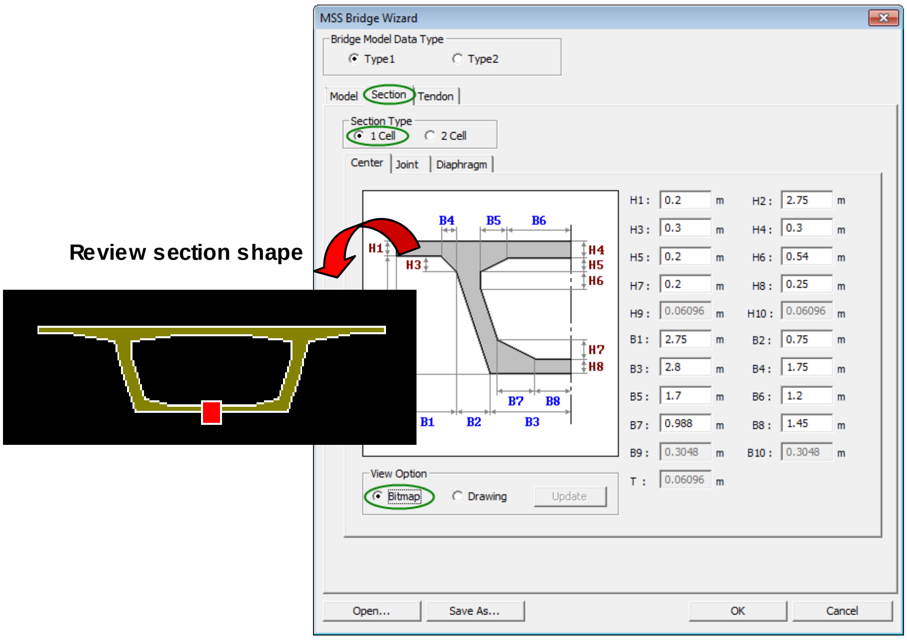


Fig. 10 Input section property for general section

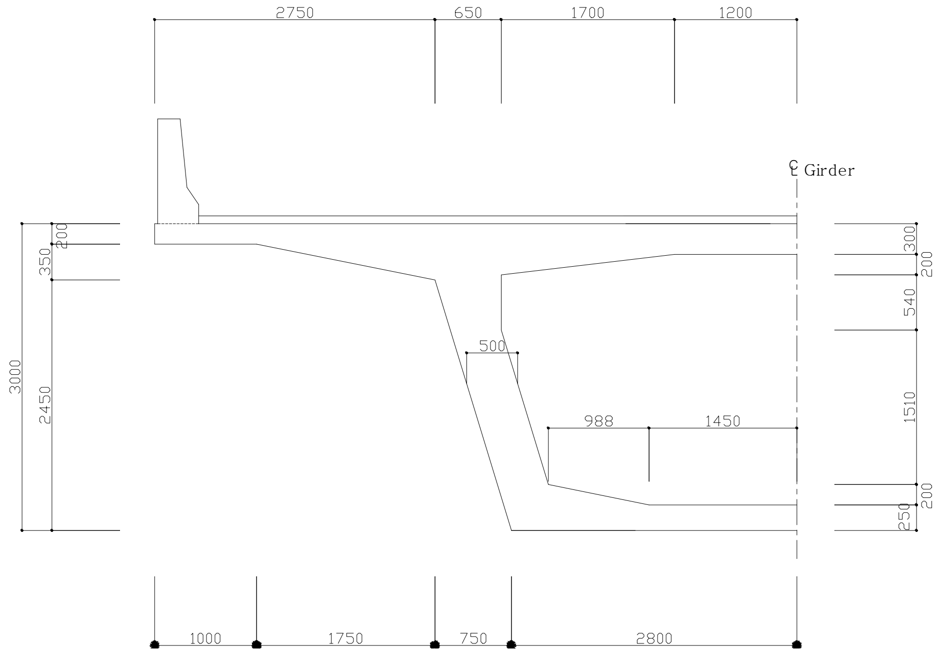


Fig. 11 PSC box general section

Input the section properties for the construction joint part referring to Fig. 13. The defined section shape can be reviewed by selecting Drawing under the View Option.

View Option>**Bitmap**

Joint tab

H3 **(0.3)**; H5 **(0.151)**; H7 **(0.07)**
B4 **(1.75)**; B5 **(1.28)**; B7 **(0.348)**

View Option>**Drawing**

Diaphragm tab

H4 **(0.3)**; H5 **(0.151)**; H6 **(0.54)**; H7 **(0.07)**
H8 **(0.25)**; B5 **(1.28)**; B6 **(1.2)**; B7 **(0.348)**; B8 **(1.45)**

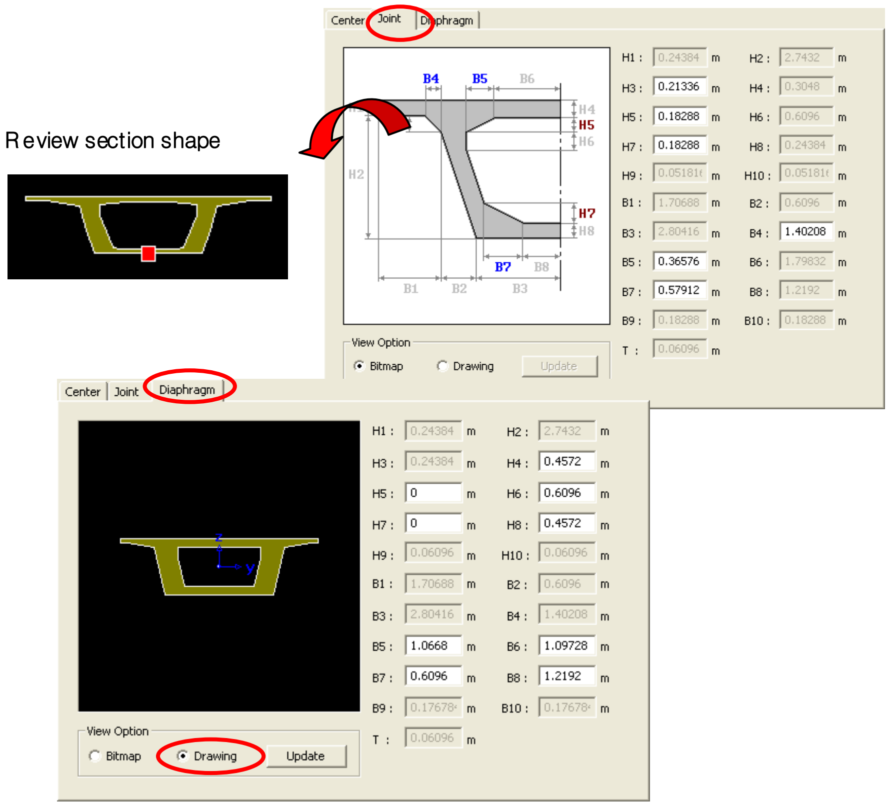


Fig. 12 Input section property for the construction joint

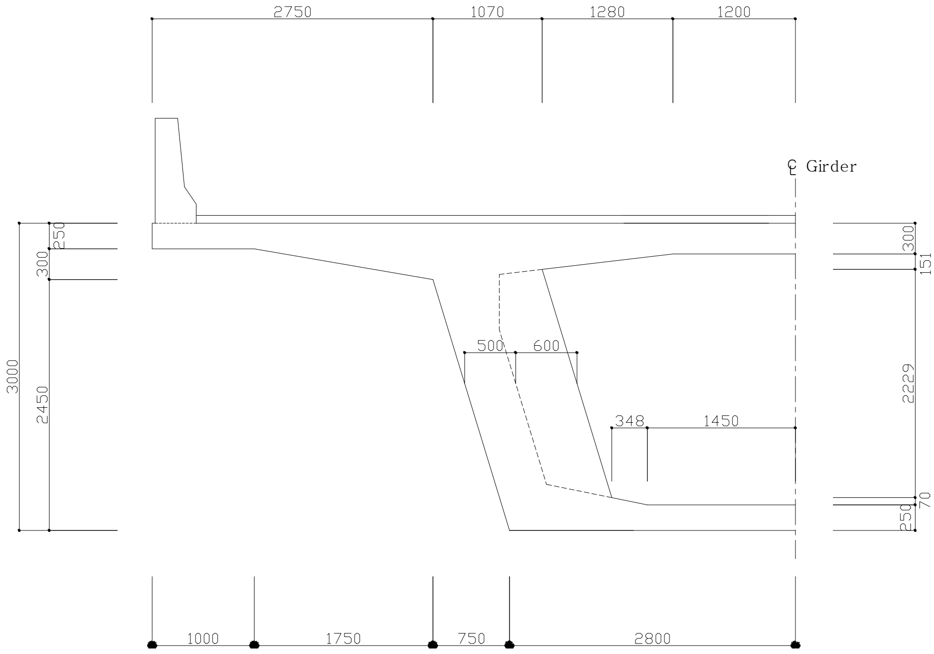


Fig. 13 PSC box Construction joint section

Input Tendon Placement

Tendons in the MSS Bridge are placed along a curvature within the webs of the PSC box girder. In the MSS Bridge Wizard, this type of standard tendon placement can be simply defined by the lowest, inflection and anchorage points.

Generally, the MSS Bridge has equal spans, and the structural system of the first span is changed from a simple beam to a continuous beam at each stage as construction proceeds. The maximum bending moment for the first span constructed is larger than that of the other spans because it is a simply supported condition. Hence, additional tendons should be provided for the first span. The additional tendons for the first span are defined by general function, and only typical tendons are defined in the MSS Bridge Wizard.

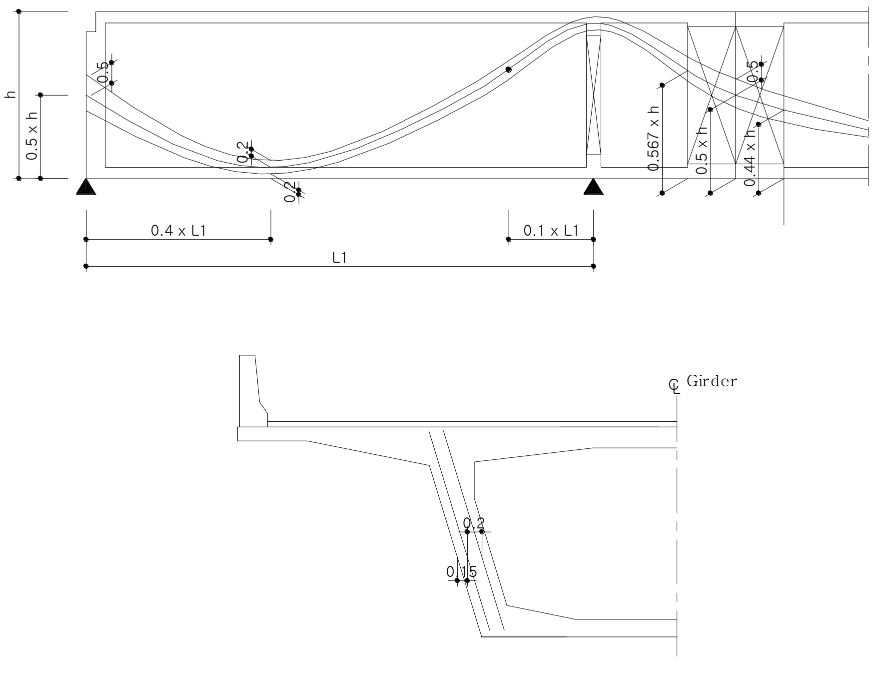


Fig. 14 Tendon Placement

Input typical tendon placement in the longitudinal direction referring to Fig. 14.

