Advanced Application 5

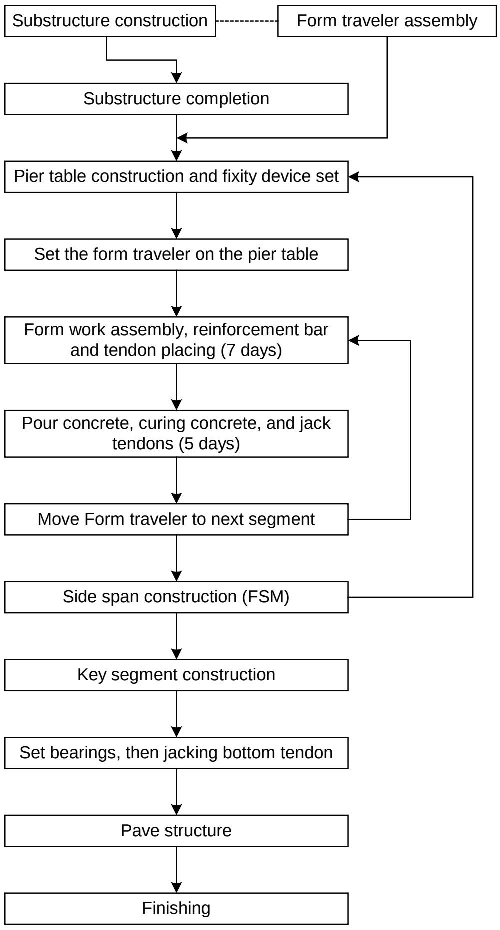
Construction Stage Analysis of a FCM Bridge using General Functions

Civil

CONTENTS

Construction Sequence and Construction Stage Analysis for FCM

In this tutorial the sequence for construction stage analysis is outlined. The example selected is a prestressed concrete box girder bridge constructed using the Free Cantilever Method (FCM). The construction stage analysis is performed using the "FCM Wizard".



Note: This example is a 3-span FCM bridge constructed with 4 Form Travelers (FT).

In the construction stage analysis, the construction sequence given below should be followed precisely. The construction stage analysis capability of MIDAS/Civil comprises an activate/deactivate concept of Structure Groups, Boundary Groups and Load Groups. The sequence of construction stage analysis for FCM is as follows:

1. Define material and section
2. Structure modeling
3. Define Structure Group
4. Define Boundary Group
5. Define Load Group
6. Input Load
7. Arrange tendons
8. Prestress tendons
9. Define time dependent material property
10. Perform structural analysis
11. Review results

Steps 2 to 8 are explained in “Construction stage analysis using FCM Wizard”. In this tutorial, the procedure for analysis of a FCM bridge from steps 1 to 8, using general functions will be explained. The procedure for steps 9 to 11 is identical to the one given in “Construction stage analysis using FCM Wizard”, and will not be repeated in this tutorial.

Assign Working Environment

To perform a construction stage analysis for a FCM bridge, open a new file (***New Project***) and save (***Save***) as 'FCM General.mcb'.

Assign the unit system as '**kN**' and '**m**'. The unit system can be changed arbitrarily during modeling, as per the convenience of the user.

> ***New Project***
> ***Save* (FCM General)**

Tools Tab / ***Unit System***

Length> **m** ; Force>**kN** ↵

The unit system selected can be changed by clicking on the unit selection button in the Status Bar located at the bottom of screen.

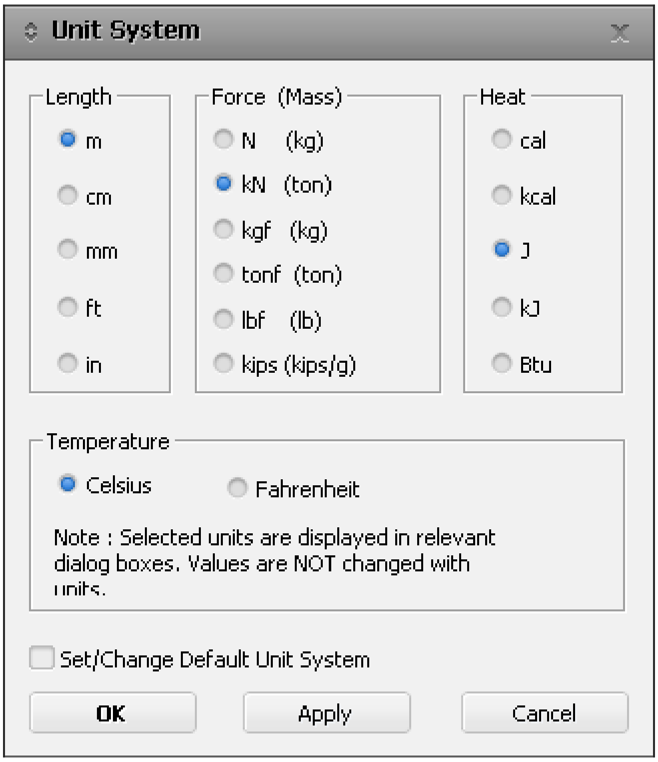
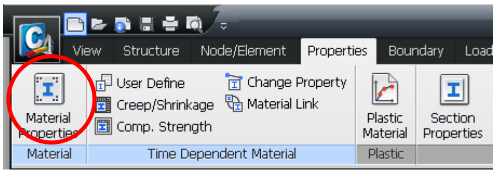


Figure 1 *Assign unit system*

Define Section and Material Properties

Define material properties for the girder, pier and tendons.

Properties Tab / ***Material Properties***



Type>**Concrete** ; Standard>**ASTM (RC)**
DB>**Grade C5000** ↵

Type>**Concrete** ; Standard> **ASTM (RC)**
DB>**Grade C4000** ↵

Name>**Tendon** ; Type>**User Defined**
Modulus of Elasticity **(2.0e8)**
Thermal Coefficient **(1.0e-5)** ↵

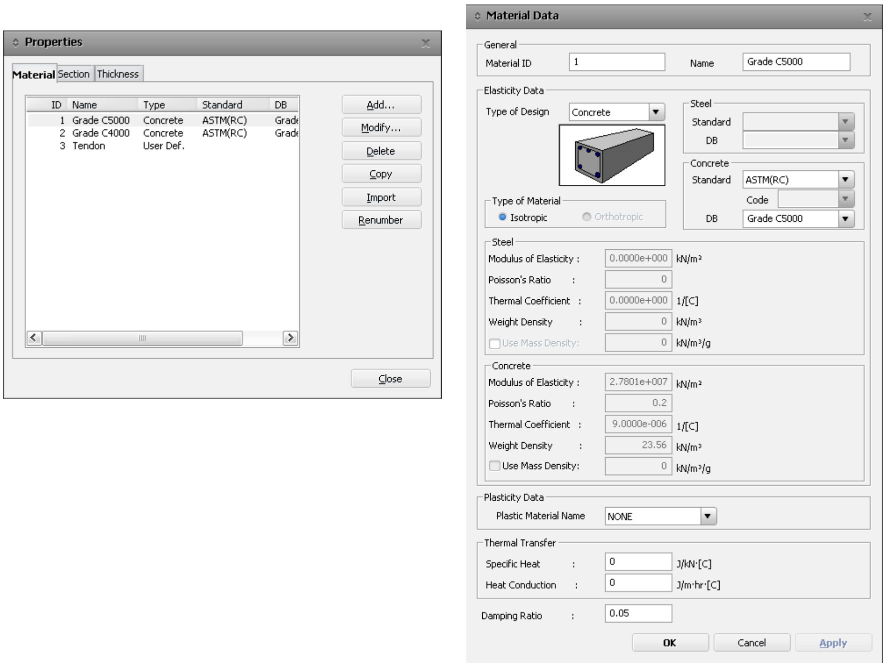


Figure 2 *Material Data input dialog box*

First, define the pier section by User Type and then define the box section. Using the Tapered Section Group function, section properties for a variable section range can easily be calculated using the definition of a variable section range (by Group) together with the input of dimensions at both ends. While using the Tapered Section Group function, it is unnecessary to define all the dimensions for each segment - only the section properties for pier and center span segment are needed.

Define the pier section.

Properties Tab / ***Section Properties***

DB/User tab

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

Section Shape>**Solid Rectangle** ; **User**>H **(1.8),** B**(8.1)** ↵

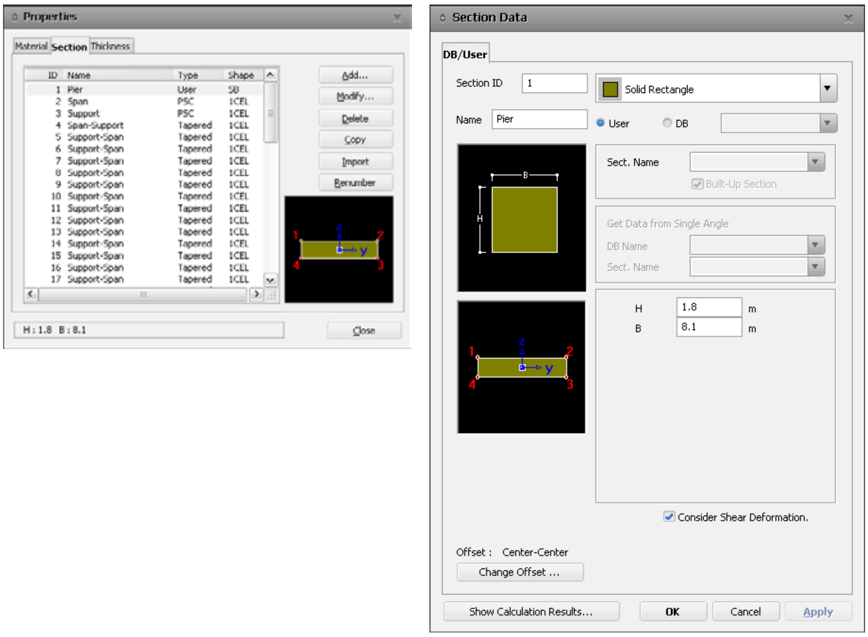


Figure 3 *Set Section dialog box*

Define the section properties of the box girder at the center span.

Properties Tab / **_Section Properties_**

PSC tab

Section ID **(2)** ; Name **(Span)**

Section Type>**1 Cell**

Joint On/Off>**JO1** (on) , **JI1** (on), **JI5** (on)

Web Thick. > Check all boxes marked Auto

Offset>**Center-Top**

Outer

HO1 **(0.25)** ; HO2 **(0.35)** ; HO3 **(2.1)**

BO1 **(2.8)** ; BO1-1 **(1.05)** ; BO3 **(3.55)**

Inner

HI1 **(0.275)** ; HI2 **(0.325)** ; HI3 **(1.59)**

HI4 **(0.25)** ; HI5 **(0.26)**

BI1 **(3.1)** ; BI1-1 **(1.35)**

BI3 **(3.1)** ; BI3-1 **(1.85)** ↵

The section offset is defined at the Center-Top because the sections are of variable shapes.

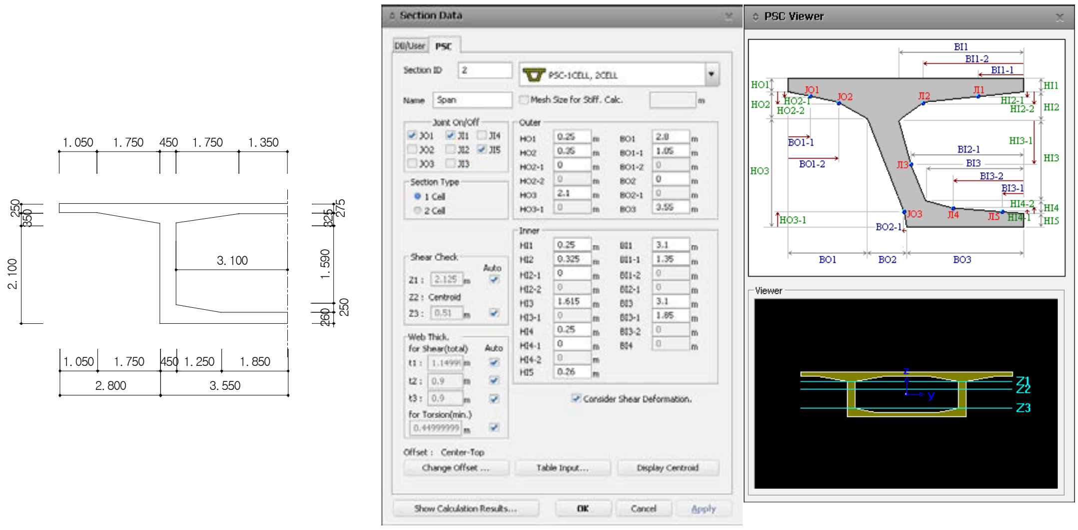


Figure 4 *Define Center Span Section*

Define the section properties of the box girder at the supports.

