Advanced Application 5

Construction Stage Analysis of a FCM Bridge using General Functions



CONTENTS

Construction sequence and construction stage analysis for FCM 1

Assign Working Environment 3

Define section and material properties 4

Structural modeling 10

Prestressed Concrete Box Girder Modeling 11

Pier Modeling 16

Assign Structure Group 17

Generate Structure Group 17

Define Boundary Groups and Input Boundary Conditions 21

Assign Load Group 24

Define and Arrange Construction Stages 26

Define Construction Stages 26

Construction Stage Arrangement 31

Load Input 34

Perform Structural Analysis 46

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 NX 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.



Tools Tab / Unit System

Length> **m** ; Force>**kN** ,

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

The unit system

Unit System X					
Length	Force (Mass)	Heat			
O m	○ N (kg)) cal			
⊖ cm	O kN (ton)) kcal			
\odot mm	okgf (kg)	O J			
⊖ ft	tonf (ton)	⊖ kJ			
⊖ in	Olbf (lb)) Btu			
	⊖ kips (kips/g)				
Temperature					
 Celsius 	Celsius OFahrenheit				
Note : Selec dialog boxe units.	cted units are displayed s. Values are NOT char	d in relevant nged with			
Set/Chang	e Default Unit System				
ОК	Cancel	Apply			

Unit system settings

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) ↓





Material Definition

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



Section definition

The section offset

is defined at the

Center-Top because

the sections are of variable shapes.

Properties Tab / 8 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) -

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



Definition of cross section in the center of the span

ନ

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) ↓



DB/User PSC Section ID 3 TPSC-1CELL, 2CELL Mesh Size for Stiff. Cal Name Suppor Joint On/Off 0 JO1 0 JI1 JI4 H01 0.25 m 801 2.8 m m 801-2 HO2 0.35 HO2-1 0 JO2 JI2 🛃 JI5 801-1 1.05 m m m m JO3 JI3 801-2 0 m H02-2 0 802 0 m ection Type BO2-1 0 H03 6.4 m O1Cell 803 3.55 m H03-1 0 m 2 Cell Shear Check 0.25 m 0.325 m 801 3.1 801-1 1.35 HT. m m Z1 6:425 m 🕑 HI2 22 Centroid H2-1 0 H2-2 0 m 811-2 m 0 ٠ Z3 1.1 m BI2-1 0 m Web Thick. HI3 5.325 m 813 3.1 HI3-1 0 m BI3-1 1.85 HI4 0.25 m BI3-2 0 for Shear(total) Auto 813-1 1.85 m t1 1.15 m 🖸 t2 0.9000 m 😨 m HI4-1 0 m 814 0 13 0.8999 m 😋 HI4-2 0 m for Torsion(min.) HI5 0.85 m 0.4499999r m Consider Shear Deform Consider Warping Effect(7th DOF) Warping Check O Auto O User Change Offset Show Calculation Results... ок Cancel

Definition of cross section at the supports

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. After completion of section property input, generate section properties for the Tapered Type using Section ID 2 and 3. $^{
m \Theta}$

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 $^{m heta}$

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

)