Tendon tab

N(**3**); G1(**0.5**); G2(**0.2**); G3(**0.5**)

S1(**0.4**); S2(**0.1**); C(**0.2**); a1(**0.567**); a2(**0.44**)

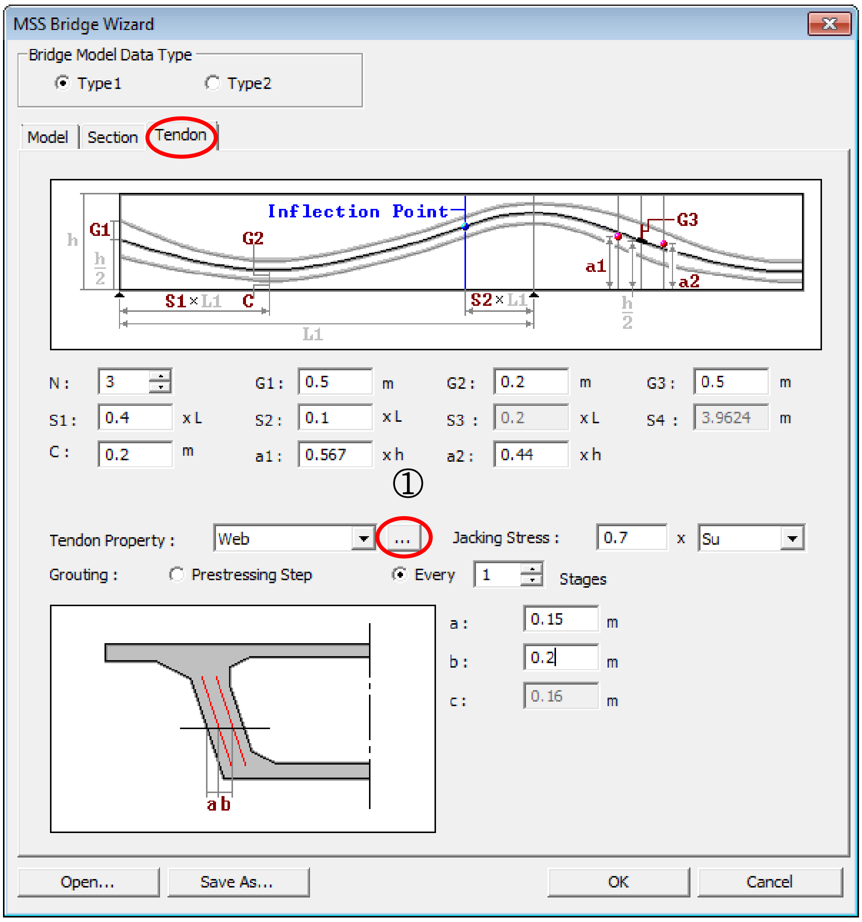


Fig. 15 Input Tendon Placement

Input the tendon properties such as tendon areas, constants related to losses and tendon strength.

Tendon Property> ... (① in Fig. 15)

Tendon Property> Add

Tendon Name>**Web** ; Tendon Type>**Internal**

Material>**2: Tendon**

Total Tendon Area ...

Strand Diameter>**15.2mm (0.6 ")**

Number of Strands>**22** ↵

Duct Diameter>**0.113**

Relaxation Coefficient>Magura **(45)** 🎧

Curvature Friction Factor>**0.3** ; Wobble Friction Factor>**0.0066**

Ultimate Strength>**1900000** ; Yield Strength>**1600000**

Bond Type>**Bonded**

Anchorage Slip (Draw in)>Begin **(0.006)**; End **(0.006)** ↵

🎧 Relaxation coefficient is a material constant included in Magura's formula, which is used to calculate relaxation effects of tendons over time. 10 for general tendons and 45 for low relaxation tendons are usually used.

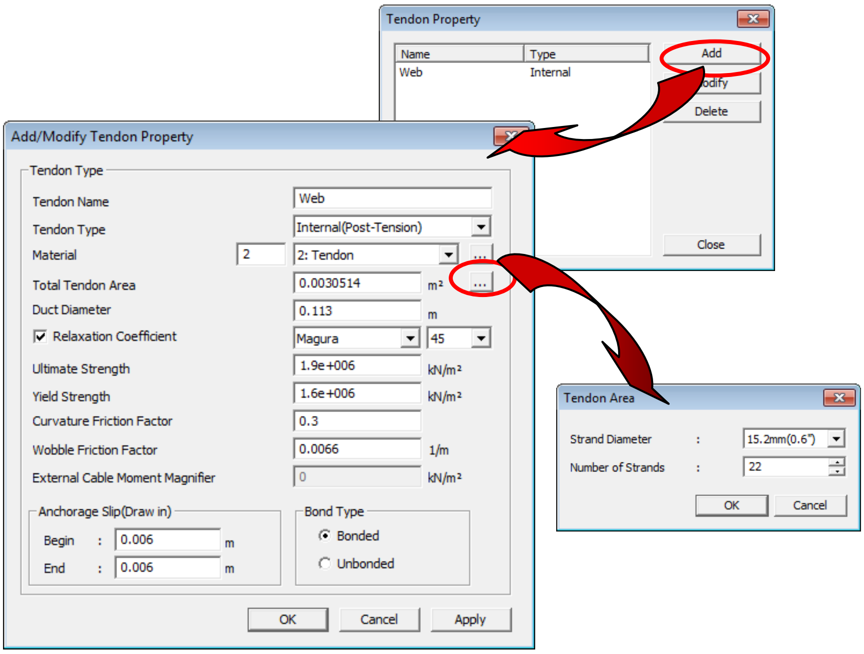


Fig. 16 Tendon Property Input

Input the tendon jacking force, grouting time and transverse arrangement. If the grouting time is assigned using the Prestressing Step, the stress at the time of tendon jacking is calculated based on section properties that include the effects of tendon area. If the grouting time is assigned using Every (n) Stages, the stress at the time of tendon jacking is calculated based on the net section, and the tendons tensioned during the n stages are grouted simultaneously at the n-*th* construction stage. The tendon jacking force is specified as 70% of the ultimate strength. The transverse arrangement of the tendons is also defined. The tendons in the MSS Bridge are normally arranged parallel to the webs of the PSC box girder. The transverse tendon arrangement as such can be defined by setting the distance between the outer side of web and outer tendon, and the spacing between the outer and inner tendons.

Jacking Stress **(0.7)**×**(Su)**
Grouting>**Every (1) Stages**
a **(0.15)**; b **(0.2)**

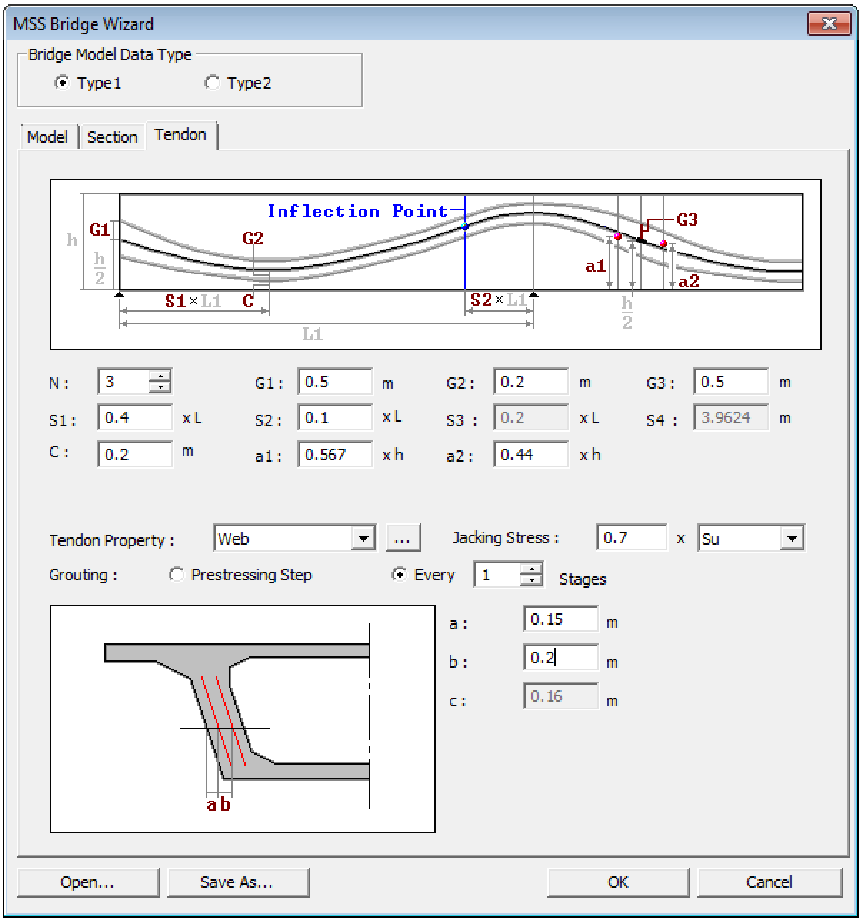


Fig. 17 Transverse Tendon Arrangement

After completion of data input, click OK button to end the MSS Bridge Wizard session, and review the model. Specific parts of the model can be magnified and reviewed by using the *Zoom Window* and *Zoom Fit* functions.

Point Grid (off), ***Point Grid Snap*** (off), ***Line Grid Snap*** (off)
Node Snap (on), ***Element Snap*** (on)

Display

Misc tab

Tendon Profile Point (on) ↵

Zoom Fit, Hidden ***Hidden*** (on)

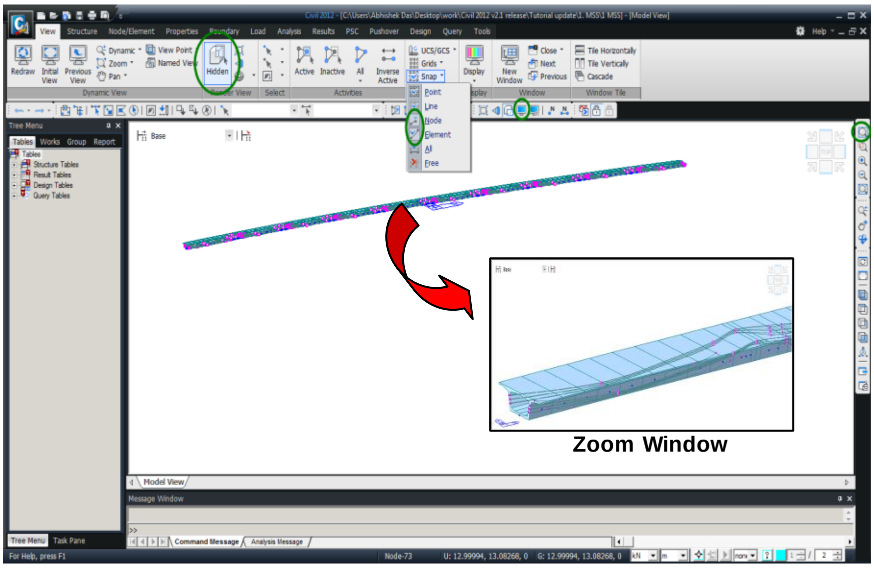


Fig. 18 Bridge Model Generated by the MSS Bridge Wizard

Modify Existing Data and Input Additional Data

Define Construction Stage

There are two working modes, Base Stage mode and Construction Stage mode, in MIDAS/Civil for performing construction stage analysis.

In the Base Stage mode, all structural modeling data, loading conditions and boundary conditions can be defined, but analysis is not performed in this mode. The Construction Stage represents the state of model for which analysis is performed. In the Construction Stage mode, the structural model data cannot be modified or deleted, except that the boundary conditions and loads pertaining to each construction stage may be changed.

Construction Stage is defined by activating/deactivating the element groups, boundary groups and load groups, instead of individual elements, boundary conditions and load conditions. Boundary conditions and load conditions that are included in an activated Boundary Group or Load Group in the Construction Stage mode can be modified or deleted.