Properties Tab / ***Section Properties***

PSC tab

Section ID **(3)** ; Name **(Support)**

Section Type>**1 Cell**

Joint On/Off>**JO1** (on) , **JI1** (on), **JI5** (on)

Offset>**Center-Top**

Outer

HO1 **(0.25)** ; HO2 **(0.35)** ; HO3 **(6.4)**

BO1 **(2.8)** ; BO1-1 **(1.05)** ; BO3 **(3.55)**

Inner

HI1 **(0.275)** ; HI2 **(0.325)** ; HI3 **(5.3)**

HI4 **(0.25)** ; HI5 **(0.85)**

BI1 **(3.1)** ; BI1-1 **(1.35)**

BI3 **(3.1)** ; BI3-1 **(1.85)** ↵

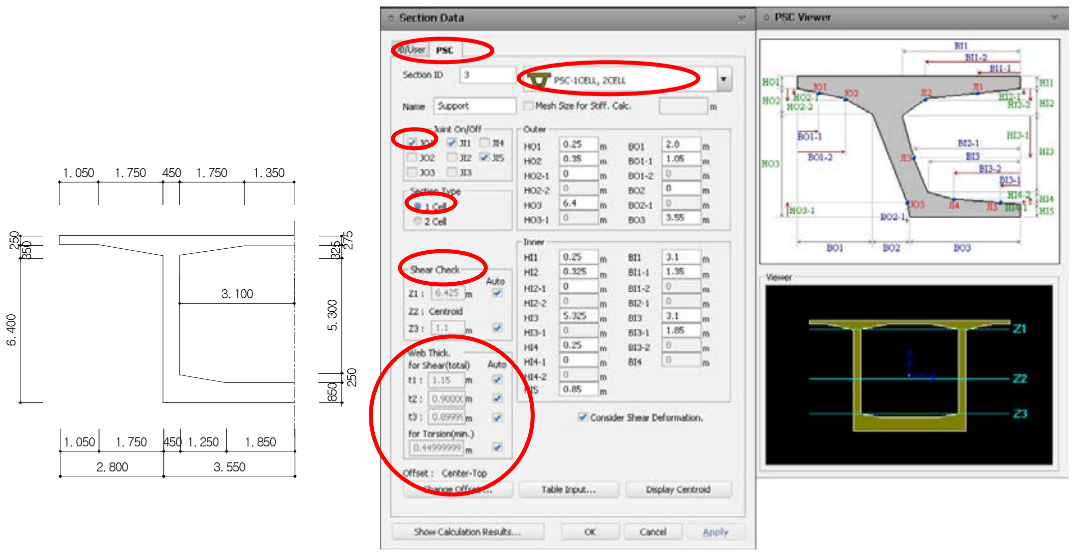


Figure 5 *Define Box Section at Supports*

After completion of section property input, generate section properties for the Tapered Type using Section ID 2 and 3.

🎧 To generate a Tapered Section Group using Tapered Type sections, predefine Tapered Type sections.

🎧 Each segment is designed as a linear tapered member because it is difficult to fabricate a curved formwork. Hence, define the section changes within a tapered segment as linear, and model each segment as one element.

Properties Tab / ***Section***

Tapered tab

Section ID **(4)** ; Name **(Span-Support)**

Section Type>**PSC-1 Cell** ; Joint On/Off>**JO1** (on)

Size-I> Import... **(Span)**

Size-J> Import... **(Support)**

y Axis Variation>**Linear** ; z Axis Variation>**Linear**

Offset>**Center-Top** Apply

Section ID **(5)** ; Name **(Support-Span)**
Section Type>**PSC-1 Cell** ; Joint On/Off>**JO1** (on)
Size-I> Import... **(Support)**
Size-J> Import... **(Span)**
y Axis Variation>**Linear** ; z Axis Variation>**Linear**
Offset>**Center-Top** ↵

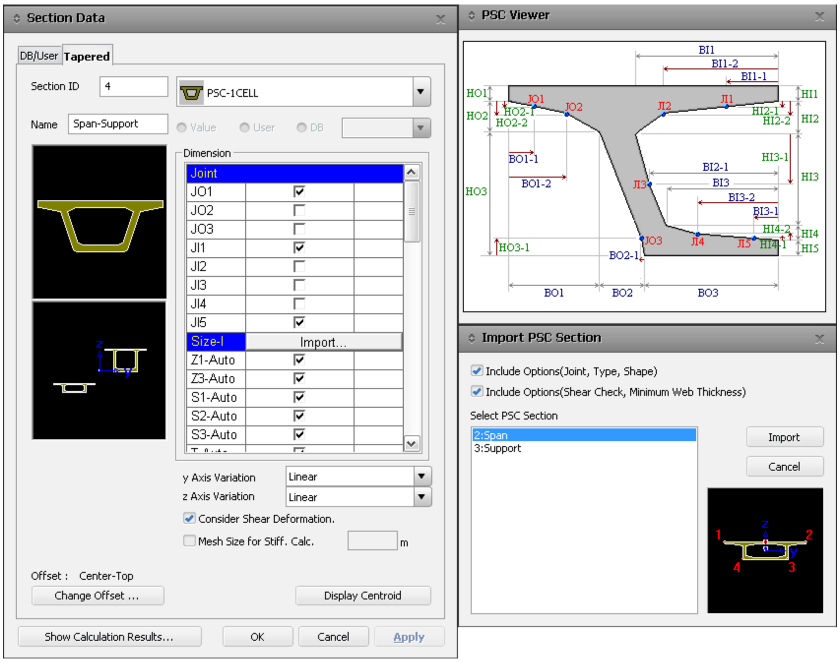


Figure 6 *Tapered Sections*

Structural Modeling

Model FCM Bridge using general functions of MIDAS/Civil.

To perform construction stage analysis, construction stages must first be defined. In MIDAS/Civil, there are two working modes - Base Stage mode and Construction Stage mode.

In Base Stage mode, any structural model, load condition and boundary condition can be defined, but the structural analysis is not performed. In Construction Stage mode, the structural analysis is performed, but the structural model input data cannot be modified or deleted except for the boundary conditions and load conditions.

Construction stages do not comprise of individual elements, boundary conditions or load conditions, but comprise of Activation and Deactivation commands for the Structure Group, Boundary Group and Load Group. In the Construction Stage mode, the boundary conditions and load conditions included in the activated Boundary Group and Load Group, respectively, can be modified or deleted.

In the analysis of FCM bridge, the loads that are applied during construction (tendon prestress, form traveler and self-weight of the segments) are complicated. Hence, the construction stages are predefined and then the load condition is defined in each construction stage. The structural systems and boundary conditions are defined in Base Stage mode.

The modeling procedure is as follows:

1. Prestressed concrete box girder modeling
2. Pier modeling
3. Define Time Dependent Material Property
4. Assign Structure Group
5. Assign Boundary Group and input boundary condition
6. Assign Load group

Prestressed Concrete Box Girder Modeling

Model the prestressed concrete box girder bridge. Model one segment as one beam element and divide the pier table at the intersection of the pier and at the center location. In the FSM zone, divide at the location of each bottom tendon anchorage.

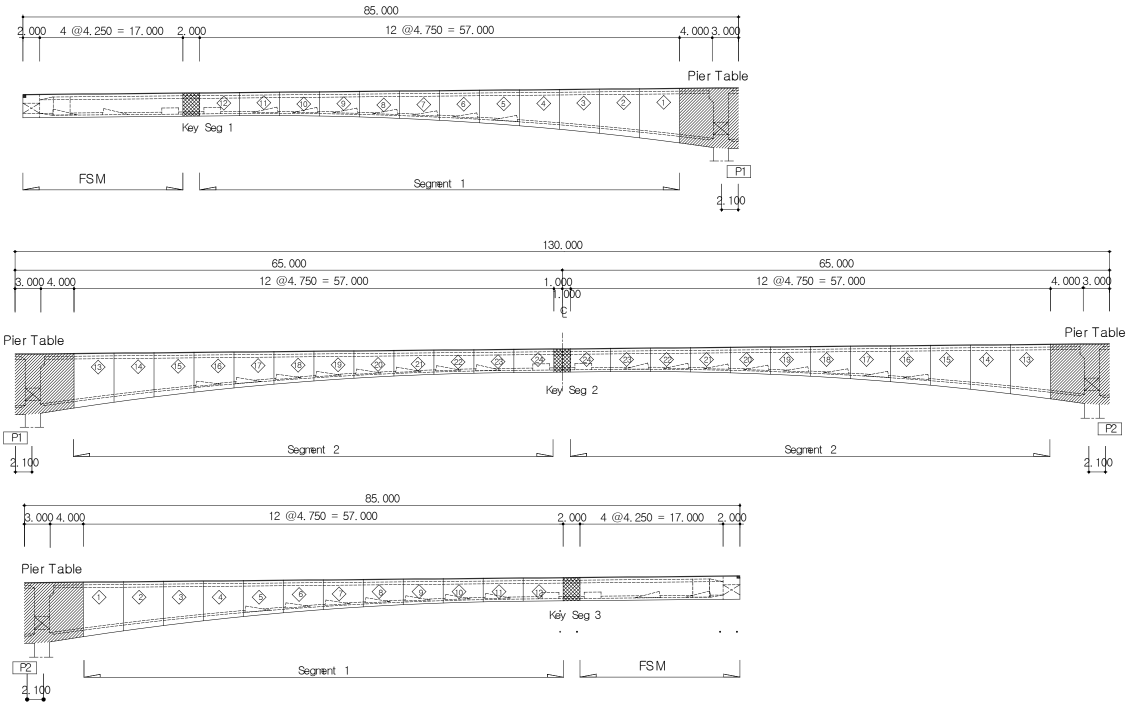


Figure 7 *Segment Division*

First generate nodes, and then model left side of the prestressed concrete box girder using the Extrude Element function (***Extrude Elements)***.

Front View, ***Auto Fitting*** (on),
View Tab >Snap> ***Point Grid Snap*** (off) ***Line Grid Snap*** (off), ***Node Snap*** (on), ***Element Snap*** (on)
Node/Element Tab > ***Create Nodes***
Coordinate (x, y, z) **(0, 0, 0)** ↵
Node/Element Tab > ***Extrude Elements***
Select All
Extrude Type>**Node → Line Element**
Element Type>**Beam** ; Material>**1: Grade C5000**
Section>**2: Span** ; Generation Type>**Translate**
Translation>**Unequal Distance** ; Axis>**x**
Distances **(2@1, 4@4.25, 2@1, 12@4.75, 4, 2@0.9, 2@1.2, 2@0.9, 4, 12@4.75, 1)** ↵

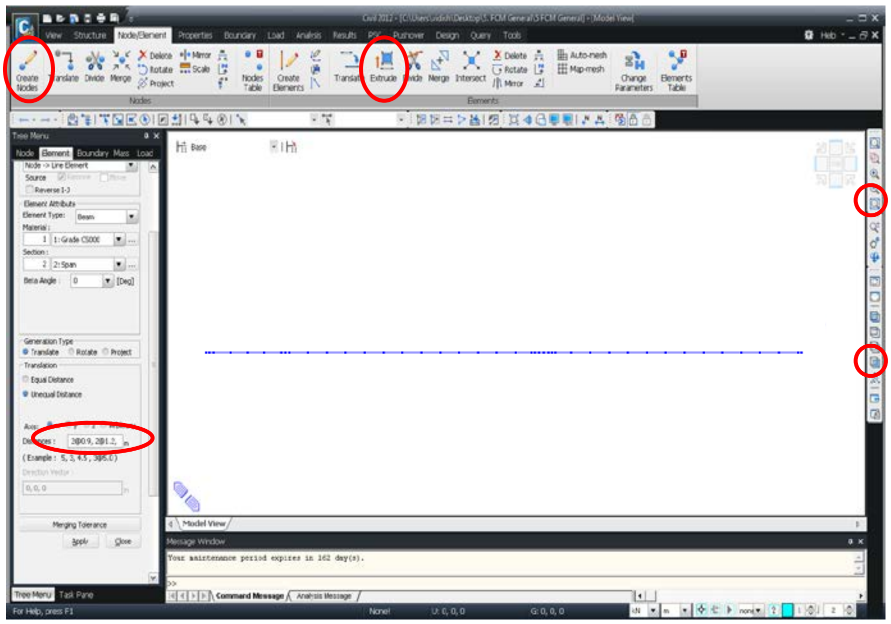


Figure 8 *Generation of left half of bridge using beam elements*

Copy the generated elements symmetrically for the right half of the bridge using the Mirror Element function (***Mirror Elements)***. Select ***Reverse Element Local*** so that local axes of the elements on the left half coincide with the local axes of the elements on the right half.

Model / Elements / ***Mirror Elements***

Select all

Mode>**Copy** ; Reflection>y-z plane x : **(150)**

Reverse Element Local (on) ↵

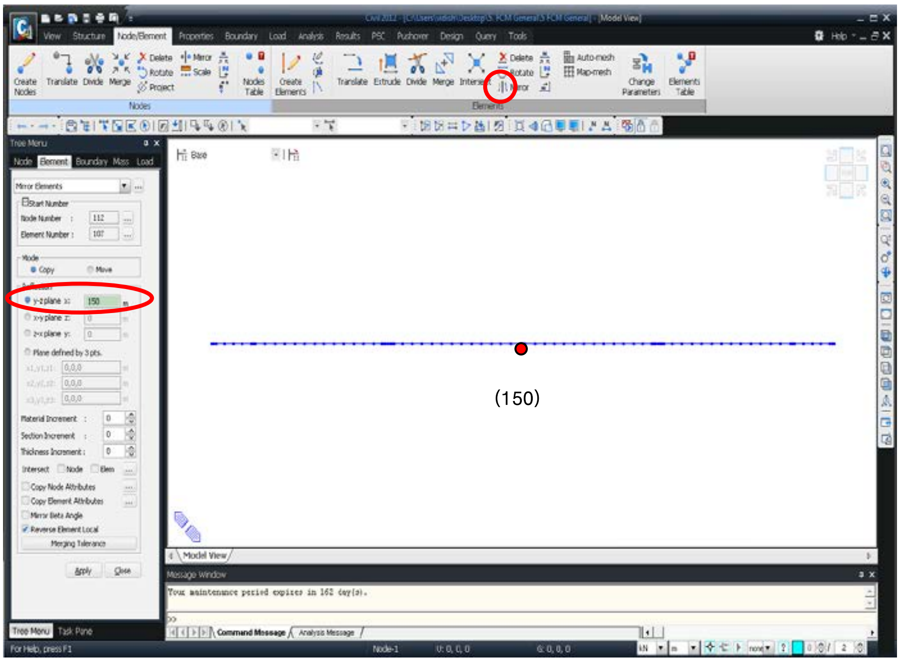


Figure 9 *Copy the beam elements symmetrically*

Change section properties for the tapered and pier table elements using **Select Identify Element (** ***Select Identity-Elements*****)** and **Works Tree** functions. Segment 12, which is connected to the key segment, is constructed as a uniform section to coincide with the formwork of the key segment. Change segments 1 to 11, and the end portions of the pier table elements, to a tapered section. The segments on the left half of the bridge are transformed from “Span” to “Span-Support” sections. The segments on the right half of the bridge are transformed from “Span” to “Support-Span” sections. The segments in the pier table are changed to “Support” sections.

Tree Menu>**Works tab**

Select Identity-Elements **(22 to 27, 63 to 68)** ↵
Works>Properties>Section>**3: Support** ***Drag&Drop***

Select Identity-Elements **(10 to 21, 69 to 80)** ↵
Works>Properties>Section>**4: Span-Support** ***Drag&Drop***

Select Identity-Elements **(28 to 39, 51 to 62)** ↵
Works>Properties>Section>**5: Support-Span** ***Drag&Drop***

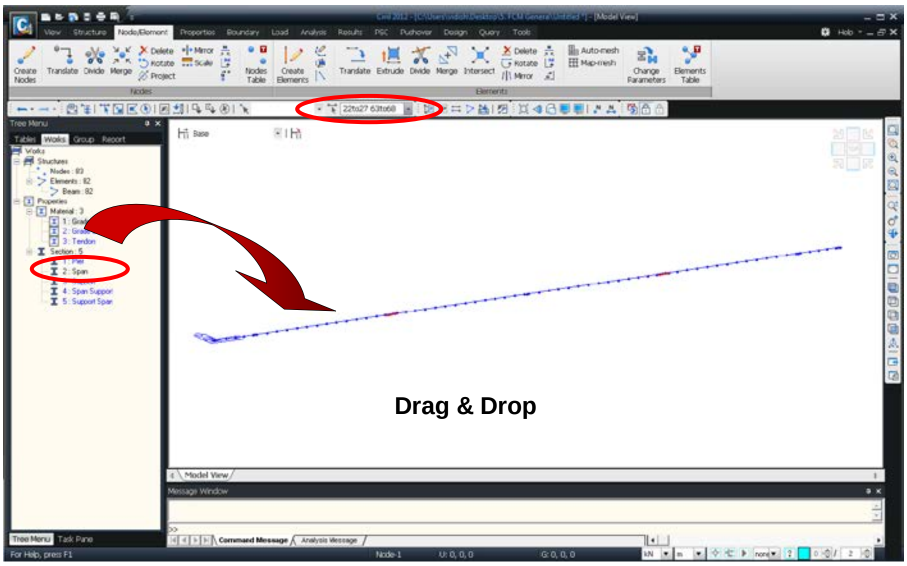


Figure 10 *Section change*

Section properties of the tapered members can be automatically calculated from the defined section properties at each end of the tapered section by assigning a Tapered Section Group.

Assign beam elements in tapered members to variable section group by the Tapered Section Group function (***Tapered Section Group)***.

Properties Tab / ***Tapered Section Group***

Iso View, ***Hidden*** (on)

Group Name **(1stspan)** ; Element List **(10 to 21)**

Section Shape Variation>z-Axis>**Polynomial (2.0)**

Symmetric Plane>From>**i** ; Distance **(0)** Add

Group Name **(2ndspan1)** ; Element List **(28 to 39)**

Section Shape Variation>z-Axis>**Polynomial (2.0)**

Symmetric Plane>From>**j** ; Distance **(0)** Add

Group Name **(2ndspan2)** ; Element List **(69 to 80)**

Section Shape Variation>z-Axis>**Polynomial (2.0)**

Symmetric Plane>From>**i** ; Distance **(0)** Add

Group Name **(3rdspan)** ; Element List **(51 to 62)**

Section Shape Variation>z-Axis>**Polynomial (2.0)**

Symmetric Plane>From> **j** ; Distance **(0)** Add

Select all the 4 Group Names

Click Convert to Tapered Section...

New Start Section Number>**1**

Use New Section Name Suffix>(on)

Start Suffix Number>**1** ↵

Click Close

Select Polynomial and 2.0 because the section height changes in a parabolic form.

In Tapered Section Group, the parabola function is determined uniquely by the defined coordinates of two points on the parabola and the center point. Since the j end of segment 12 is the center point of the parabola, select the i end and input a zero distance.

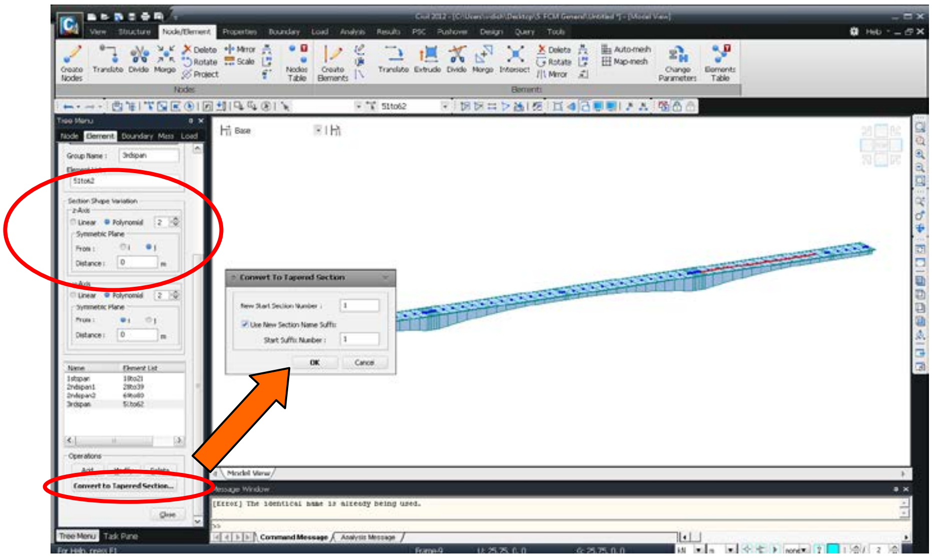


Figure 11 *Assign tapered section group*

Pier Modeling

After copying the nodes of the prestressed concrete box girder, model the pier using the Extrude Element function (***Extrude Elements)***. To model the 40 m high pier, divide the pier length into six equal length elements.

Hidden (off), ***Front View***
Node/Element Tab / ***Translate Nodes***
***Select Identity-Nodes* (23, 27, 65, 69)** ↵
Mode>**Copy** ; Translation>**Equal Distance**
dx, dy, dz **(0, 0, -7)** ; Number of Times **(1)** ↵
Model / Elements / ***Extrude Elements***
Select Recent Entities
Extrude Type>**Node → Line Element**
Element Type>**Beam** ; Material>**2: Grade C4000**
Section>**1: Pier** ; Generation Type>**Translate**
Translation>**Equal Distance**
dx, dy, dz **(0, 0, -40/6)** ; Number of Times **(6)** ↵

Since the upper center point of the box section is used as the base of the box girder model, copy the nodes to a distance of –7 m (total height of support section) in the Z-direction.

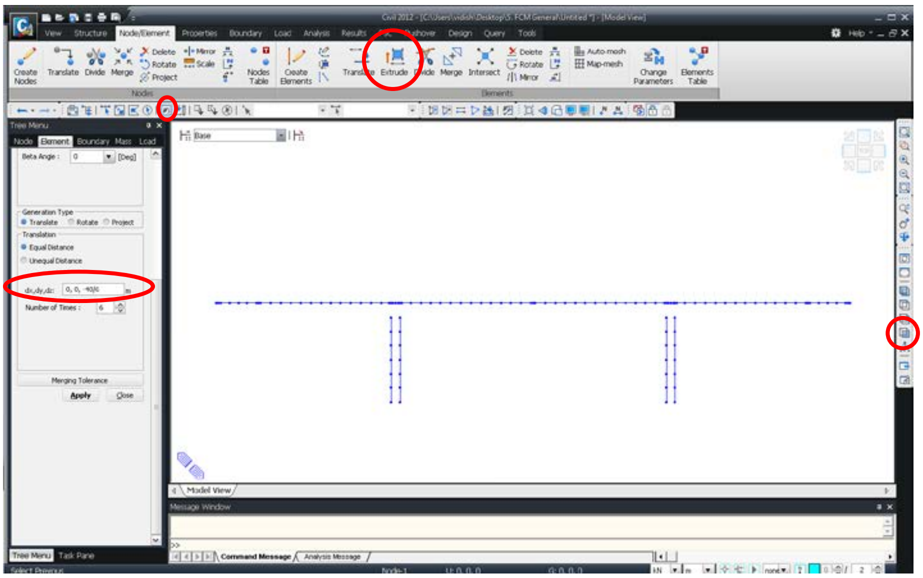


Figure 12 *Generate piers*

Assign Structure Group

Figure 13 shows the construction sequence and expected duration for each construction stage. As shown in the figure, there is a 60-day difference in construction schedule between Piers 1 and 2. Hence, there will also be a 60-day difference between both elements when the key segments are being constructed.

Increase the age of some elements by Time Load using the Construction Stage function. A detailed explanation can be found in "Time Dependent Analysis - Define and Composition of Construction Stages in the "Analysis of Civil Structures" manual.

It will be assumed that both piers are constructed at the same time and both cantilevers are constructed through the same stages before the key segment construction. And just before the key segment construction, the age of one cantilever will be increased. Define the elements constructed at the same time as each group by defining Structure Group because the generation and deletion of elements will be defined using the activation and deactivation command in Construction Stage function.

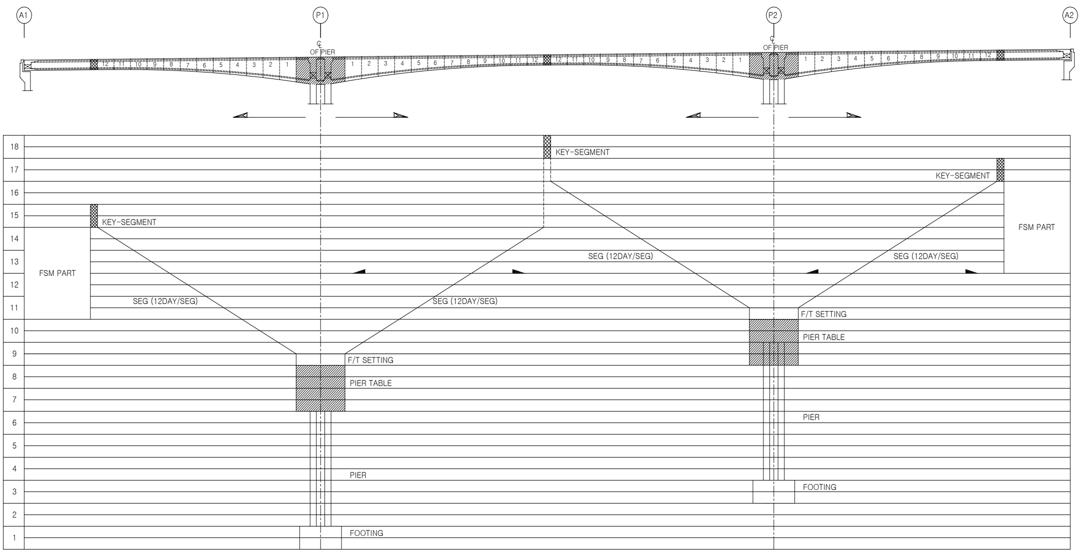


Figure 13 *Construction sequence*

Generate Structure Group

Tree Menu > ***Group Tab***

Structure Group / ***Define Structure Group (Right Click > New…)***

Name **(Pier)** ; Suffix **(1to2)** Add

Name (**PierTable**) ; Suffix (**1to2**) Add

Name **(P1Seg)** ; Suffix **(1to12)** Add

Name **(P2Seg)** ; Suffix **(1to12)** Add

Name **(KeySeg)** ; Suffix **(1to3)** Add

Name **(FSM)** ; Suffix **(1to2)** Add

By appending suffix numbers to Name, multiple Structure Groups can be generated simultaneously.

Generated Structure Group can be confirmed using the Group Tab and Tree Menu.

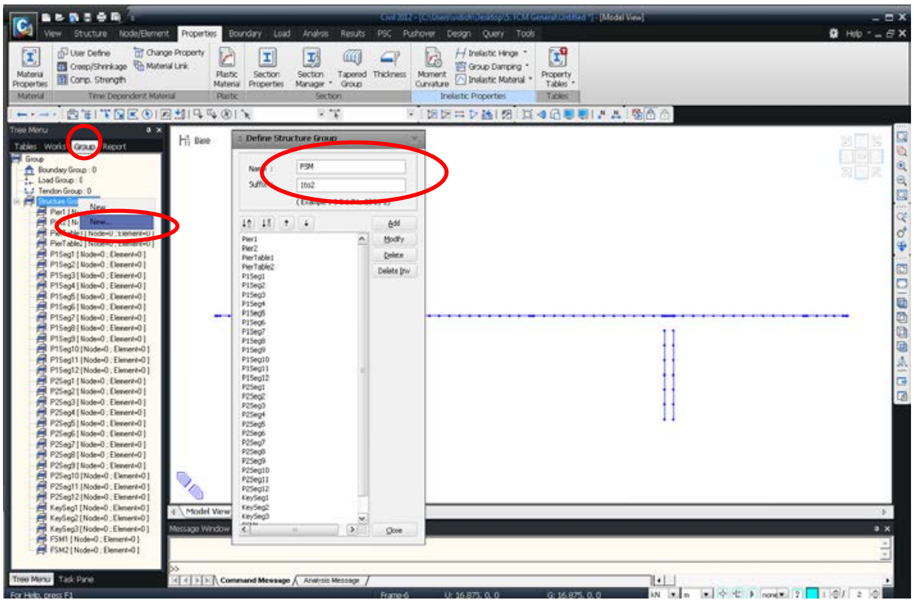


Figure 14 *Element Group Generation*

Assign beam elements to Structure Groups using **Select Identity-Element (** ***Select Identity-Elements)*** and the **Works Tree** functions.