Modifying or deleting nodes or elements is not permitted in the Construction Stage Mode. Other than activating boundary and load conditions, modifying and deleting are permitted only in the Base Stage Mode.

We will now review the construction stages automatically defined by the MSS Bridge Wizard. Construction stage information can be reviewed by using the ***Stage Toolbar*** and ***Works Tree***. All activated/deactivated ***Structure Groups***, ***Boundary Groups*** and ***Load Groups*** for the current stage can be reviewed systematically by using the ***Works Tree,*** if the construction stage is representing any construction stage other than the Base Stage. Moreover, changes to the structural system by construction stages can be reviewed visually in ***Model View*** by sequentially changing the construction stages.

We will now review the structural systems and loads by selecting each construction stage using the ***Stage Toolbar***.

Display

Boundary tab

Support (on)

Load tab

Nodal Load (on)↵

Tree Menu>**Works tab**

Construction Stage>**CS04**

Locate the cursor on the Stage Toolbar and review all stages one by one by adjusting the arrow keys on the keyboard.

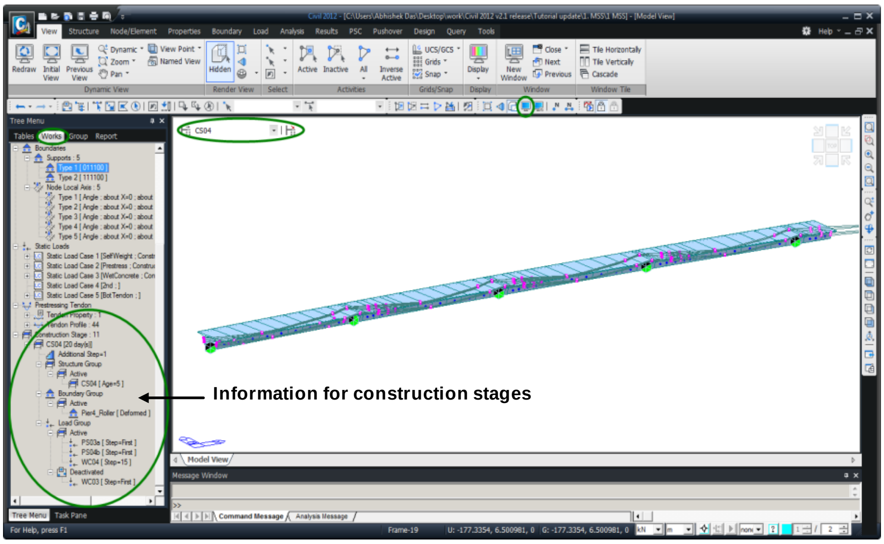


Fig. 19 Structural system for Construction stage 4

Input Additional Loads

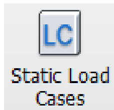
Additional tendons should be provided in the first span because the maximum bending moment acting in the first span is larger than those in the other spans. This is because the first span is a simply supported span during construction whereas the other spans are continuous. The additional tendons in the first span are inputted in construction stage 1 (CS01).

After completing construction of all the spans by the MSS method, superimposed dead loads such as pavement, barriers and railings are constructed. We will define additional construction stages, loads, and superimposed dead loads. The superimposed dead loads are applied for 10,000 days to account for creep. This will enable the generation of a camber diagram and, thus, managing the camber.

Define the additional tendons in the first span and the load conditions pertaining to superimposed (2nd) dead loads.

Change to Base Stage because load conditions can be added only in the Base Stage.

Stage>**Base**

Load / Static Loads / 

Name **(2nd)**; Type>**Construction Stage Load** Add

Name **(BotTendon)**; Type>**Construction Stage Load** Add ↵

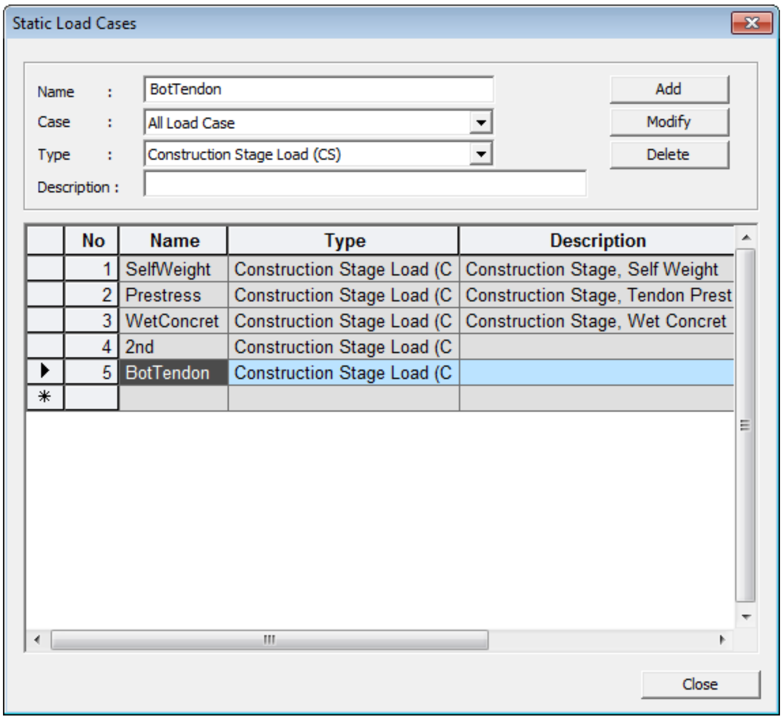


Fig. 20 Define Load Conditions

Define prestress for the additional tendons and generate a Load Group corresponding to superimposed dead loads.

Hidden (off)

Display

Boundary tab

Support (off)

Load tab

Nodal Load (off)

Misc tab

Tendon Profile Point (off) ↵

Group tab

Group / Load Group / **New…**(right click mouse)

Name>**2nd** Add

Name>**BotTendon** Add ↵

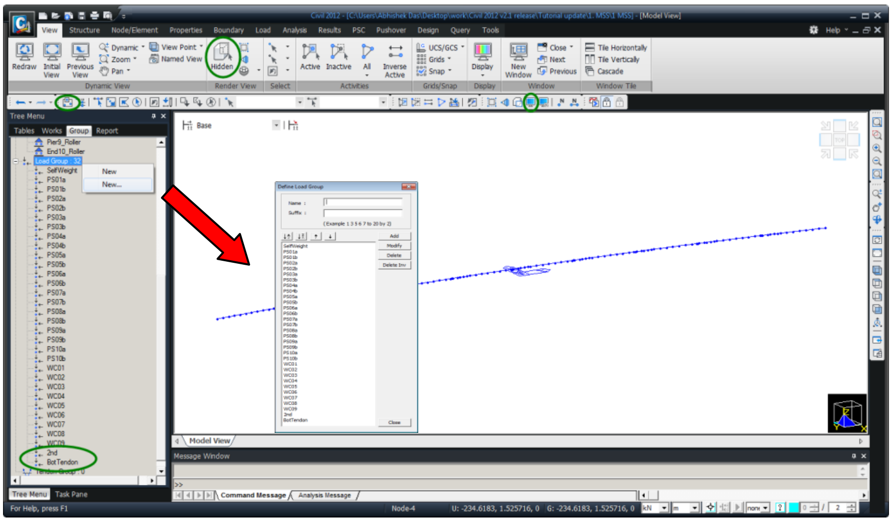


Fig. 21 Load Group Generation

The prestress load due to the additional tendons in the first span acts at the construction stage CS01. The procedure to apply this additional prestress load in the first span is as follows:

1. Define the tendon profile.
2. Assign the prestress load due to the defined tendon profile in the load group “BotTendon”.
3. Activate the load group “BotTendon” in CS01.

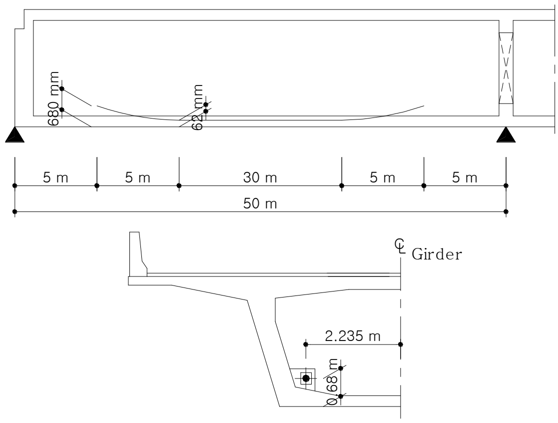


Fig. 22 Additional Tendon in the 1st span

The reference point of the tendon profile is the Bottom Center of the PSC box section since the PSC box section is defined relative to the same location.

Define the tendon profile to be added in the first span, referring to Fig. 22. The additional tendons are placed over 40m length between the i-*th* end of Element 2 and the j-*th* end of Element 9. The start and end points of the tendons are each located at 5m from the bridge end and the second support. The length of a single element is 50/10 = 5*m*.

Loads / Temp./Prestress / Tendon Profile

Tendon Profile> Add

Tendon Name>**Bot1** ; Tendon Property>**Web**

Assigned Elements>**2to9**

Input Type>**3D** ; Curve Type>**Spline**

Straight Length of Tendon>Begin **(0)** ; End **(0)**

Profile>Reference Axis>**Curve** (on)

1>x (**0**), y (**0**), z (**0.68**), **fix** (off)

2>x (**5**), y (**0**), z (**0.062**), **fix** (on), Ry (**0**), Rz (**0**)

3>x (**35**), y (**0**), z (**0.062**), **fix** (on), Ry (**0**), Rz (**0**)

4>x (**40**), y (**0**), z (**0.68**), **fix** (off)

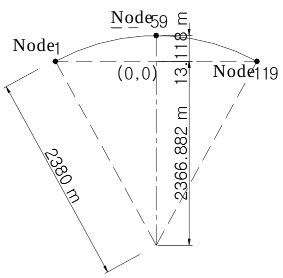
Profile Insertion Point **(Node 2)**

Radius Center (X, Y) **(0, -2366.882)**; Offset **(-2.235)**

Direction>**CW** ↵

The slope is fixed by a given value if "fix" is 'on' or generated within the program if fix is 'off'.

Select Curve and input the tendon profile referenced to circular curvature on the X-Y plane since the current example bridge is curved.



Tendon Name **(Bot2)**; Tendon Property>**Web**

Assigned Elements **(2to9)**

Straight Length of Tendon>Begin **(0)**; End **(0)**

Profile>Reference Axis>**Curve** (on)

1>x (**0**), y (**0**), z (**0.68**), **fix** (off)

2>x (**5**), y (**0**), z (**0.062**), **fix** (on), Ry (**0**), Rz (**0**)

3>x (**35**), y (**0**), z (**0.062**), **fix** (on), Ry (**0**), Rz (**0**)

4>x (**40**), y (**0**), z (**0.68**), **fix** (off)

Profile Insertion Point **(Node 2)**

Radius Center (X, Y) **(0, -2366.882)** ; Offset **(2.235)**

Direction>**CW**↵

The center coordinates (x,y) of the circle for this bridge are (0,-2366.882) because this bridge is modeled with Node 59 as the symmetric point and the y-coordinate of the Node 59 is 13.118.