Tree Menu>**Group tab**

***Select Identity-Elements* (83to103by4 84to104by4)** ↵

Group>Structure Group>**Pier1** ***Drag&Drop***

***Select Identity-Elements* (85to105by4 86to106by4)** ↵

Group>Structure Group>**Pier2** ***Drag&Drop***

***Select Identity-Elements* (21to28)** ↵

Group>Structure Group>**PierTable1** ***Drag&Drop***

***Select Identity-Elements* (62to69)** ↵

Group>Structure Group>**PierTable2** ***Drag&Drop***

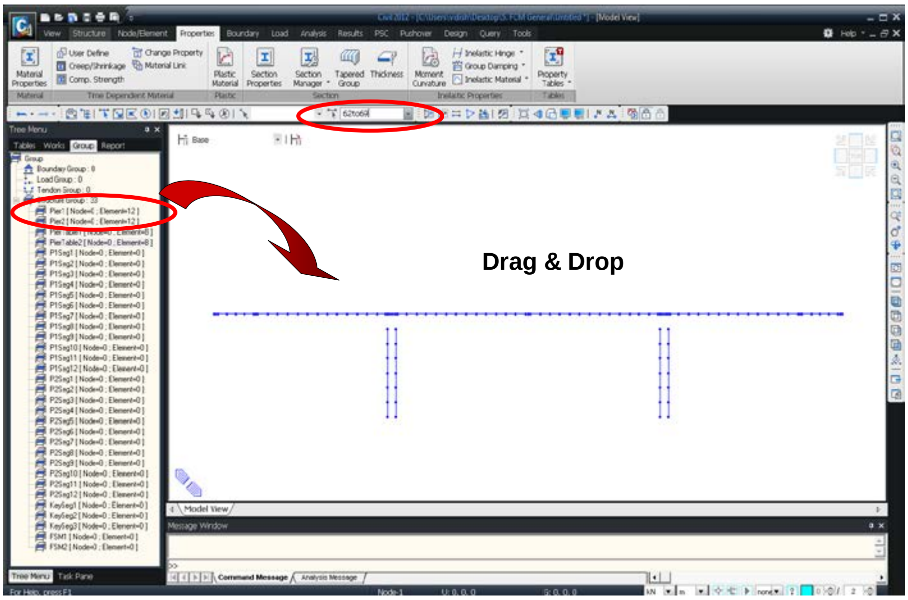


Figure 15 *Structure Group arrangement*

Assign corresponding beam elements to the other remaining Structure Groups. by referring to Table 1.

Table 1 Element group arrangement

Element Group	Element Number	Element Group	Element Number
P1Seg1	20, 29	P2Seg4	58, 73
P1Seg2	19, 30	P2Seg5	57, 74
P1Seg3	18, 31	P2Seg6	56, 75
P1Seg4	17, 32	P2Seg7	55, 76
P1Seg5	16, 33	P2Seg8	54, 77
P1Seg6	15, 34	P2Seg9	53, 78
P1Seg7	14, 35	P2Seg10	52, 79
P1Seg8	13, 36	P2Seg11	51, 80
P1Seg9	12, 37	P2Seg12	50, 81
P1Seg10	11, 38	KeySeg1	7, 8
P1Seg11	10, 39	KeySeg2	41, 82
P1Seg12	9, 40	KeySeg3	48, 49
P2Seg1	61, 70	FSM1	1~6
P2Seg2	60, 71	FSM2	42~47
P2Seg3	59, 72		

Define Boundary Groups and Input Boundary Conditions

Corresponding groups can be selected by double-clicking a particular group in the Group Tree.

After completion of modeling, confirm the Structure Groups for each segment.

Input the boundary conditions for the generated model. In construction stage analysis, all information required in the structural analysis, such as elements, loads and boundary conditions, are activated/deactivated using the Group concept. To input boundary conditions, define a Boundary Group.

Group tab

Group>Boundary Group>**New** (**BC_Pier)**

Group>Boundary Group>**New (BC_FsmLeft)**

Group>Boundary Group>**New (BC_FsmRight)**

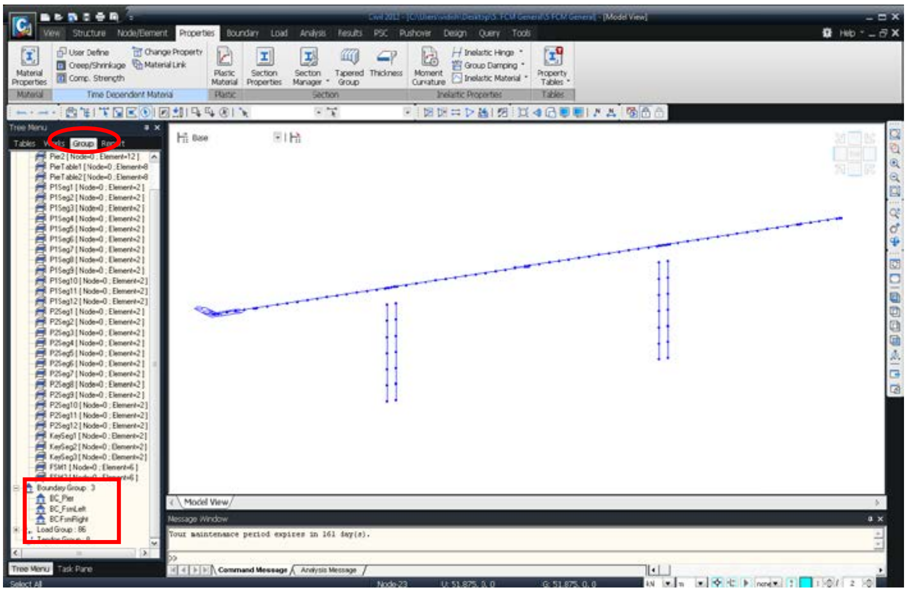


Figure 16 *Define Boundary Group*

Define boundary conditions. Define fixity condition at the bottom of the pier and longitudinal roller condition at both ends of box girder.

Boundary Tab / ***Define Supports***

Select Single (Nodes : **1**)
Boundary Group Name> **BC_FsmLeft**
Support Type>**Dy** (on), **Dz** (on), **Rx** (on) and **Rz** (on) ↵

Select Single (Nodes : **43**)
Boundary Group Name> **BC_FsmRight**
Support Type>**Dy** (on), **Dz** (on), **Rx** (on) and **Rz** (on) ↵

Select Window (Nodes : **108 ~ 111**)
Boundary Group Name> **BC_Pier**
Support Type>**D-All** (on) and **R-All** (on) ↵

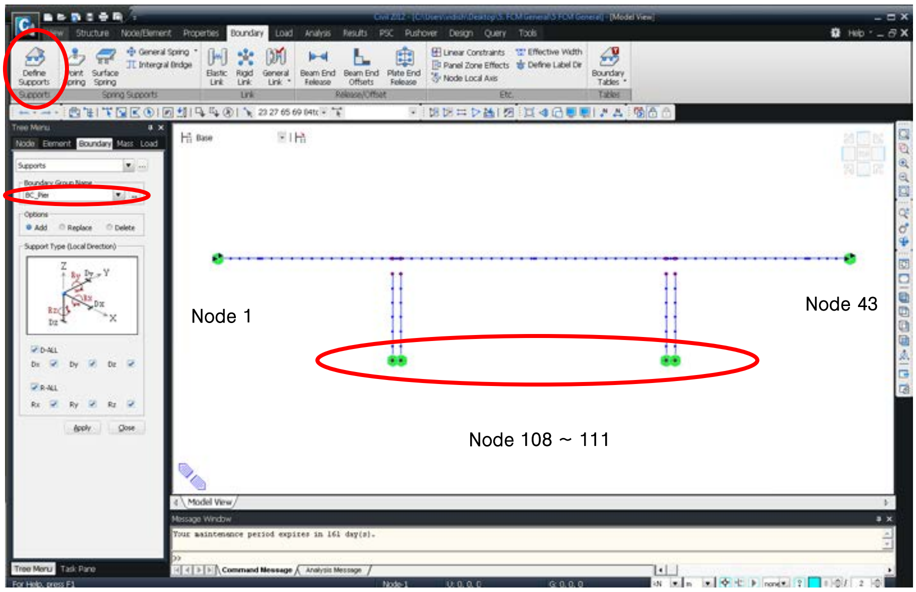


Figure 17 *Define Boundary Conditions*

Assign multiple rigid link conditions simultaneously by selecting Copy Rigid Link and inputting the spacing.

Connect the pier and box girder by Elastic Link - Rigid Link Type to ensure the monolithic behavior at the intersection point.

Boundary Tab / ***Elastic Link***

Boundary Group Name>**BC_Pier**

Link Type>**Rigid Link**

Copy Elastic Link (on)

Axis>**x** ; Distance **(4.2, 125.8, 4.2)**

2 Nodes **(84, 23)**

Zoom Fit

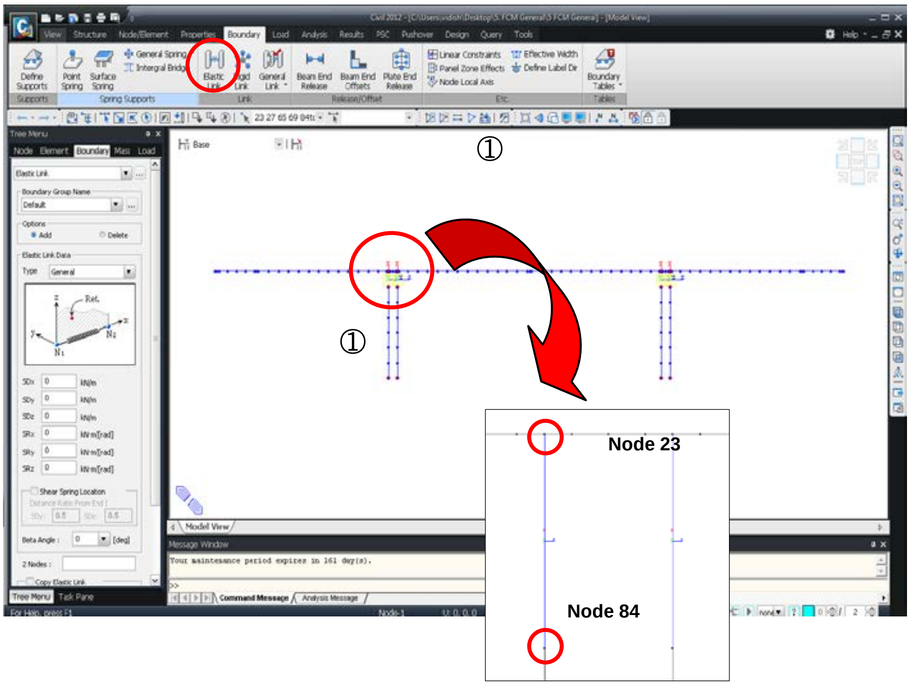


Figure 18 *Elastic Connection between the pier and girder*

Assign Load Groups

There are four types of loads in the construction stage analysis. They are the self-weight of structure, tendon prestress, form traveler load and the self-weight of wet concrete. After the structure self-weight is activated, the self-weights of the activated Structure Group are automatically considered during analysis. Therefore, only the balance three types of loads need to be inputted at each construction stage. Static loads in each construction stage are as follows:

- Self-weight of the activated elements at initial age
- Prestress for the activated elements at initial age (PS)
- Form traveler load acting on the cantilever ends of activated elements (FT)
- Self-weight of wet concrete on the formwork (WC)
- Time Load for Construction Stage to account for aging effect
- Superimposed dead loads (wearing coat, parapet, railings, etc.)

"Time Loads for Construction Stage" function has the capability to advance the time for a specific element. Hence, using this function, the effect of creep and shrinkage can be calculated. The technique on how to consider time difference between pier tables by "Time Loads for Construction Stage" is described in "Define Construction Stage".

Define load conditions for each load.

Load Tab >Static Loads Option > Static Load Cases

Name **(Self)** ; Type > Construction Stage Load Add

Name **(PS)** ; Type > Construction Stage Load Add

Name **(FT)** ; Type > Construction Stage Load Add

Name **(WC)** ; Type > Construction Stage Load Add

Name **(Time)** ; Type > Construction Stage Load Add

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

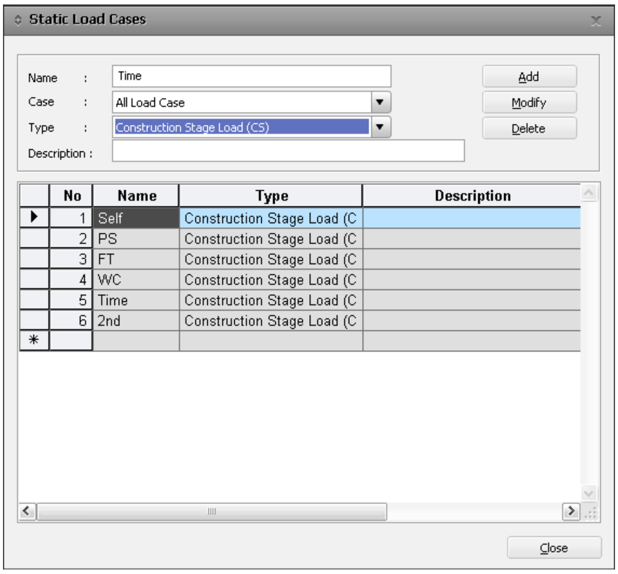


Figure 19 *Define static load cases*

Define load group for each load condition.