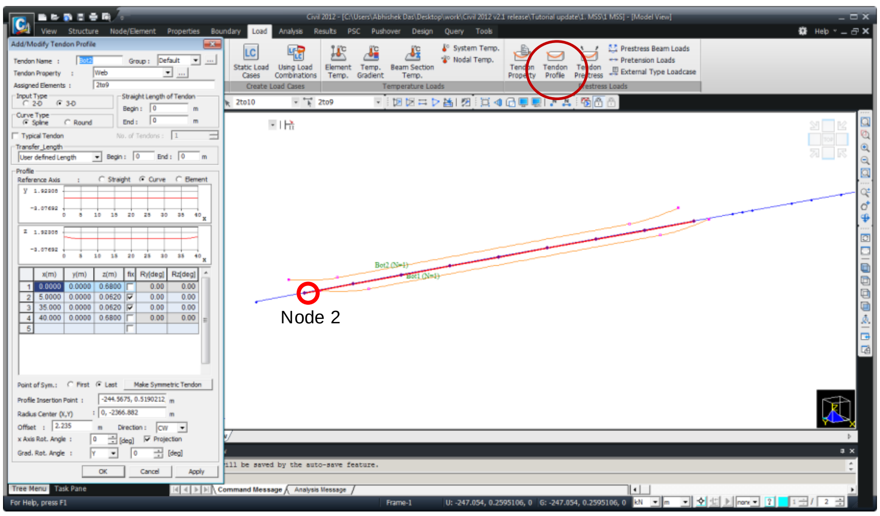


Fig. 23 Define Tendon Profile

Assign the prestress loads, which are defined by the tendon profile, to the BotTendon load group, and apply them to the structural system.

The tendons are jacked only at one end. Therefore, input prestress only at that end. The jacking force is 70% of the ultimate strength.

Input 1 because this tendon is grouted at the construction stage CS02 after prestressing the tendons.

Load/ Temp./Prestress / Tendon Prestress

Load Case Name>**BotTendon**; Load Group Name>**BotTendon**

Tendon>**Bot1, Bot2** > Selected Tendons

Stress Value>**Stress**; 1st Jacking>**Begin**

Begin **(0)**; End **(1330000)**

Grouting: after **(1)** Stage Add

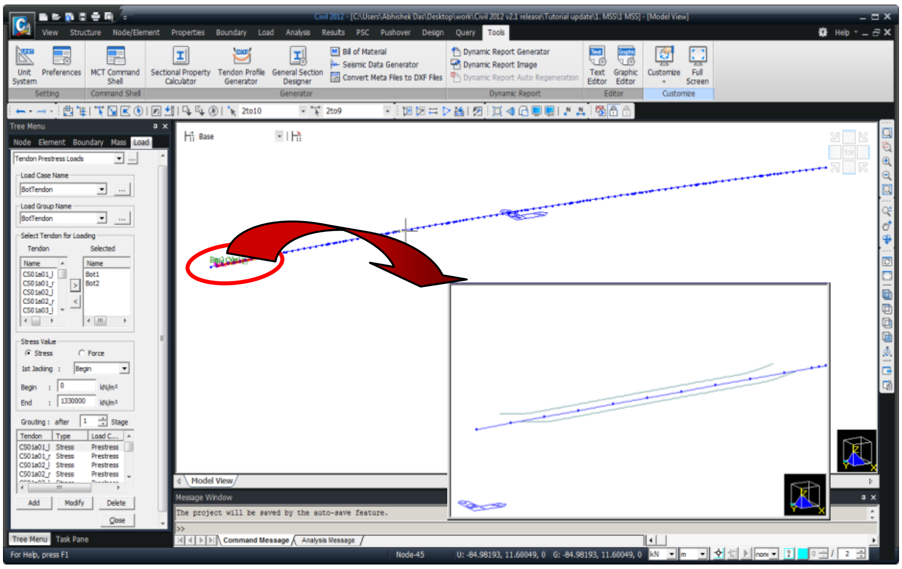


Fig. 24 Loading Prestress

Activate the load group "BotTendon", which represents prestress for the first span, at the construction stage CS01.

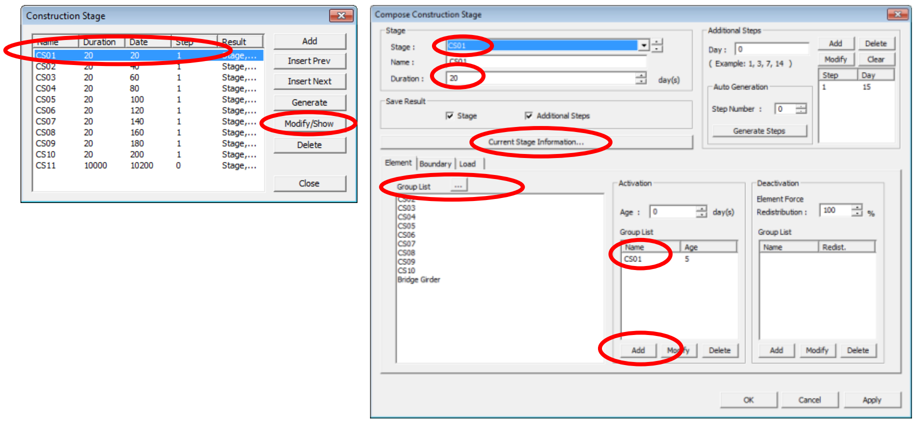
Load / Construction Stage / Define C.S **_Define Construction Stage_**

Name>**CS01**; Modify/Show

Load tab

Group List>**BotTendon**

Activation>Active Day>**First**; Add ↵



The names for element groups, boundary groups and load groups, which are automatically generated by the Bridge Wizard, can be found in "Define Structure (Boundary, Load) Group" in the On-line manual.

Fig. 25 Activation of Prestress

Apply superimposed (2^{nd}) dead load. Input the superimposed (2^{nd}) dead load in construction stage CS11 because the effect of creep and prestress losses during the period of 10,000 days should be considered in the analysis. Assign the superimposed (2^{nd}) dead load to the 2nd load group and apply it to the structural system. The magnitude of the superimposed (2^{nd}) dead load is 38 kN/m acting in the –Z-direction.

Load / Static Loads / Element ***Element beam Loads***

Select all

Load Case Name>**2nd**; Load Group Name>**2nd**

Options>**Add**; Load Type>**Uniform Loads**

Direction>**Global Z**; Projection>**No**

Value>**Relative**; x1 **(0)**, x2 **(1)**, W **(-38)** ↵

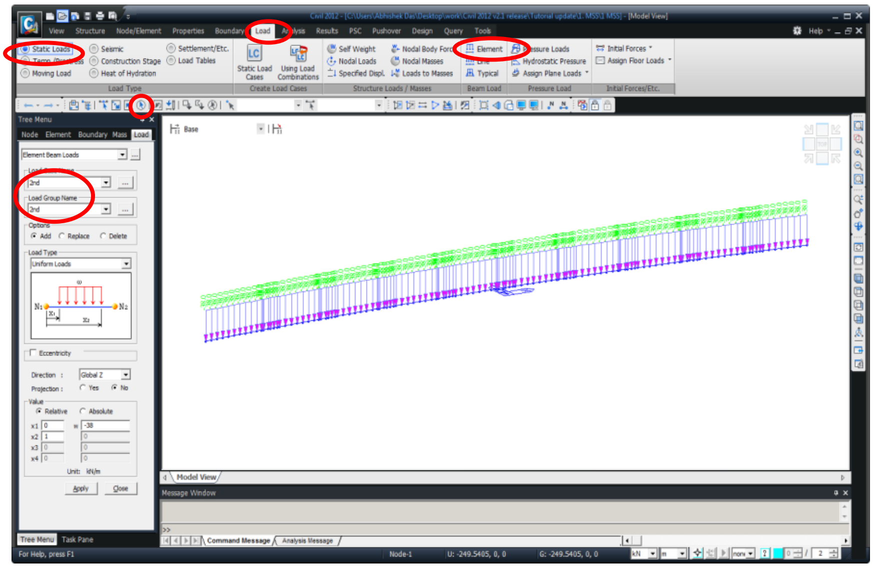


Fig. 26 Superimposed dead load

Activate the Load Group 2nd in construction stage CS11.

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

Name>**CS11**; Modify/Show

Stage>Duration **>"10000"**

Save Result>**Stage** (on); **Additional Step** (on)

Load tab

Group List>**2nd**

Activation

Active Day>**First**

Add ↵

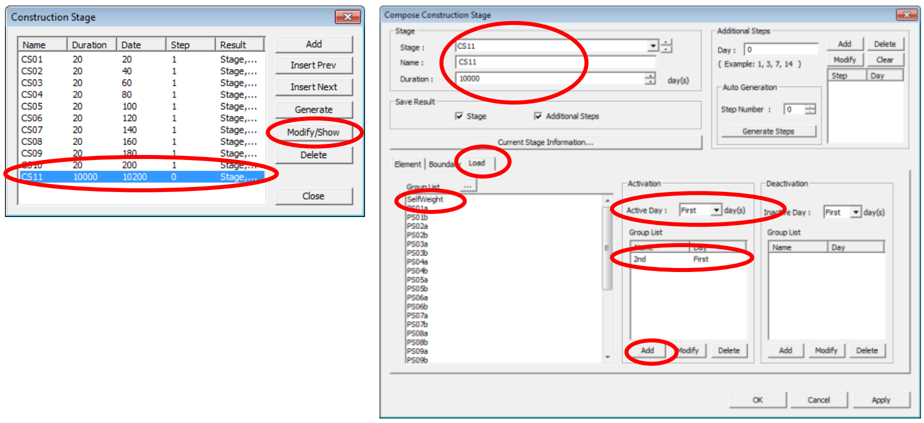


Fig. 27 Modification for CS11

Define and Link Time Dependent Material Properties

After completing the modeling of entire concrete structure for both superstructure and substructure, define the time dependent material properties (concrete strength gain curve, creep coefficient and shrinkage coefficient) for all the sections, and then link these properties to each section.

Creep and shrinkage coefficients are a function of sectional shape ("Notational Size of Member"). Therefore, we must first determine the variable section dimensions and then input the time dependent material properties.

According to CEB-FIP standards, the creep and shrinkage coefficients for concrete vary with the member dimensions. Therefore, in order to perform an analysis considering time dependent material properties, the material property of each section with different dimensions must be linked one-to-one to the corresponding time dependent material properties. That is, the number of different time dependent material properties and general material properties should correspond to the number of elements with different sectional dimensions. The time dependent and general material properties must be defined and linked for each element having different sectional properties.

In MIDAS/Civil, the time dependent material properties are automatically calculated accounting for member age and applied to each corresponding material. Using the ***Change Element Dependent Material Property*** function, the time dependent material properties (according to the CEB-FIP standards) and the corresponding material properties are generated and automatically linked to each corresponding element.

In order to automatically link the time dependent material properties and general material property, the section properties must be defined as DB/User Type or PSC Type.

The procedure for defining creep and shrinkage coefficients for tapered elements using the ***Change Element Dependent Material Property*** function is as follows:

1. Define creep and shrinkage material properties using CEB-FIP standards.
2. Link time dependent material properties to general material property.
3. Link the coefficients related to member dimensions (Notational Size of Member) to each element using ***Change Element Dependent Material Property*** function.

After completing the above procedure, creep and shrinkage coefficients are calculated based on the coefficients defined in Step 3 for those elements having the ***Change Element Dependent Material Property*** values during construction stage analysis. The coefficients defined in Step 1 are not applied.