Group

Group Tab / ***Right Click on Load Group (New…)***

Name (**Self**) Add

Name (**PS-PierTable**) ; Suffix (**1to2**) Add

Name (**PS-P1Seg**) ; Suffix (**1to12**) Add

Name (**PS-P2Seg**) ; Suffix (**1to12**) Add

Name (**PS-KeySeg**) ; Suffix (**1to3**) Add

Name (**FT-PierTable**) ; Suffix (**1to2**) Add

Name (**FT-P1Seg**) ; Suffix (**1to11**) Add

Name (**FT-P2Seg**) ; Suffix (**1to11**) Add

Name (**FT-KeySeg**) ; Suffix (**1to3**) Add

Name (**WC-P1Seg**) ; Suffix (**1to12**) Add

Name (**WC-P2Seg**) ; Suffix (**1to12**) Add

Name (**WC-KeySeg**) ; Suffix (**1to3**) Add

Name (**TimeLoad**) Add

Name (**2nd**) Add

Generated Load Group can be confirmed by using **Group Tab**, **Tree Menu**.

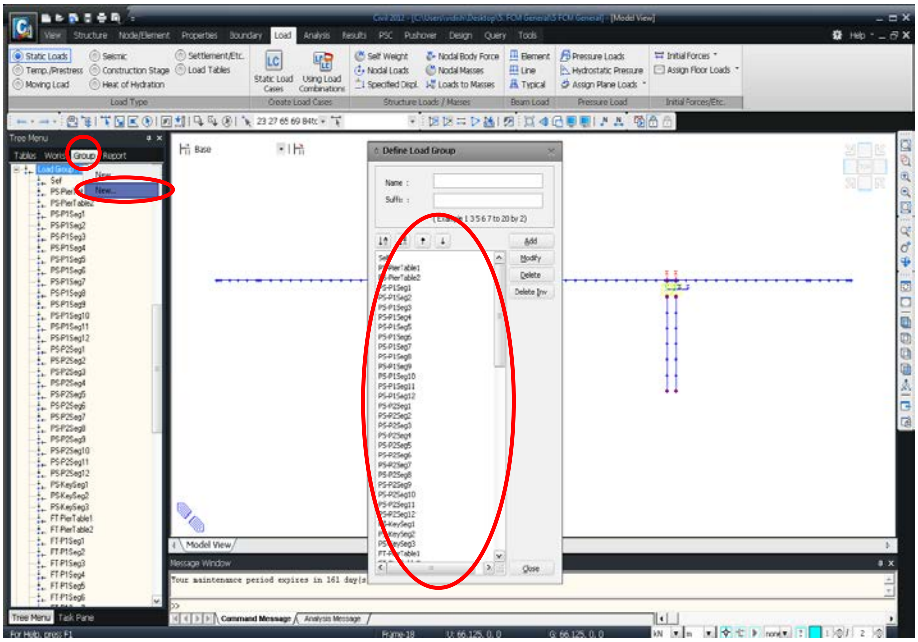


Figure 20 *Defined Load Group*

Define and Arrange Construction Stages

Define Construction Stages

Define the construction stage arrangement and pier table, construction of segments 1 to 12 and construction of key segments 1 to 3 (See Fig. 21). The construction of FCM Bridge is completed by constructing each segment, side span key segments and finally the center span key segment. There are no changes in boundary conditions during construction in this example, since the bridge is a frame-type FCM bridge.

In addition to the frame type FCM Bridge, there are FCM bridges with internal hinges and a continuous girder type FCM Bridge.

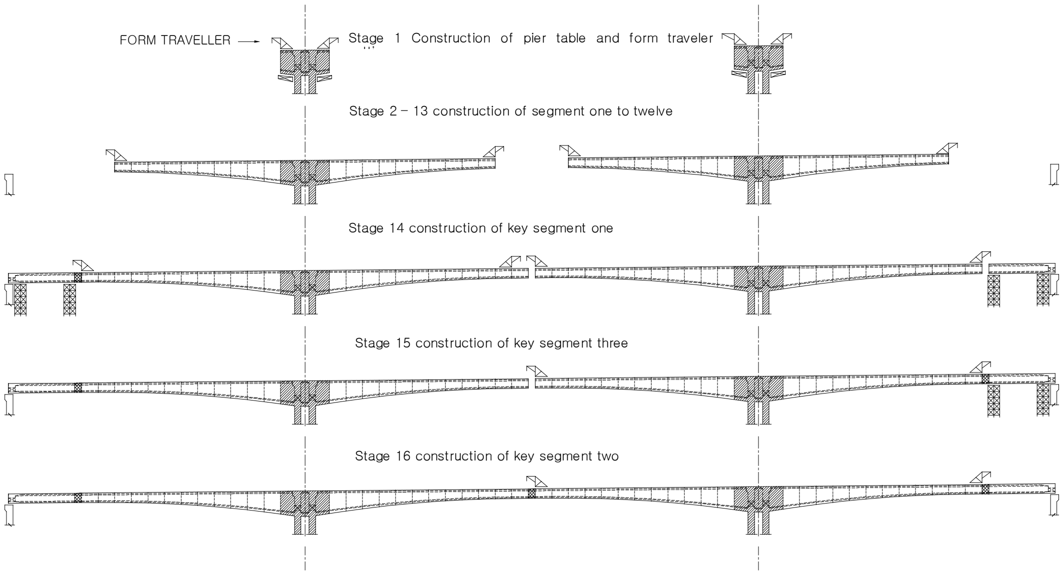


Figure 21 *Construction Sequence*

Activation/deactivation of Structure Group and Boundary Group in the construction stage analysis of the FCM Bridge is rather simple. However, in the case of a Load Group, prestress and form traveler loads are applied with the activation of the Structure Group of a particular segment, whereas the wet concrete load is applied when the concrete is poured for the next segment (see Fig. 22).

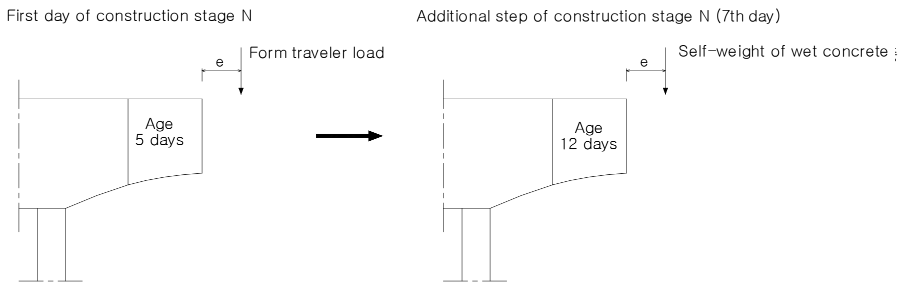


Figure 22 *Loads at Construction Stage N*

When loads are applied to the same structure with different time stages, as shown in Fig. 22, activate loads by using the Additional Step function. Define unique Additional Steps for each construction stage. Assume the required time step for the form traveler movement, formwork/rebar installation and duct placement as 7 days, and for concrete curing 5 days. Each segment activated at the beginning point of each construction stage is loaded with prestress and form traveler at the age of 5 days.

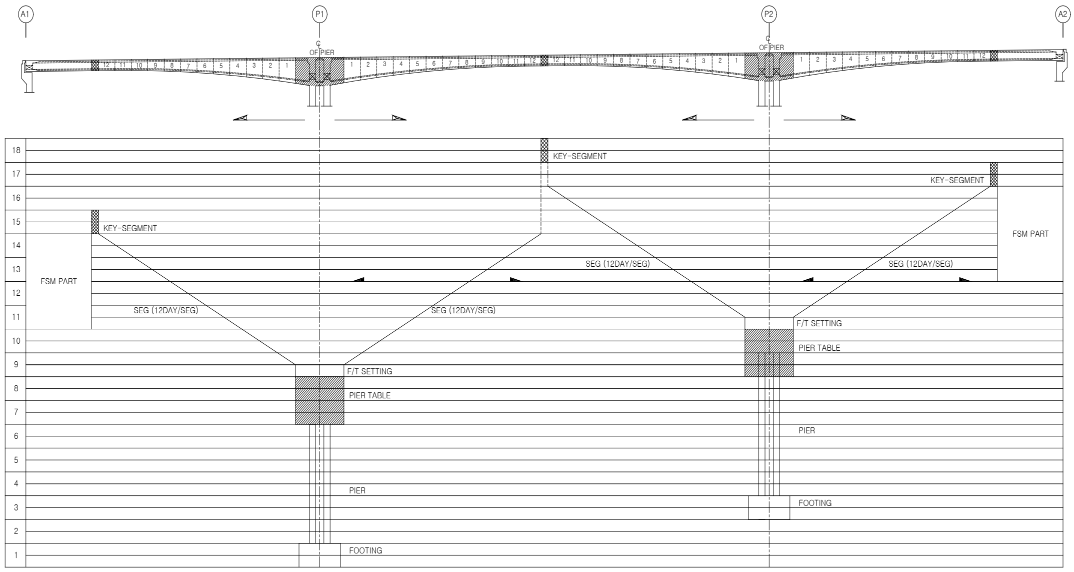


Figure 23 *Construction Schedule*

According to the construction schedule, shown in Fig. 21, segments in Pier 1 and Pier 2 are constructed simultaneously. Fig. 23 illustrates the assumed construction schedule in which each horizontal line represents a 15-day duration. Therefore, it can be seen that segments in Pier 2 are constructed 60 days after segments in Pier 1. Due to the age differences between both cantilever segments, the effects due to creep, shrinkage and prestress losses will be different. Hence, the deflections at the tip of both the cantilevers will be different due to the 60-day age difference. To minimize the residual stresses during key segment construction, the deflections at both cantilever tips should be predicted precisely. Hence, in the construction stage analysis, the age difference between the cantilevers should be taken into account.

The effects due to age difference are considered by using ***Time Loads for Construction Stage*** function. Using this function, the time duration for age difference can be applied to specified elements only. The analysis steps, using the ***Time Loads for Construction Stage*** function, are as follows:

1. Arrange construction stages assuming the pier table and segments 1 to 12 are constructed simultaneously from both the piers.
2. Load self-weight of wet concrete of key segments (load WC-KeySeg1 at the end of left cantilever at pier 1 and load WC-KeySeg3 at the end of right cantilever at pier 2)
3. Define a stage that has 0 time duration, activate KeySeg1 and FSM1. Then activate the Time Load (60 days) for pier 1 and FSM1 on the "Last Day" in the construction stage.
4. Activate KeySeg3 and FSM3, and load self-weight of wet concrete of KeySeg2.
5. Define next stage and activate KeySeg2.

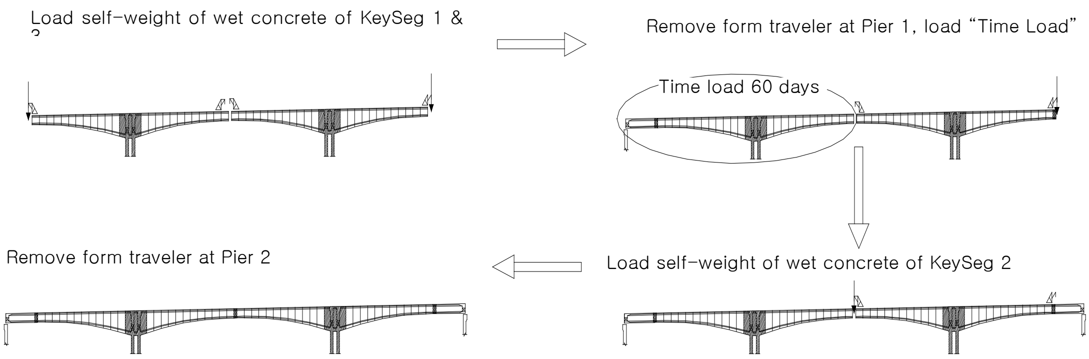


Figure 24 *Consideration of age difference using Time Load for Construction Stage Analysis*

The summary for the construction stages in terms of activation/deactivation of the Structure, Load and Boundary Group at each construction stage is as follows.

1. Construction stage 1
   - Activate Structure Group for the pier and pier table
   - Activate Boundary Group (BC_Pier) for the pier and pier table
   - 1^{st} day: Activate prestress, form traveler load and self-weight
   - 7^{th} day: Activate self-weight of wet concrete (segment 1)
2. Construction stage 2
   - Activate segment 1
   - 1^{st} day: Deactivate form traveler load and self-weight of wet concrete; activate form traveler load and prestress
   - 7^{th} day: Activate self-weight of the wet concrete (segment 2)
3. Construction stage 3-12: same as step (2)
4. Construction stage 13
   - Activate segment 12
   - 1^{st} day: Deactivate form traveler load and self-weight of wet concrete; activate form traveler load and prestress
   - 20^{th} day: Activate self-weight of the wet concrete (key segments 1 and 3)
5. Construction stage 14
   - Activate KeySeg 1 and FSM1
   - 1^{st} day: Deactivate form traveler load at pier 1 and self-weight of wet concrete at KeySeg 1; activate prestress
   - Last day: activate time load for FSM1
6. Construction stage 15
   - Activate KeySeg 3, FSM3
   - 1^{st} day: Deactivate self-weight of the wet concrete of KeySeg 3; activate prestress and self-weight of wet concrete of KeySeg 2
7. Construction stage 16
   - Activate KeySeg 2
   - 1st day: deactivate form traveler load and self-weight of wet concrete; activate prestress
8. Construction stage 17
   - 1^{st} day: activate superimposed dead load

Define construction stages first. Assign duration for CS1 to CS12 as 12 days. Assign duration for CS13 and CS15 as 30 days because the construction duration of the key segment is 30 days according to the construction schedule. Define additional step as 30-10 = 20 days, assuming the initial age of the key segment is 10 days. Assign CS16 with 0 time duration. Apply superimposed dead load at CS17. Assign 10000 days as duration for CS17 to consider the effects of long term loads, creep and shrinkage.