Input time dependent material properties as follows:

28 day strength:	f'_c = 6000 psi (41368.6 kN/m^2)
Relative Humidity:	RH = 70 %
Notational Size:	Any value (auto-calculated later)
Concrete Type:	Normal Concrete
Timing of Formwork Removal :	3 days

Properties / Creep/Shrinkage (*Time Dependent Material (Creep & Shrinkage)*)

Name>**Creep&Shrinkage** ; Code>**CEB-FIP(1990)**

Compressive strength of concrete at the age of 28 days >**41368.6**

Relative Humidity of ambient environment (40 ~ 99)>**70**

Notational size of member>**1**

Type of cement>**Normal or rapid hardening cement (N, R)**

Age of concrete at the beginning of shrinkage>**3**

Show Result... ↵

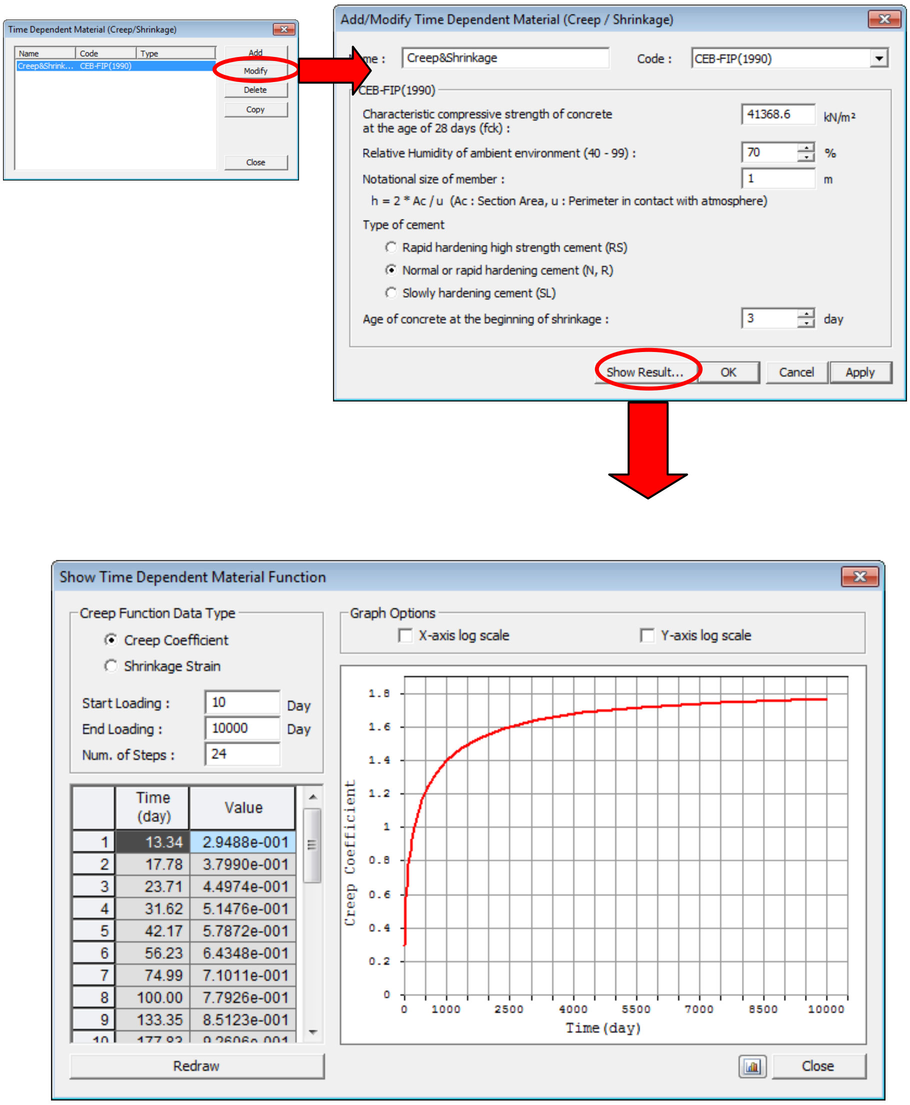


Fig. 28 Define Creep and Shrinkage Material Properties

The strength of concrete increases with time. In this example, we will use the strength gain function specified in CEB-FIP.

Properties / Comp. Strength ***Time Dependent Material (Comp. Strength)***

Name **(Comp.Strength)**; Type>**Code**

Development of Strength>Code>**CEB-FIP**

Concrete Compressive Strength at 28 Days (S28) **(41368.6)**

Cement Type (a)>**N, R : 0.25** Redraw Graph ↵

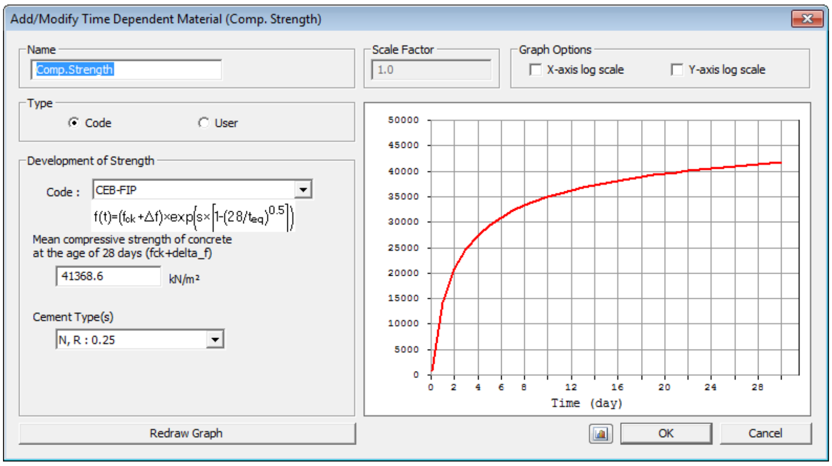


Fig. 29 Define concrete compressive strength gain function relative to time

Link time dependent material properties to the corresponding general material property.

Properties / Material Link /***Time Dependent Material Link***

Time Dependent Material Type

Creep/Shrinkage>**Creep&Shrinkage**

Comp. Strength>**Comp.Strength**

Select Material for Assign

Materials>**1:Grade C6000** > Selected Materials Add / Modify

---

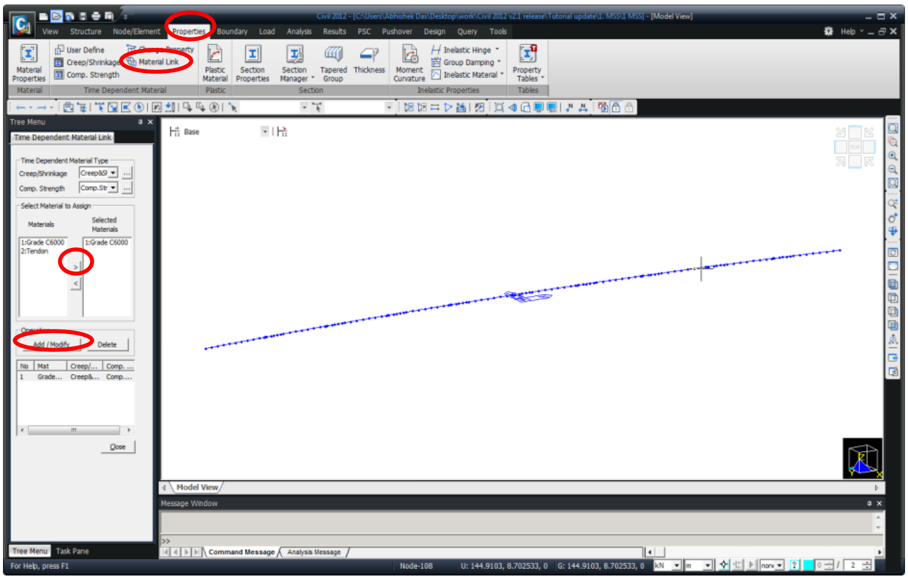


Fig. 30 Connect time dependent material properties

When the ***Change Element Dependent Material Property*** function is used to define h (Notational Size of Member), the h value defined in the Time Dependent Material is ignored. The creep and shrinkage functions for each element are then calculated using the Notational Size of Member for each element defined by the ***Change Element Dependent Material Property*** function.

If the Auto Calculate option is selected, the Notational Size of Member for each section is calculated automatically and applied to the calculation of creep and shrinkage coefficients. If the Input option is selected, the creep and shrinkage coefficients are calculated from the defined value.

Properties / Change Property ***Change Element Dependent Material Property***

Select all

Option>**Add/Replace**

Element Dependent Material

Notational Size of Member>**Auto Calculate** ; **CEB-FIP** ↵

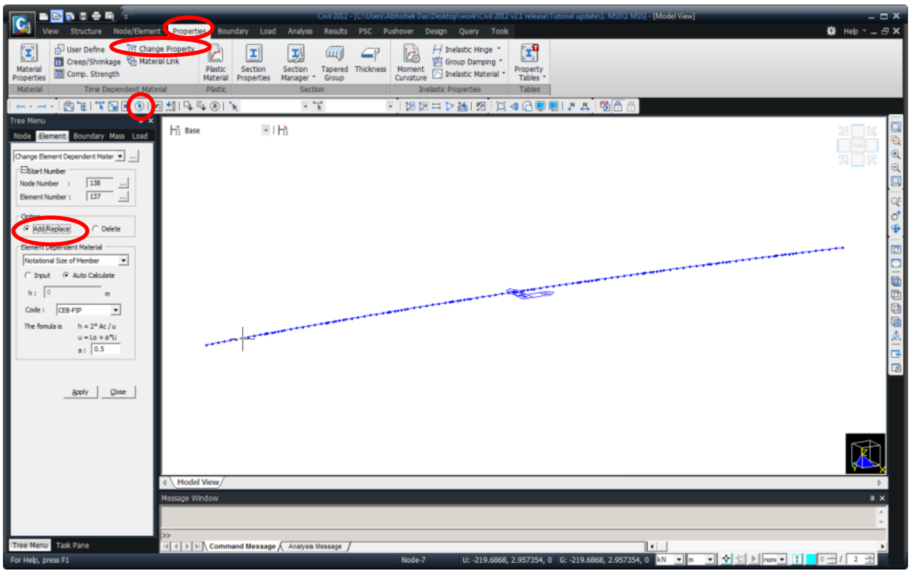


Fig. 31 Input Notational Size of Member

Perform Structural Analysis

Once the structural model and construction stages are completed, we will select the option for considering the time dependent material properties and tendon prestress loss due to elastic shortening. We will also assign the convergence condition for creep and number of iterations.

Analysis / 

Construction Stage Analysis Control

Final Stage>**Last Stage**

Analysis Option>**Include Time Dependent Effect** (on)

Time Dependent Effect

  Creep Shrinkage (on)

  Type>**Creep & Shrinkage**

  Convergence for Creep Iteration

    Number of Iterations **(5)** ; Tolerance **(0.01)**

  Auto Time Step Generation for Large Time Gap (on)

  Tendon Tension Loss Effect (Creep & Shrinkage) (on)

  Variation of Comp. Strength (on)

  Tendon Tension Loss Effect (Elastic Shortening) (on)

  Beam Section Property Changes > **Change with Tendon** (on)

  Save Output of Current Stage (Beam/Truss) (on) ↵

If Auto Time Step Generation for Large Time Gap is checked 'on', additional time steps are automatically generated for a longer duration to consider the long-term effects more precisely.

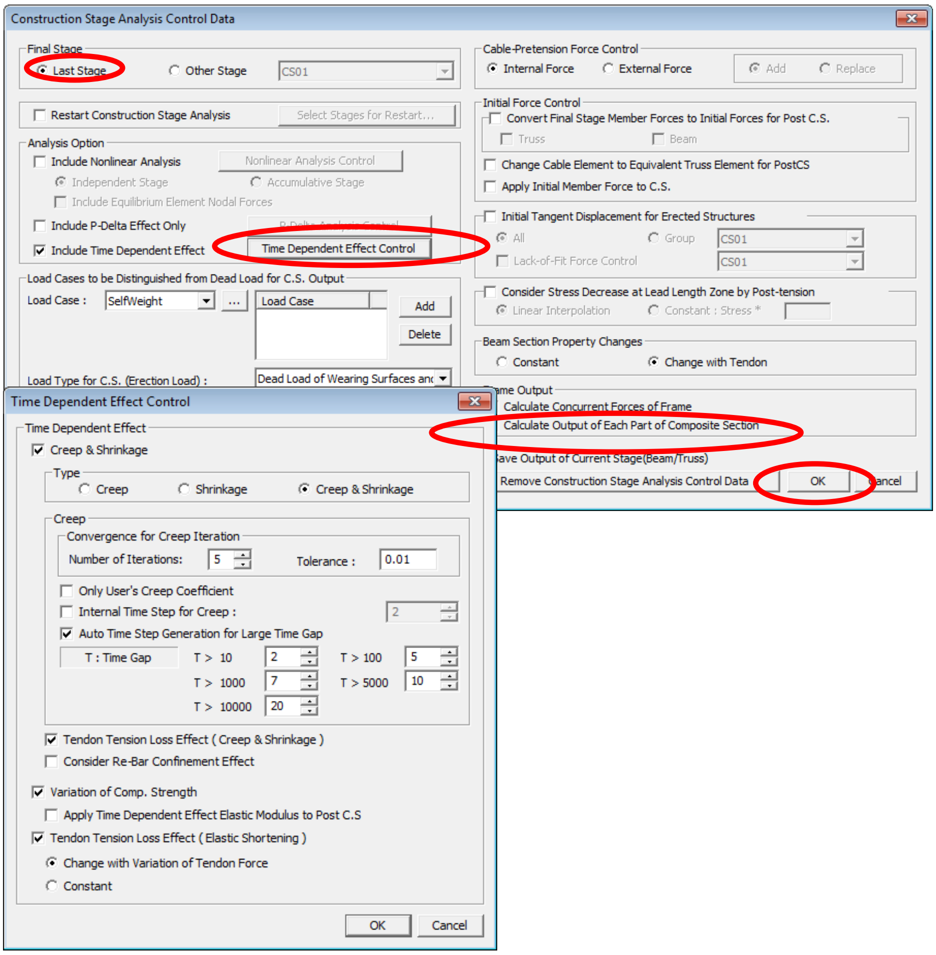
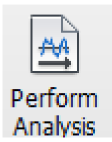


Fig. 32 Assign Construction Stage Analysis Conditions

All input is now completed. We will finally perform structural analysis.

Analysis /		*Perform Analysis*

Review Analysis Results

Refer to Results > Bridge Girder Diagrams in the On-line manual.

Refer to Results > Stage/Step History Graph in the On-line manual.

There are two methods of reviewing analysis results from construction stage analysis. One is to review accumulated stresses and displacements of all the members at each specific construction stage, and the other is to review the change in stresses and displacements in specific elements due to the preceding construction stages. Either way, the construction stage analysis results can be checked in MIDAS/Civil by means of graphs and tables.

Review Stresses and Member Forces by Graphs

In the construction stages of MSS, the maximum stress occurs in the construction stage 1, when the structural system is a simple beam. We now review the stresses at the bottom of the section for the construction stage 1.

MSS Bridge Wizard automatically generates Structure Groups for reviewing section stresses. Bridge Girder represents the element group pertaining to the main girders.

All stresses at upper/lower and left/right ends can be reviewed by selecting Axial, Bending My and Bending Mz.

If Draw Allowable Stress Line is checked on and the allowable stresses for compression and tension are specified, the allowable stresses are shown on the stress graph as dotted lines.

Stage>**CS01**

Results / ***Bridge Girder Diagrams***

Load Cases/Combinations>Step List>**First Step, User Step:1** (on)

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

Diagram Type>**Stress** ; X-Axis Type>**Node**

Bridge Girder Elem. Group>**Bridge Girder**

Components>**Combined**

Combined (Axial+Moment)>**1 (-y,+z)**

Allowable Stress Line>**Draw Allowable Stress Line** (off)

Generation Option>**Current Stage-Step** ↵

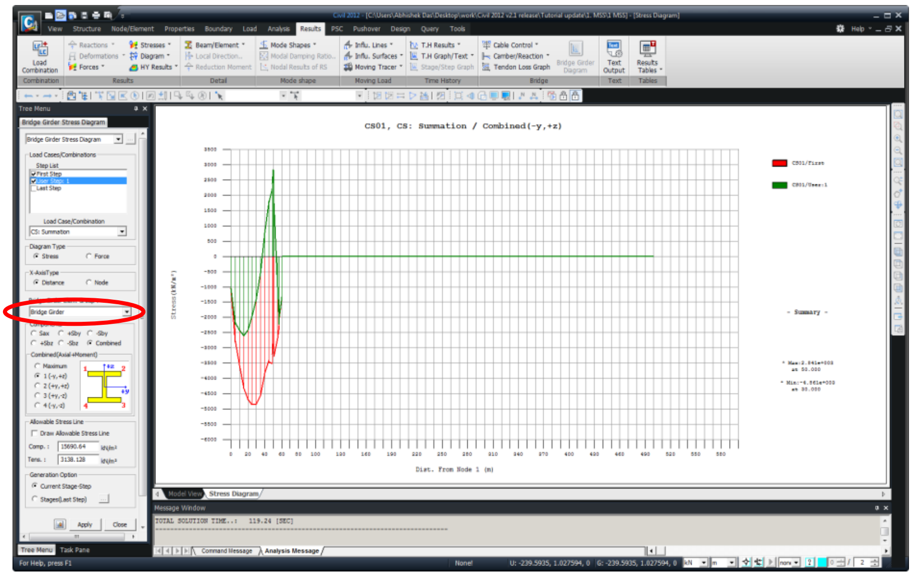


Fig. 33 Stress graph for Bottom at Construction stage 1

A more detailed stress graph for any specific part can be reviewed. Simply place the mouse on the part of interest and magnify it by dragging the mouse while keeping the mouse pressed. We can revert to the original condition by right-clicking the mouse and clicking ***Zoom Out All*** on the graph.

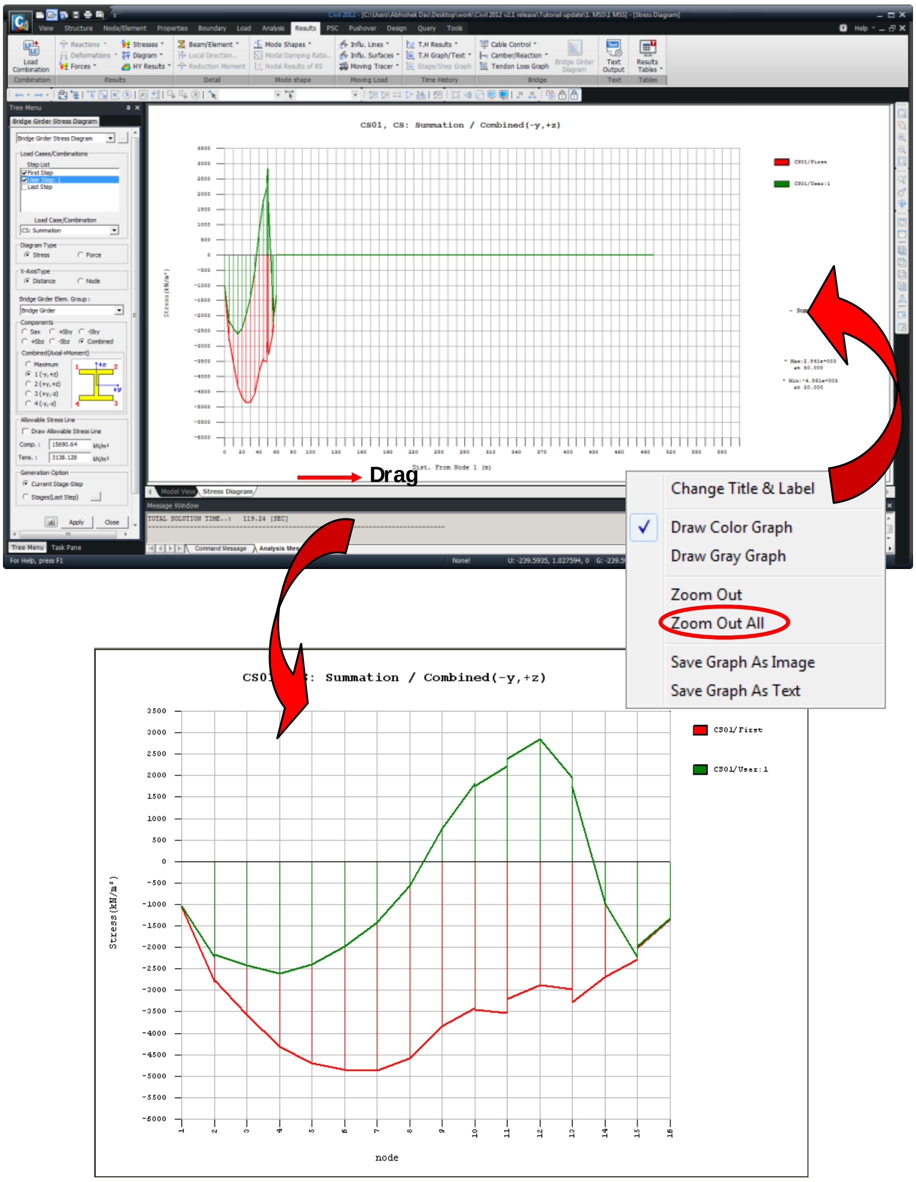


Fig. 34 Magnify Stress Graph

Review the stress changes by construction stages at the second support (i-th end of Element 11) using ***Stage/Step History Graph***.

Activate Model View. The Stage/Step History Graph menu can be used only when Model View is in an activated state.

Model View

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

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

Beam Force/Stress>Name **(Top)**; Element No. **(12)**; **Stress Point>I-Node**; Components>**Bend(+z)**

Combine Axial (on) ↵

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

Beam Force/Stress>Name **(Bot)**; Element No. **(12)**; **Stress Point>I-Node**; Components>**Bend(-z)**

Combine Axial (on) ↵

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

Check Function to Plot>**Top** (on); **Bot** (on)

Load Cases/Combinations>**Summation**

Graph Title>**Stress History** Graph

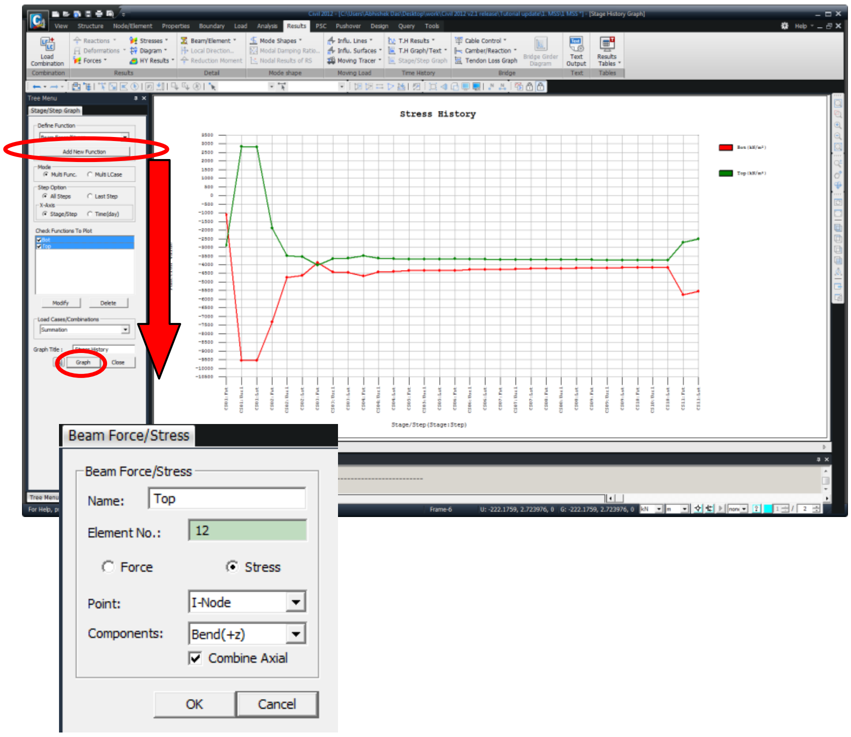


Fig. 35 Graph of Stress changes by construction stages

Context Menu can be prompted by right-clicking the mouse on the **Stage/Step History Graph**. Stress changes for each construction stage can be saved in a text file using the ***Save Graph As Text*** function.