Load Tab / Construction Stage option / ***Define C.S.***
[Generate]

Name **(CS)** ; Suffix **(1 to 12)** ; Duration **(12)**
Additional Steps>Day **(7)** [Add]
Save Result>**Stage** (on) ; **Additional Steps** (on) ↵

Name **(CS13)** ; Suffix **( )** ; Duration **(30)**
Additional Steps> [Clear] ; Day **(20)** [Add] ↵

Name **(CS14)** ; Suffix **( )** ; Duration **(0)**
Additional Steps> [Clear] ↵

Name **(CS15)** ; Suffix **( )** ; Duration **(30)**
Additional Steps>Day **(20)** [Add] ↵

Name **(CS16)** ; Suffix **( )** ; Duration **(0)**
Additional Steps> [Clear] ↵

Name **(CS17)** ; Suffix **( )** ; Duration **(10000)**
Additional Steps> [Clear] ↵

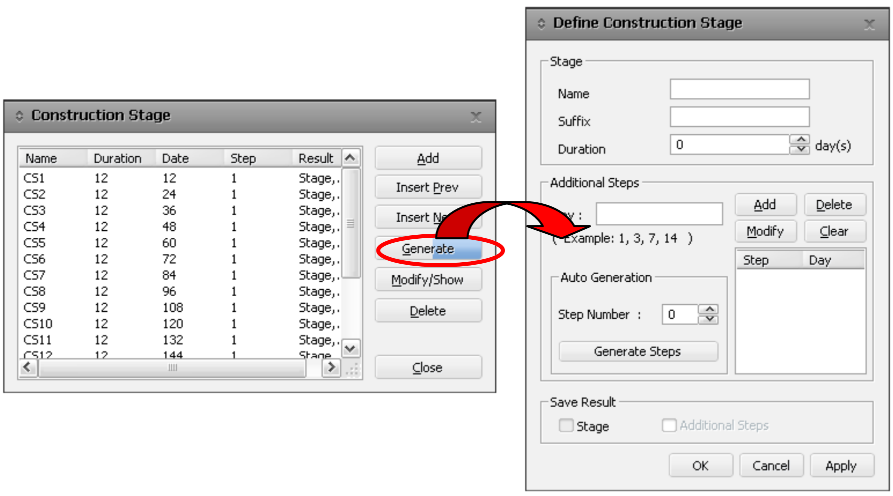


Figure 25 *Defining Construction Stage*

Construction Stage Arrangement

Define the construction stage assuming 100 days for the initial age of the pier and 15 days for the pier table.

Define the construction stage CS1 with reference to the construction stages summarized earlier.

Load / Construction Stage Option / ***Define Construction Stage***

Name>**CS1** Modify/Show

Name **(CS1)** ; Duration **(12)**

Element tab

Group List>**Pier1, Pier2** ; Activation>Age **(100)** Add

Group List>**PierTable1, PierTable2**

Activation>Age **(15)** Add

Boundary tab

Group List>**BC_Pier**

Activation>Spring/Support Position>**Original** (on) Add

Load tab

Group List>**Self, PS-PierTable1, PS-PierTable2**

FT-PierTable1, FT-PierTable2

Activation>Active Day>**First** Add

Group List>**WC-P1Seg1, WC-P2Seg1**

Activation> Active Day>**7** Add ↵

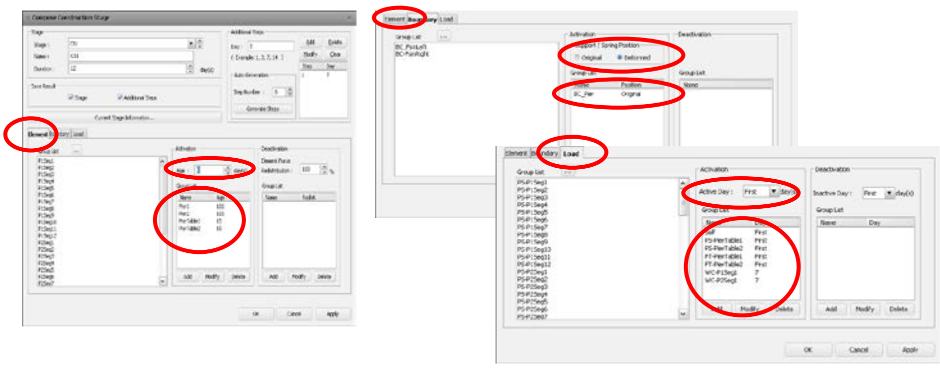


Figure 26 *Define Construction Stage 1*

Define other construction stages using the same procedure outlined in stage CS1. Repeated input to define other construction stages can be easily performed by using the MCT **Command Shell** function. The procedure to define construction stages using the MCT **Command Shell** function is as follows:

Tools>***MCT Command Shell***

Command or Data **(*STAGE)** Insert Data

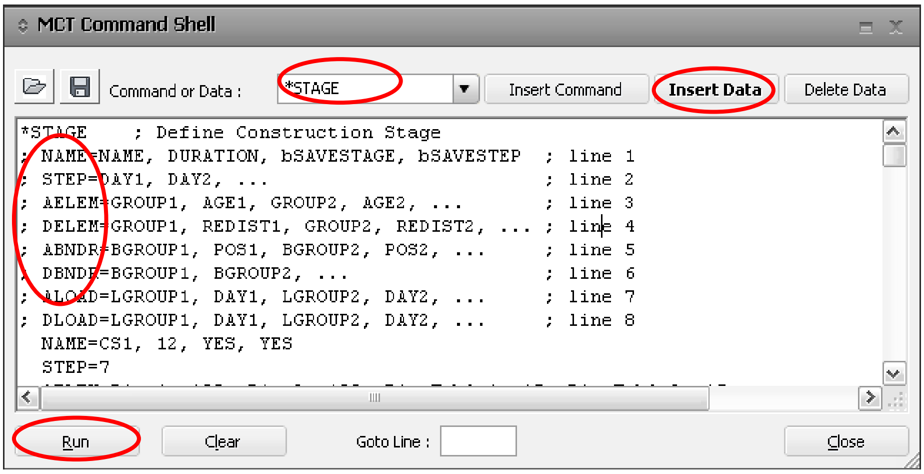


Figure 27 *MCT Command Shell*

As shown in Fig. 27, the construction stage information is divided into eight commands, and each command is as follows:

NAME : construction stage name, flag for saving output

STEP : time step

AELEM : activate structure group and its initial age

DELEM : deactivated structure group and redistribution factor for 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

According to the above procedure, the information for construction stage 2 can be input as follows:

```
*STAGE
NAME=CS2, 12, YES, NO
STEP=7
AELEM=P1Seg1, 5, P2Seg1, 5
ALOAD=FT-P1Seg1, FIRST, FT-P2Seg1, FIRST, PS-P1Seg1, FIRST
          PS-P2Seg1, FIRST, WC-P1Seg2, 7, WC-P2Seg2, 7
DLOAD=WC-P1Seg1, FIRST, WC-P2Seg1, FIRST
          FT-PierTable1, FIRST, FT-PierTable2, FIRST
```

Click Run (

) after input.

The construction stages can thus be easily defined using the above procedure.

Load Input

Input loads for each construction stage. Construction stage loads consist of form traveler, wet concrete, self-weight of segments, prestress, time load and superimposed load. Input construction stage load as following sequences.

1. Self-weight of structure
2. Form traveler
3. Wet concrete
4. Prestress
5. Time load
6. Superimposed load

Input the self-weight first. To automatically load the self-weight of the generated structure, define self-weight of the structure and load at CS1.

Load / ***Self Weight***

Load Case Name>**Self**

Load Group Name>**Self**

Self Weight Factor>Z **(-1)** 

Input the form traveler load. The form traveler load is assumed to be a 800 kN vertical load with a 2000 kN-m bending moment about the y-axis, applied at the tip of the cantilever.

Once the stage mode is selected, the Structure Groups, Load Groups and Boundary Groups assigned to the current stage are automatically activated, and the loads can be easily entered. The loads are inputted at each construction stage using the Stage Toolbar.

Stage>**CS1**

Iso View

Load Tab / Static Loads Option/ ***Nodal Loads***

Select Single (Node : **21**)
Load Case Name>**FT** ; Load Group Name>**FT-PierTable1**
Options>**Add** ; FZ **(-800)**, MY **(-2000)**

Select Single (Node : **29**)
Load Case Name>**FT** ; Load Group Name>**FT-PierTable1**
Options>**Add** ; FZ **(-800)**, MY **(2000)**

Select Single (Node : **71**)
Load Case Name>**FT** ; Load Group Name>**FT-PierTable2**
Options>**Add** ; FZ **(-800)**, MY **(-2000)**

Select Single (Node : **63**)
Load Case Name>**FT** ; Load Group Name>**FT-PierTable2**
Options>**Add** ; FZ **(-800)**, MY **(2000)**

The loads could be more easily input using the MCT command Shell. The MCT command for Nodal Loads is "CONLOAD". A more detailed explanation can be found in the "MCT Command Quick Reference" in the on-line manual appendix.

The form traveler load is defined according to the construction stages using the same procedure given in stage CS1.

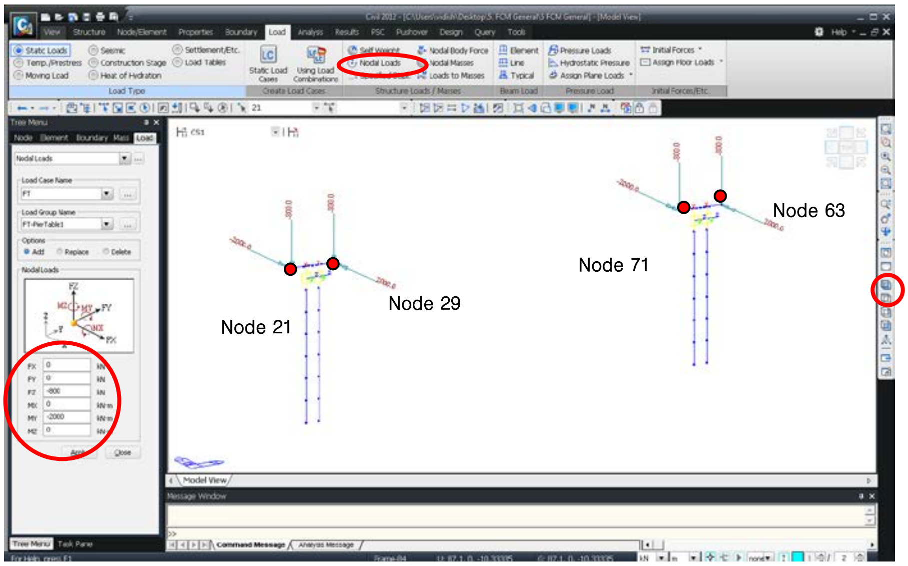


Figure 28 *Form Traveler Load Input*

By using the Bill of Material function, the length, surface area and weight of each member can be easily calculated. A detailed explanation can be found in Tools > Bill of Material in the on-line manual.

The sections in Tapered Section Group should be transformed to Tapered Type section because the weight of each Tapered Section Group is calculated instead of each element.

The mode should be changed to Base Mode because section information can be modified only when in Base Stage.

Input the new starting number for generated sections.

Input the self-weight of wet concrete after the form traveler load. The self-weight of wet concrete is calculated from the Bill of Material function. Before calculating the weights of each element using the Bill of Material function, transform each section composed of Tapered Section Group to Tapered Type section. By transforming the section, sections 101-112 are generated as shown in Fig. 29.

Stage>**Base**

Properties Tab/ ***Tapered Group***

Name>**1stspan** Convert to Tapered Section...

New Start Section Number **(101)** ↵

Properties Tab / ***Section Properties***

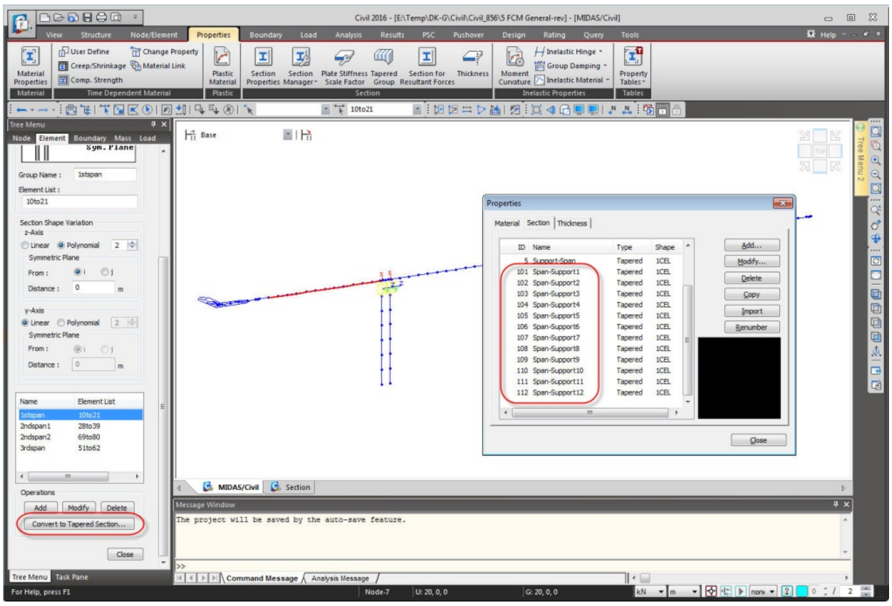


Figure 29 *Transform to Tapered Type Section*

Calculate the self-weight of each segment using the Bill of Material function. In Fig. 30, sections 101 to 111 represent segments 1 to 11, respectively, and section 112 represents the variable section of the pier table. The length, surface area and weight can be confirmed for each section.