Save Graph As Text

File Name>**StressHistory.txt** ↲

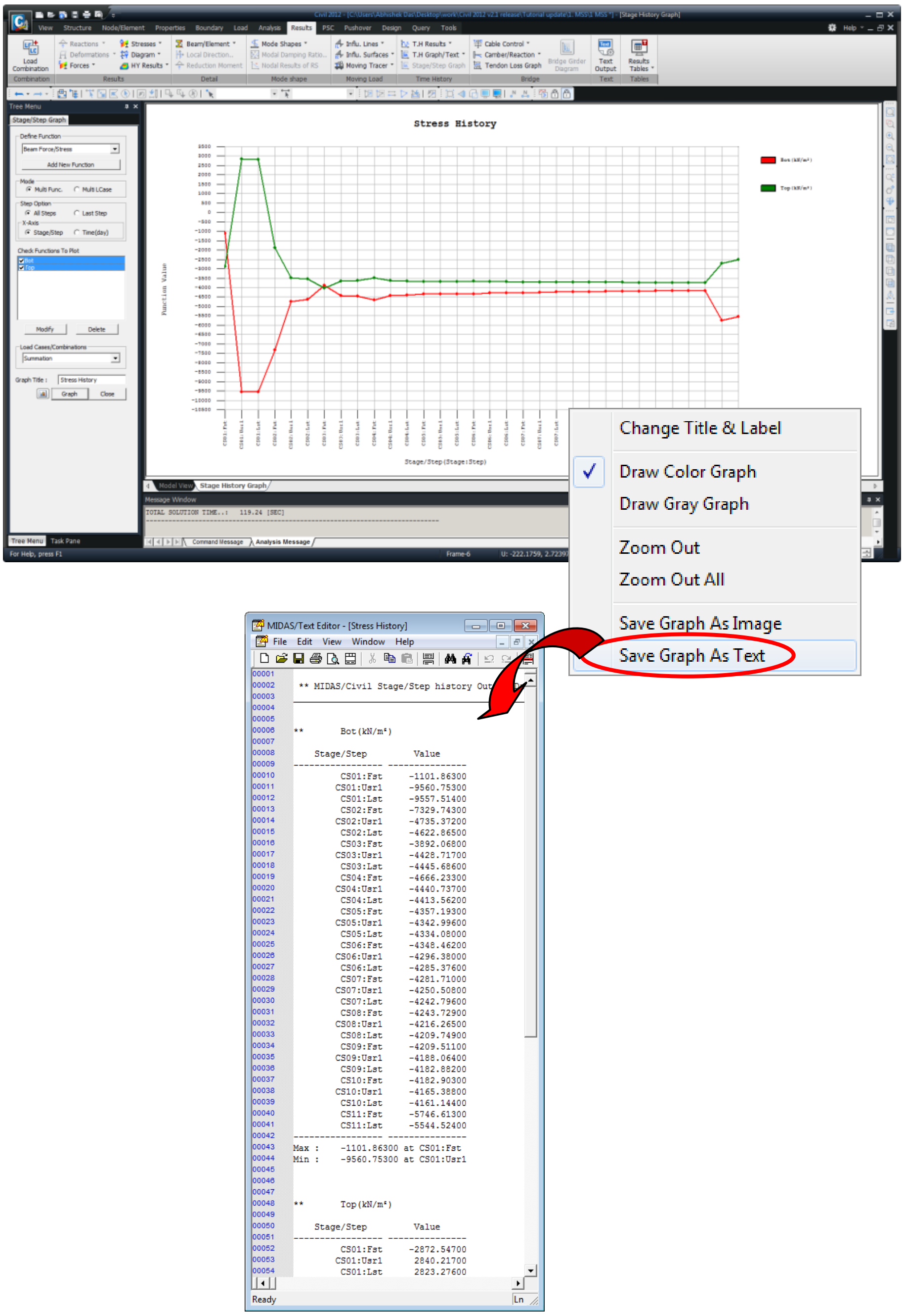


Fig. 36 Save stresses for each construction stage as a text file

Review the member force changes by construction stages at the second support (i-th end of Element 11) using ***Stage/Step History Graph***.

Model View

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

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

Beam Force/Stress

Name **(Moment)**; Element No. **(12)**; **Force**

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

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

Check Load Cases to Plot

Dead Load (on); **Tendon Primary** (on)

Tendon Secondary (on); **Creep Primary** (on)

Shrinkage Primary (on); **Creep Secondary** (on)

Shrinkage Secondary (on); **Summation** (on)

Defined Functions>**Moment**

Graph Title>**Moment** Graph

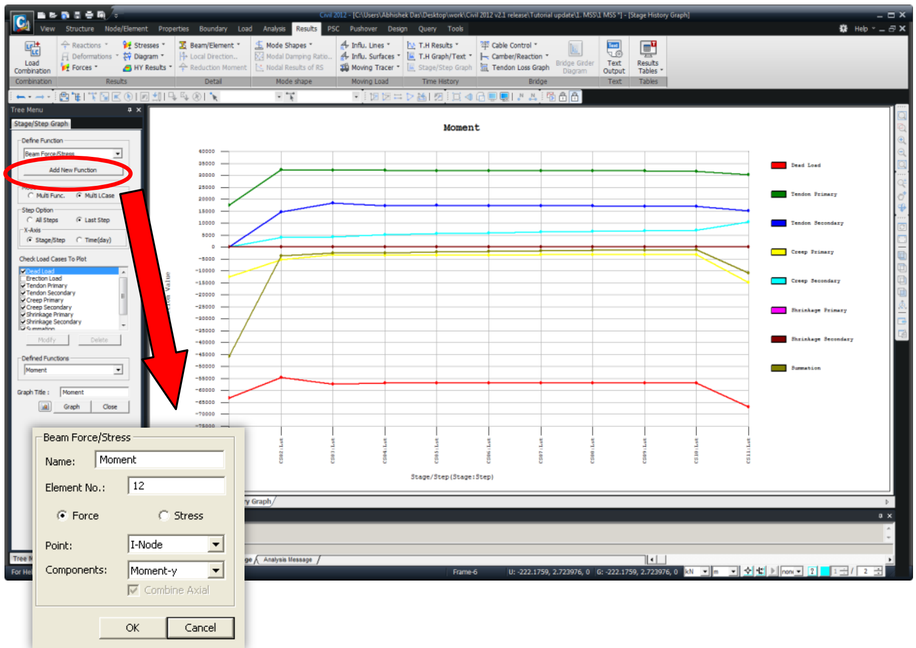


Fig. 37 Graph of Member force changes by Construction stages

Stress Review using Tables

Construction stage analysis results can be sorted by elements, loads, construction stages, output location within elements, etc. using **Records Activation Dialog**. We will now review the stress changes for each construction stage at the pier top in a spreadsheet table format.

Results / 

Results Table / Beam / ***Stress***

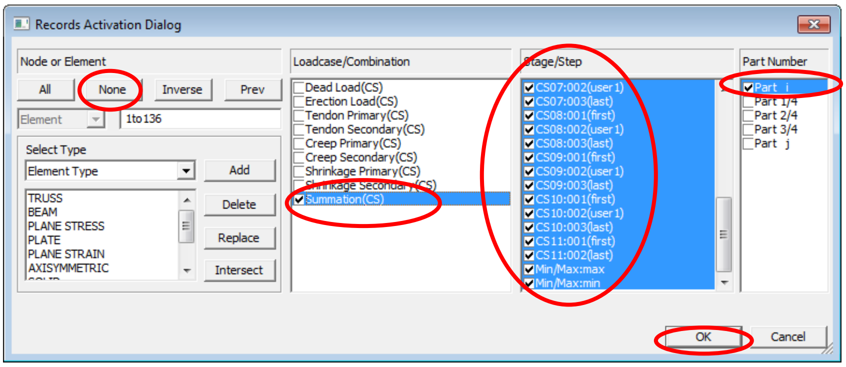
Node or Element>Element **(12)**

Loadcase/Combinations>**Summation (CS)** (on)

Stage/Step>**CS01:001(first) ~ CS11:002(last)** (on)

Part Number>**Part i** (on) ↵

All construction stages between CS01 and CS11 can be selected simultaneously by selecting CS01 and CS11 while pressing the Shift key.



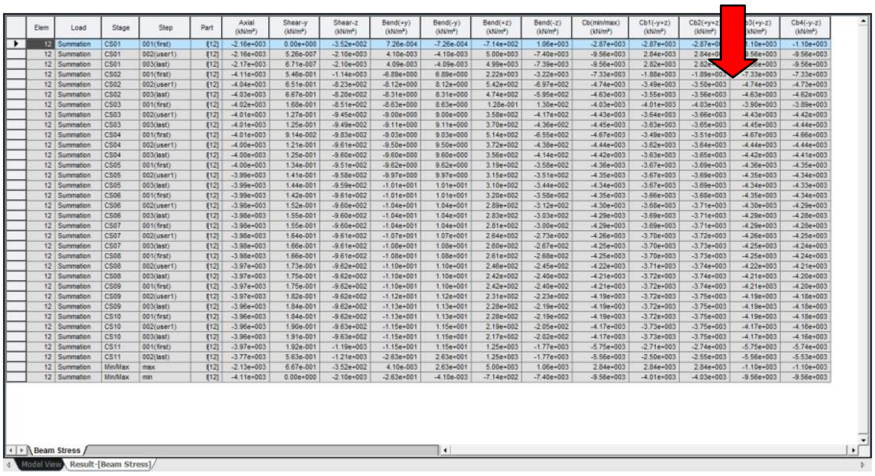


Fig. 38 Stress Table for each construction stage

Review Prestress Loss

We will now review the change in tension force as the construction stage progresses. We can only review the tendons that are contained in the current stage in the ***Tendon Time-Dependent Loss Graph*** dialog box. To review change in tension forces, first change the construction stage to the stage that contains the tendon of interest and then select the ***Tendon Time-Dependent Loss Graph*** menu. The change of tension forces caused by the construction process can be reviewed by animation by clicking the Animate button.

Results / Tendon Loss Graph ***Tendon Time-dependent Loss Graph***

Tendon>**Bot1**

Animate

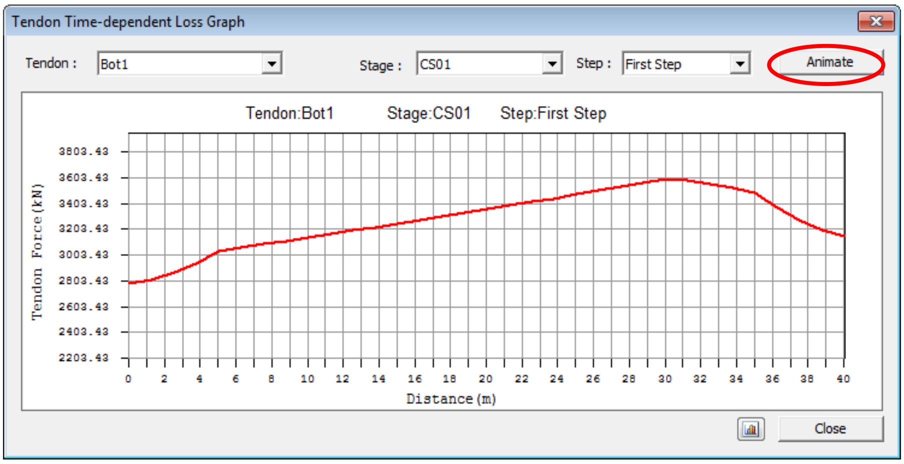


Fig. 39 Prestress Graph

Review Tendon Coordinates

In MIDAS/Civil, the tendon coordinates for each quarter station within a single element can be reviewed in a tabular form.