Tools / ***Bill of Material***

Select BOM outputs>**Beam-Truss Element BOM type1** (on) ↵

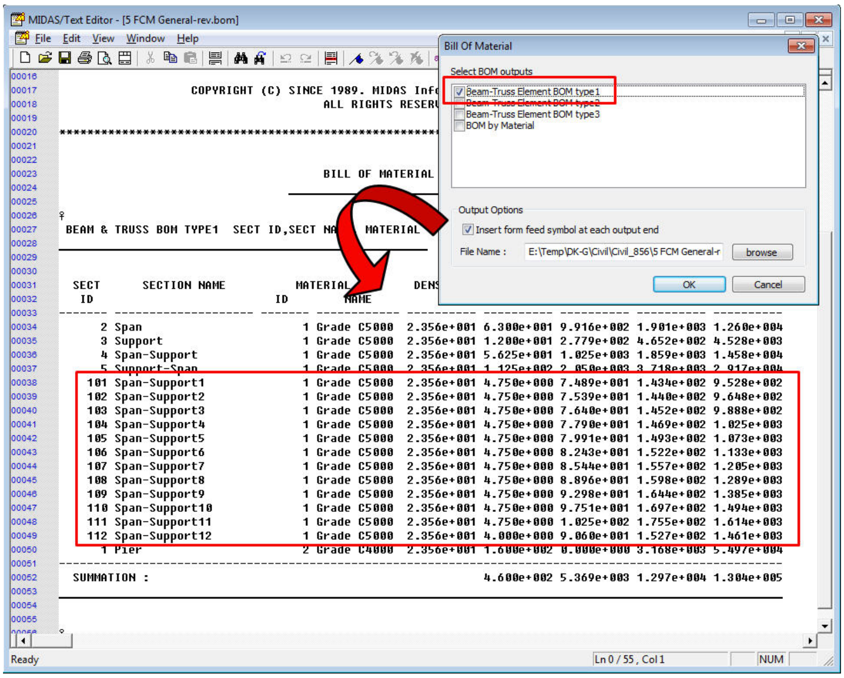


Figure 30 *Bill of Material*

Input the self-weight of the wet concrete. The self-weight of the wet concrete is represented by a vertical load and a y-axis bending moment. The vertical loads are the self-weight of each segment constructed at the cantilever tip in each construction stage. The bending moment is calculated by assuming the eccentricity for the wet concrete as 2.5m.

Input the self-weight of the wet concrete using the MCT Command Shell. The MCT command for nodal load is "*CONLOAD".

Stage>**CS1**

Load Tab/ Static Loads Option /***Nodal Loads***

Select Single (Node : **21**)

Load Case Name>**WC** ; Load Group Name>**WC-P1Seg1**

Options>**Add** ; FZ **(-1614)**, MY **(-1614*2.5)**

Select Single (Node : **29**)

Load Case Name>**WC** ; Load Group Name>**WC-P1Seg1**

Options>**Add** ; FZ **(-1614)**, MY **(1614*2.5)**

Select Single (Node : **71**)

Load Case Name>**WC** ; Load Group Name>**WC-P2Seg1**

Options>**Add** ; FZ **(-1614)**, MY **(-1614*2.5)**

Select Single (Node : **63**)

Load Case Name>**WC** ; Load Group Name>**WC-P2Seg1**

Options>**Add** ; FZ **(-1614)**, MY **(1614*2.5)**

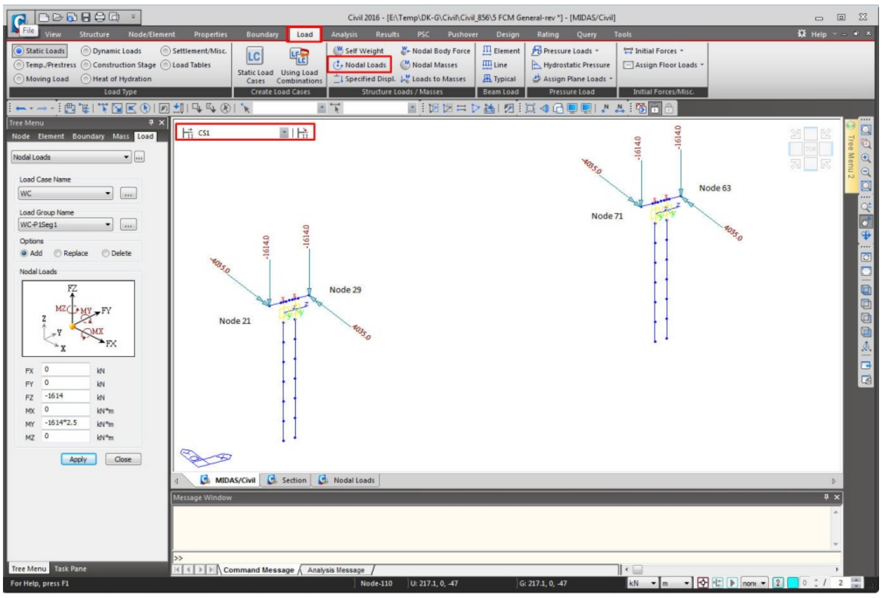


Figure 31 *Input self-weight of wet concrete*

Input prestress. From the defined starting, inflection and ending point, the optimum tendon profile can be generated automatically within the program. Three dimensional tendon coordinates about the x-axis define the tendon profile. Before defining the tendon coordinates, the tendon properties should be input.

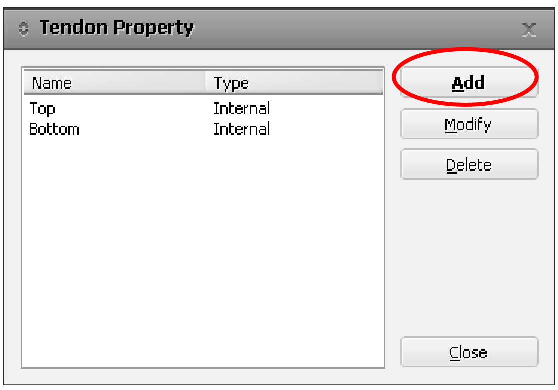
Stage>**Base**

Load Tab/ (Temp./Prestress Loads)/ ***Tendon Property***

Tendon Name **(TOP)** ; Tendon Type>**Internal**
Material>**3: tendon**
Total Tendon Area **(0.0026353)**
or ...
Tendon Area>**15.2mm(0.6 ")**
Number of Tendon Area **(19)** ↵
Duct Diameter **(0.103)** ; Relaxation Coefficient (**45**)
Curvature Friction Factor **(0.2)** ; Wobble Friction Factor **(0.001)**
Ultimate Strength **(1900000)** ; Yield Strength **(1600000)**
Load Type>**Post-Tension**
Anchorage Slip>Begin **(0.006)** ; End **(0.006)** ↵

Tendon Name **(BOTTOM)** ; Tendon Type>**Internal**
Material>**3: tendon**
Total Tendon Area **(0.0026353)**
or ...
Tendon Area>**15.2mm(0.6 ")**
Number of Tendon Area **(19)** ↵
Duct Diameter **(0.103)** ; Relaxation Coefficient (**45**)
Curvature Friction Factor **(0.3)** ; Wobble Friction Factor **(0.0066)**
Ultimate Strength **(1900000)** ; Yield Strength **(1600000)**
Load Type>**Post-Tension**
Anchorage Slip>Begin **(0.006)** ; End **(0.006)** ↵

The relaxation Coefficient is a constant used in Magura's formula. It is generally used to calculate relaxation effects of the tendon material over time. It can be assumed to be 10 for normal relaxation strand and 45 for low relaxation strand. A detailed explanation of the Relaxation Coefficient can be found under "Prestress Loss" in the Analysis of Civil Structures.



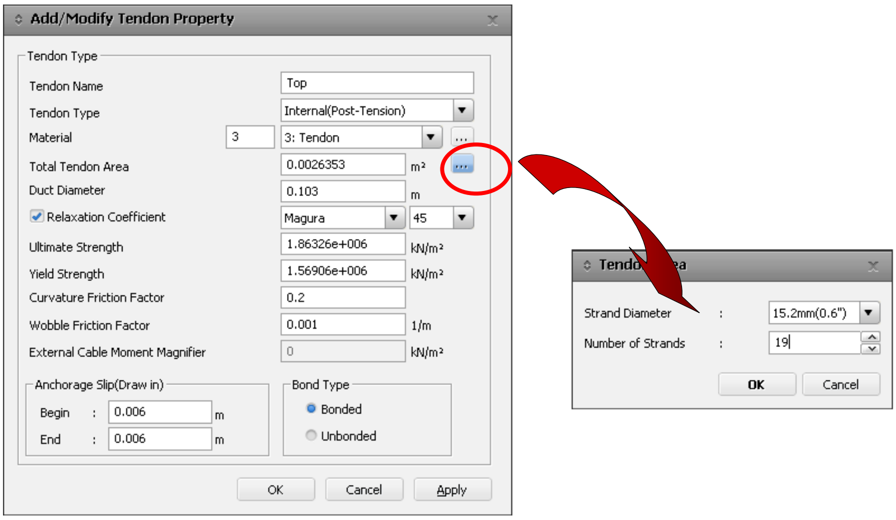


Figure 32 *Input Tendon Properties*

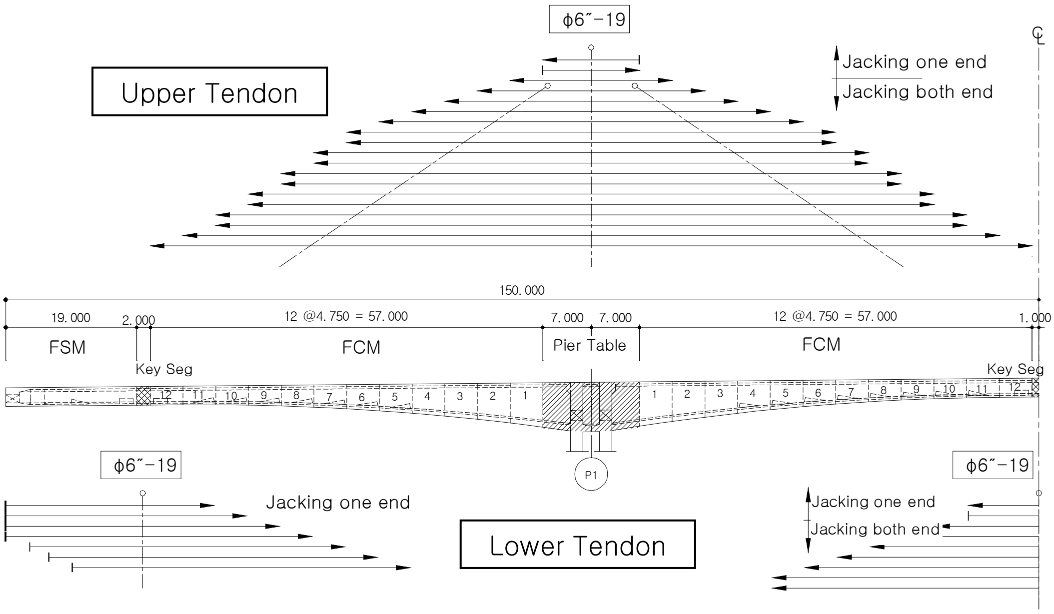


Figure 33 *Tendon Arrangement*

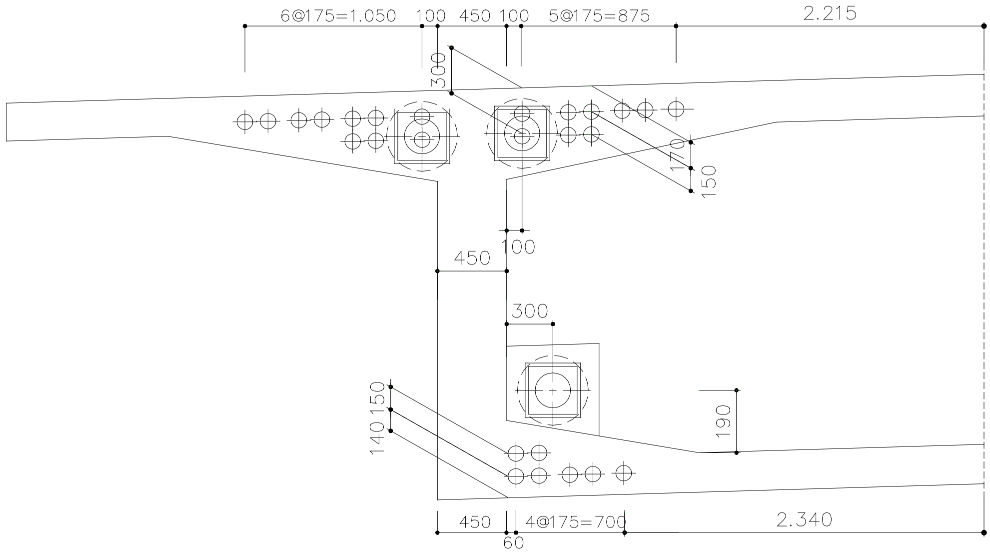


Figure 34 *Tendon Arrangements for the Side Span*

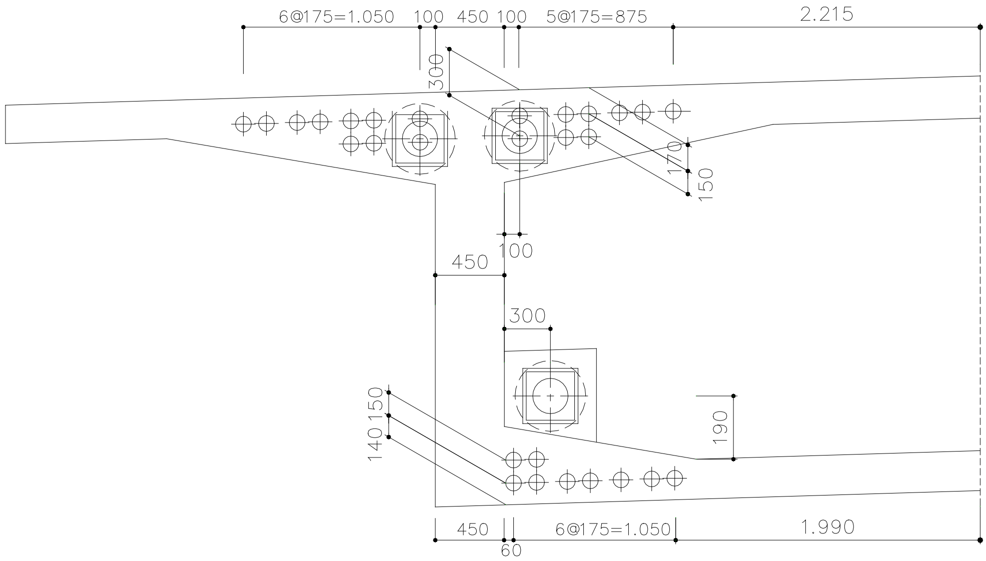


Figure 35 *Tendon Arrangements for the Center Span*

The base point for the tendon profile is the upper center point of the prestressed concrete box section because the box section is defined with reference to the center-top.

The slope is a fixed value if FIX is checked on. Otherwise a curve with a calculated slope is generated.

Define 1st tendon for pier table 1 using Figs. 33 to 35.

Tree Menu > Group>Structure Group>PierTable1>**Active**
Load Tab / (Temp./ Prestress) Option / ***Tendon Profile*** Add
Tendon Name **(P1TC1R)** ; Tendon Property>**TOP**
Select All or Assigned Elements **(21to28)**
Input Type > **3D** ; Curve Type > **Spline**
Straight Length of Tendon>Begin **(0)** ; End **(0)**
Profile
1>x (**0**), y (**0**), z (**-0.3**), **fix** (off)
2>x (**2**), y (**0**), z (**-0.15**), **fix** (on), Ry (**0**), Rz (**0**)
3>x (**12**), y (**0**), z (**-0.15**), **fix** (on) , Ry (**0**), Rz (**0**)
4>x (**14**), y (**0**), z (**-0.3**), **fix** (off)
Tendon Shape>**Straight**
Profile Insertion Point **(78, -3.09, 0)**
X Axis Direction>**X** ↵

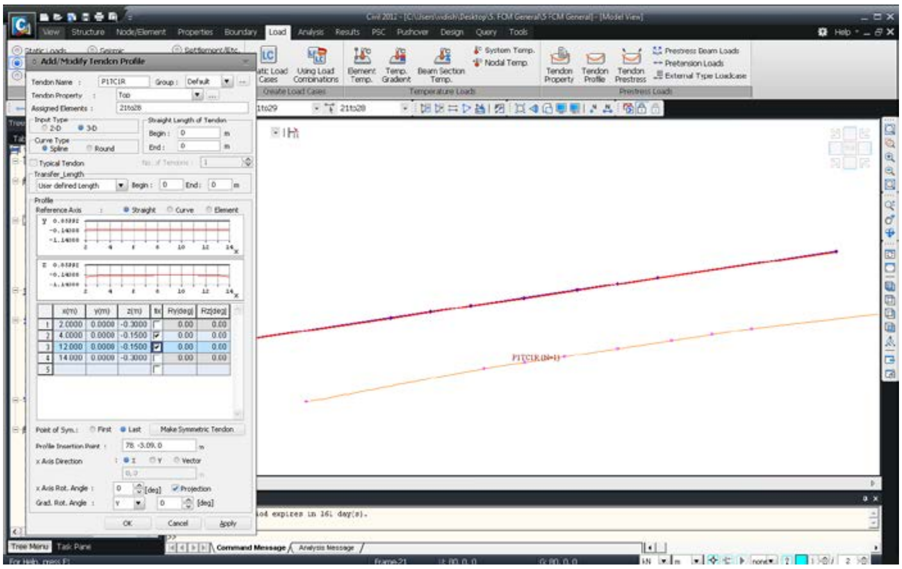


Figure 36 *Define the Tendon Profile*

Copy pre-defined tendon P1TC1R to define additional tendons profiles with the same y coordinates.

Name>P1TC1R Copy/Move

Name>P1TC1R-Copy Modify

Tendon Name **(P1TC2R)**

Profile Insertion Point **(78, -3.74, 0)** ↵

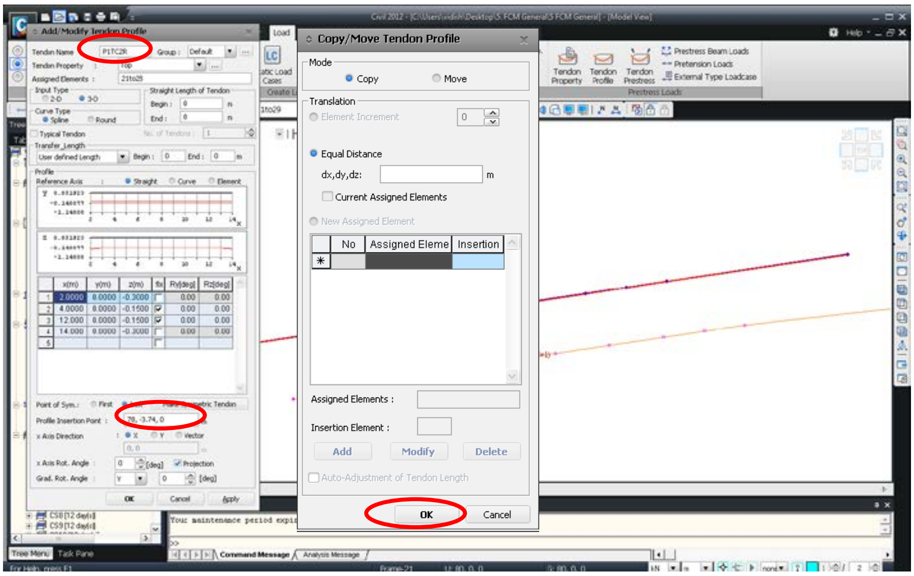


Figure 37 *Copy the Tendon Profile*

Define each of the tendon profiles using the same procedure.

A tendon profile may be defined more easily using the MCT Command Shell. The MCT command for tendon profile definition is "*TDN-PROFILE".

After defining all tendon profiles, apply the prestress to each construction stage using the defined tendon profile.

Stage>**CS1**
Load Tab/ (Temp./ Prestress) Option / ***Tendon Prestress***
Load Case Name>**PS** ; Load Group Name>**PS-PierTable1**
Tendon>**P1TC1L**, **P1TC1R** > Selected Tendons
Stress Value>**Stress** ; 1st Jacking>**Begin**
Begin **(1330000)** ; End **(0)**
Grouting : after **(1)** Add

Load Case Name>**PS** ; Load Group Name>**PS-PierTable1**
Selected Tendons>**P1TC1L**, **P1TC1R** > Tendon
Tendon>**P1TC2L**, **P1TC2R** > Selected Tendons
Stress Value>**Stress** ; 1st Jacking>**Begin**
Begin **(1330000)** ; End **(0)**
Grouting : after **(1)** Add

Load Case Name>**PS** ; Load Group Name>**PS-PierTable2**
Selected Tendons>**P1TC2L**, **P1TC2R** > Tendon
Tendon>**P2TC1L**, **P2TC1R** > Selected Tendons
Stress Value>**Stress** ; 1st Jacking>**Begin**
Begin **(1330000)** ; End **(0)**
Grouting : after **(1)** Add

Load Case Name>**PS** ; Load Group Name>**PS-PierTable2**
Selected Tendons>**P2TC1L**, **P2TC1R** > Tendon
Tendon>**P2TC2L**, **P2TC2R** > Selected Tendons
Stress Value>**Stress** ; 1st Jacking>**Begin**
Begin **(1330000)** ; End **(0)**
Grouting : after **(1)** Add

Select "Both" in "1st Jacking" when both the ends are stressed.

Input the construction stage in which the tendon is grouted. The stress is calculated for net section before the grouting stage and for composite section after grouting. The tendon is grouted after jacking when '1' is selected in "Grouting".

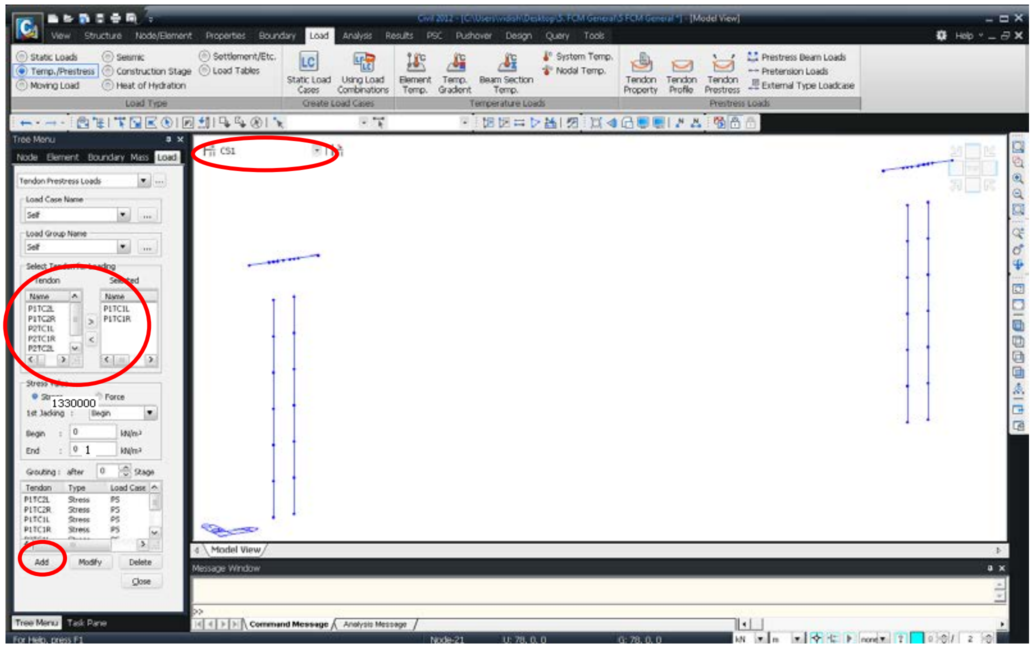


Figure 38 *Prestress Load*

Apply prestress at each construction stage using the same procedure.

The prestress may be defined more easily using the MCT Command Shell. The MCT command for prestress is "*TDN-PRESTRESS".

Input the construction time duration periods. Input the duration of construction period between pier 1 and pier 2 as 60 days. Since the time period of 60 days is applied at CS14, change stage to CS14 and then input the time period.

Stage>**CS14**

Load Tab / Construction Stage Options / C.S.Loads /***Time Loads for Construction Stage***

Select Window (Fig.39, ①)

Load Group Name>**TimeLoad**

Options>**Add**

Time Loads **(60)** ↵

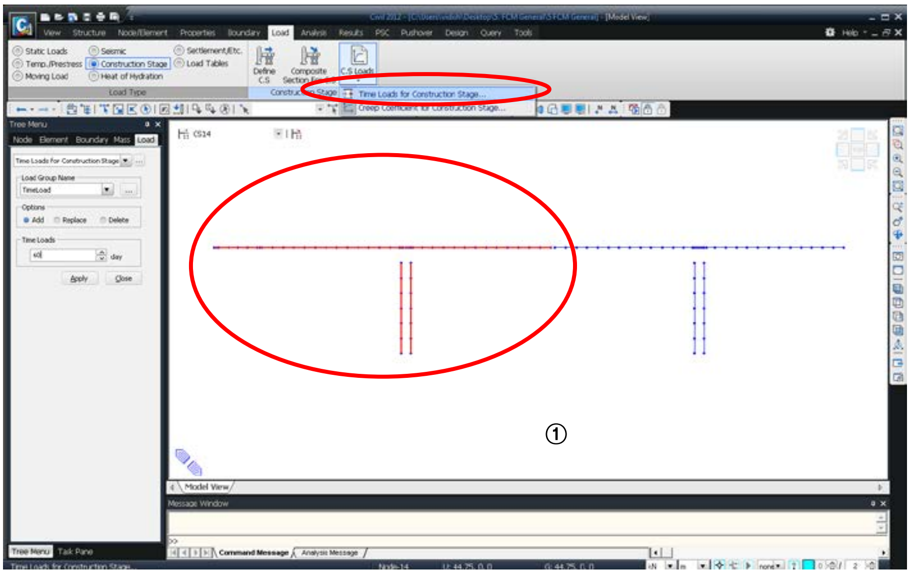


Figure 39 *Input Time Load*

Perform Structural Analysis

We will now perform structural analysis.

Analysis / ***Perform Analysis***

Refer to tutorial on “Construction Stage Analysis using FCM Wizard” for analysis output results.