Results / 

Result Tables / Tendon / ***Tendon Coordinates***

Tendon Name	No	x (m)	y (m)	z (m)
Bot2	20	23.7500	2.2360	0.0620
Bot2	21	25.0000	2.2350	0.0620
Bot2	22	26.2500	2.2360	0.0620
Bot2	23	27.5000	2.2363	0.0620
Bot2	24	28.7500	2.2360	0.0620
Bot2	25	30.0000	2.2350	0.0620
Bot2	26	31.2500	2.2360	0.0620
Bot2	27	32.5000	2.2363	0.0620
Bot2	28	33.7500	2.2360	0.0620
Bot2	29	35.0000	2.2350	0.0620
Bot2	30	36.2500	2.2360	0.1146
Bot2	31	37.5000	2.2363	0.2540
Bot2	32	38.7500	2.2360	0.4520
Bot2	33	40.0000	2.2350	0.6796
CS01a01_I	0	-249.5405	0.0000	0.0000
CS01a01_I	1	0.0000	2.9485	0.9750
CS01a01_I	2	1.2500	2.9287	0.9106
CS01a01_I	3	2.5000	2.9091	0.8464
CS01a01_I	4	3.7500	2.8897	0.7829
CS01a01_I	5	5.0000	2.8705	0.7203
CS01a01_I	6	6.2500	2.8517	0.6590
CS01a01_I	7	7.5000	2.8334	0.5992
CS01a01_I	8	8.7500	2.8157	0.5414
CS01a01_I	9	10.0000	2.7988	0.4859
CS01a01_I	10	11.2500	2.7827	0.4334
CS01a01_I	11	12.5000	2.7676	0.3841
CS01a01_I	12	13.7500	2.7537	0.3388
CS01a01_I	13	15.0000	2.7412	0.2980
CS01a01_I	14	16.2500	2.7304	0.2627
CS01a01_I	15	17.5000	2.7216	0.2337
CS01a01_I	16	18.7500	2.7150	0.2123
CS01a01_I	17	20.0000	2.7112	0.2000
CS01a01_I	18	21.2500	2.7107	0.1984
CS01a01_I	19	22.5000	2.7137	0.2081
CS01a01_I	20	23.7500	2.7203	0.2295

Tendon Coordinates

Model View | Tendon Coordinates

Fig. 40 Tendon Coordinates Table

Review Tendon Elongation

Review tendon elongation values given in the table below.

Results / Results Tables Result Tables / Tendon / ***Tendon Elongation***

	Tendon Name	Stage	Step	Tendon Elongation		Element Elongation		Summation	
				Begin (m)	End (m)	Begin (m)	End (m)	Begin (m)	End (m)
▶	Bot1	CS01	001(first	0.0000	0.2214	0.0000	0.0006	0.0000	0.2220
	Bot2	CS01	001(first	0.0000	0.2217	0.0000	0.0006	0.0000	0.2223
	CS01a01_l	CS01	001(first	0.0000	0.2809	0.0000	0.0007	0.0000	0.2816
	CS01a01_r	CS01	001(first	0.0000	0.2809	0.0000	0.0007	0.0000	0.2816
	CS01a02_l	CS01	001(first	0.0000	0.2869	0.0000	0.0007	0.0000	0.2876
	CS01a02_r	CS01	001(first	0.0000	0.2869	0.0000	0.0007	0.0000	0.2876
	CS01a03_l	CS01	001(first	0.0000	0.2916	0.0000	0.0007	0.0000	0.2923
	CS01a03_r	CS01	001(first	0.0000	0.2916	0.0000	0.0007	0.0000	0.2923
	CS01b01_l	CS02	001(first	0.0000	0.2828	0.0000	0.0006	0.0000	0.2835
	CS01b01_r	CS02	001(first	0.0000	0.2828	0.0000	0.0006	0.0000	0.2835
	CS01b02_l	CS02	001(first	0.0000	0.2899	0.0000	0.0006	0.0000	0.2905
	CS01b02_r	CS02	001(first	0.0000	0.2899	0.0000	0.0006	0.0000	0.2905
	CS01b03_l	CS02	001(first	0.0000	0.2959	0.0000	0.0006	0.0000	0.2965
	CS01b03_r	CS02	001(first	0.0000	0.2959	0.0000	0.0006	0.0000	0.2965
	CS02a01_l	CS03	001(first	0.0000	0.2482	0.0000	0.0005	0.0000	0.2487
	CS02a01_r	CS03	001(first	0.0000	0.2482	0.0000	0.0005	0.0000	0.2487
	CS02a02_l	CS03	001(first	0.0000	0.2548	0.0000	0.0005	0.0000	0.2553
	CS02a02_r	CS03	001(first	0.0000	0.2548	0.0000	0.0005	0.0000	0.2553
	CS02a03_l	CS03	001(first	0.0000	0.2608	0.0000	0.0005	0.0000	0.2614
	CS02a03_r	CS03	001(first	0.0000	0.2608	0.0000	0.0005	0.0000	0.2614
	CS02b01_l	CS02	001(first	0.0000	0.2319	0.0000	0.0006	0.0000	0.2324
	CS02b01_r	CS02	001(first	0.0000	0.2319	0.0000	0.0006	0.0000	0.2324
	CS02b02_l	CS02	001(first	0.0000	0.2400	0.0000	0.0006	0.0000	0.2406
	CS02b02_r	CS02	001(first	0.0000	0.2400	0.0000	0.0006	0.0000	0.2406
	CS02b03_l	CS02	001(first	0.0000	0.2479	0.0000	0.0006	0.0000	0.2485
	CS02b03_r	CS02	001(first	0.0000	0.2479	0.0000	0.0006	0.0000	0.2485
	CS03a01_l	CS04	001(first	0.0000	0.2482	0.0000	0.0005	0.0000	0.2487
	CS03a01_r	CS04	001(first	0.0000	0.2482	0.0000	0.0005	0.0000	0.2487
	CS03a02_l	CS04	001(first	0.0000	0.2548	0.0000	0.0005	0.0000	0.2553
	CS03a02_r	CS04	001(first	0.0000	0.2548	0.0000	0.0005	0.0000	0.2553
	CS03a03_l	CS04	001(first	0.0000	0.2608	0.0000	0.0005	0.0000	0.2614
	CS03a03_r	CS04	001(first	0.0000	0.2608	0.0000	0.0005	0.0000	0.2614
	CS03b01_l	CS03	001(first	0.0000	0.2319	0.0000	0.0006	0.0000	0.2324
	CS03b01_r	CS03	001(first	0.0000	0.2319	0.0000	0.0006	0.0000	0.2324

Tendon Elongation

Model View / Tendon Coordinates / Tendon Elongation

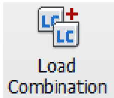
Fig. 41 Tendon Elongation Table

Review Section Forces by Load Combinations

Ultimate strength checks should be performed for load combinations of section forces due to dead load, live load, temperature changes and support settlements for the completed structure. The analysis for load combinations, other than the load combination defined by Construction Stage Load, can be performed in the PostCS and can be combined with construction stage analysis results. In this example, loads other than construction stage loads have not been defined. Hence, we will define a load combination for construction stage loads and review the section forces. First, we will define a load combination.

Change to PostCS because load combinations can only be defined and/or deleted in the Base Stage or PostCS Stage.

Stage>**PostCS**

Results / 

Combinations

Name>**Dead** ; Active> **Active** ; Type>**Add**

Load Case>**Dead Load (CS)**; Factor **(1.3)**

Load Case>**Tendon Secondary (CS)** ; Factor **(1.0)**

Load Case>**Creep Secondary (CS)** ; Factor **(1.3)**

Load Case>**Shrinkage Secondary (CS)** ; Factor **(1.3)**

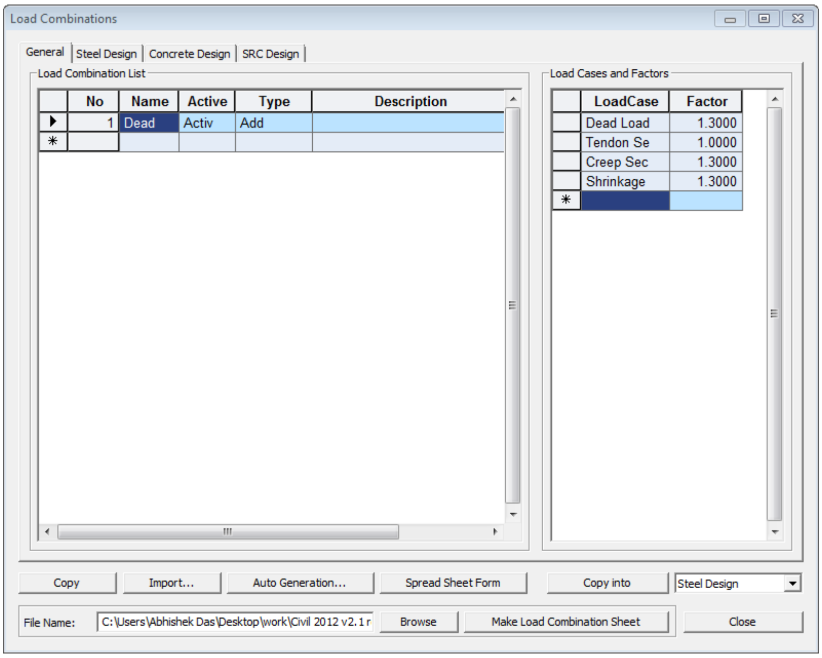


Fig. 42 Define Load combination

Review the bending moments caused by the factored load combination.

Results / Forces / Beam Diagrams...

Front View

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

Components>**My**

DisplayOptions>**5 Points** ; **Line Fill** (on) ; Scale **(1.0)**

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

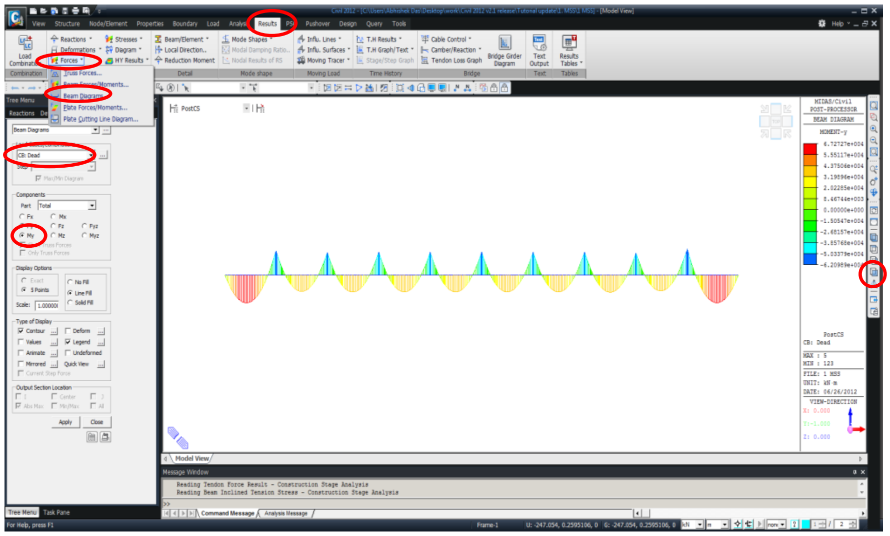


Fig. 43 Bending Moment Diagram