Section ID 4 PSC-ICELL Name Span-Support Value User DB ASCIDICE III 3 1000 m Bi1 3 1000 m Bi3 1 1.8500 m SiZe-J mport. Cac. Section Properties Z1-Auto 2 Z3-Auto 2 Z3-Auto 2 S1-Auto 2 Z3-Auto 2 S1-Auto 2 S1-A	DB/UserTapered	_				
Name Span-Support Value User OB All COLLSING Dimension HIS 0.2600 m HIS 0.1111 0.1111 B13 1.11 1.3500 m HIS 0.1111 0.1111 B13-1 1.8500 m HIS 0.11111 0.11111 0.11	Section ID 4	TPSC-1CELL			~	
Dimension HIS 0.2600 m BI1 3.1000 m BI3.1 1.8500 m Size-J Impot. Calc. Section Properties 21-Auto 21-Auto 0 S3-Auto 0 HO1 0.2500 m HO2.1 0.000 m HO2.1 0.000 m Y Axis Variation Include Options(Joint, Type, Shape) HO2.1 0.000 m Y Axis Variation Include Options(Shear Check, Minimum Web Thickness) Select PSC Section 2Span Offset : Center-Top Auto Change Offset OK Show Calculation Results OK	Name Span-Support	Value O Use	r O'DB		~	
Image: consider Shear Deformation Y Axis Variation Yaxis Yariation		Dimension				
Image offset : Organization Image offset : Offset : Center-Top Organization Calculation Results OK Calculation Results OK Calculation Results OK		[HI5	0 2600	m	_	
Bit-1 1.3500 m Bi3 3.1000 m Bi3-1 1.8500 m SizeJ Import. Z1-Auto Calc. Section Properties Z1-Auto S2-Auto S3-Auto S2-Auto S3-Auto S2-Auto S3-Auto Import PSC Section HO2 0.3500 m Import PSC Section y Axis Variation Li Consider Shear Deformation Select PSC Section Consider Warping Effect/7 Warping Check Mesh Size for Stiff. Calc. Support Offset : Center-Top OK Change Offset OK		BI1	3.1000	m		
BI3 3.1000 m BI3-1 1.6500 m Size-J Import Calc. Section Properties 23.Auto 23.Auto 0 S1-Auto 0 S2-Auto 0 S3.Auto 0 T-Auto 0 HO1 0.2500 m HO2 0.3500 m Urport PSC Section Select PSC Section Consider Shear Deformation Select PSC Section Consider Warping Check Au Mesh Size for Stiff. Caic. Support Offset : Center-Top Change Offset OK Show Calculation Results	5000	BI1-1	1.3500	m		
Bi3-1 1.8500 m Size-J Import Caic. Section Properties 21.Auto 23.Auto 0 S3.Auto 0 S3.Auto 0 T-Auto 0 S3.Auto 0 T-Auto 0 YAxis Variation Ut HO2 0.3500 m Include Options(Joint, Type, Shape) Include Options(Shear Check, Minimum Web Thickness) Select PSC Section Support Offset : Center-Top Auto Change Offset OK Show Calculation Results OK		BI3	3,1000	m		
Size-J Import Calc. Section Properties Z1-Auto Z1-Auto Calc. Section Properties Z1-Auto Calc. Section Properties Z1-Auto Calc. Section Properties Z1-Auto Calc. Section Properties Z1-Auto Calc. Section S2-Auto Calc. Section S2-Auto Calc. Section T-Auto Calc. Section Y Axis Variation Lt HO2:1 0.0000 HO2:1 0.0000 Y Axis Variation Lt Include Options(Joint, Type, Shape) Include Options(Shear Check, Minimum Web Thickness) Select PSC Section Select PSC Section Consider Shear Deformation Calc. Consider Warping Effect(77 Yarping Check Warping Check Autor Mesh Size for Stiff. Calc. Autor Mesh Size for Stiff. Calc. Autor Show Calculation Results OK Calc.		BI3-1	1.8500	m		
Calc. Section Properties Z1-Auto Z3-Auto S1-Auto S2-Auto S2-Auto S3-Auto S2-Auto S3-Auto S2-Auto S3-Auto S2-Auto S3-Auto S2-Auto S3-Auto S3-Auto T-Auto Q000000000000000000000000000000000000		Size-J	Import			
Z1-Auto Z Z3-Auto Z S1-Auto Z S2-Auto Z S3-Auto Z S3-Auto Z HO1 0.2500 m HO2 0.3500 m JOSOPHON Include Options(Joint, Type, Shape) ZAxis Variation I Include Options(Shear Check, Minimum Web Thickness) Select PSC Section Consider Warping Effect(7) Warping Check Au Mesh Size for Stiff. Calc. Offset : Center-Top Change Offset OK Show Calculation Results OK		Calc. Se	ction Properti	es		
Z3-Auto Import PSC Section YAis Variation Import PSC Section YAis Variation Import PSC Section Y Axis Variation Import PSC Section Y Axis Variation Import PSC Section Y Consider Shear Deformation Select PSC Section Consider Shear Deformation Select PSC Section Offset : Center-Top OK Change Offset OK		Z1-Auto	2			
S1-Auto Import PSC Section HO1 0.2500 m HO2 0.3500 m Jondation Li Include Options(Joint, Type, Shape) Select PSC Section Consider Shear Deformation Select PSC Section Consider Warping Effect/7t Warping Check Mesh Size for Stiff. Calc. Support Offset : Center-Top Change Offset OK Show Calculation Results OK		Z3-Auto				
S2-Auto S3-Auto S3-Auto S3-Auto T-Auto Import PSC Section HO2 0.3500 m HO2 0.3500 m HO2 0.3500 m HO2 0.3500 m HO2 0.000 m Y Axis Variation Li Include Options(Joint, Type, Shape) Z Axis Variation Z Axis Variation Li Consider Shear Deformatio Select PSC Section Consider Shear Deformatio Select PSC Section Consider Warping Effect/7t Zspan Warping Check Au Mesh Size for Stiff, Caic. Support Offset : OK Change Offset OK		S1-Auto	9			
S3-Auto Image: Change Offset Show Calculation Results OK	7	S2-Auto				
T-Auto Image: Consider Shear Deformation Yaxis Variation Lit Include Options(Joint, Type, Shape) z Axis Variation Lit Include Options(Shear Check, Minimum Web Thickness) Select PSC Section Varping Check Aut Mesh Size for Stiff, Catc. Offset : Center-Top Change Offset OK Cata		S3-Auto				
HO1 0.2500 m HO2 0.3500 m HO2-1 0.000 m y Axis Variation Li Include Options(Joint, Type, Shape) 2 Axis Variation Z Axis Variation Li Consider Shear Deformatio Consider Shear Deformatio Consider Warping Effect(7) Select PSC Section Warping Check Au Mesh Size for Stiff. Calc. Support Offset : Center-Top OK Change Offset OK		I-Auto				
HO2 0.3000 [m] HO2 0.0000 [m] HO2 0.0000 [m] Import PSC Section y Axis Variation iii Include Options(Joint, Type, Shape) iiii Z Axis Variation iii Import PSC Section iiiiiiiiiiiiiiiiiiiiiiiiiiiiiiiiiiii		HOI	0.2500	m		
Import PSC Section y Axis Variation Lin z Axis Variation Lin Consider Shear Deformation Consider Shear Deformation Consider Warping Effect(7t Warping Check Mesh Size for Stilff. Calc. Offset : Center-Top Change Offset OK		HOZ	0.3500	m		
y Axis Variation Lir z Axis Variation Lir Consider Shear Deformatio Consider Warping Effect/7t Warping Check Au Mesh Size for Stiff. Calc. Offset : Center-Top Change Offset Show Calculation Results		102-1	0.00	mport PSC Section	on	
z Axis Variation Li Include Options(John, Type, Shape) Consider Shear Deformatio Select PSC Section Consider Warping Effect/7t Select PSC Section Warping Check Au Mesh Size for Stiff. Calc. Stopport Offset : Center-Top OK Change Offset OK		y Axis Variation	Lie	Industry California	distant Trans. Char	
Offset : Center-Top OK Ca Change Offset OK Ca		z Avis Variation	11	include option	staouur, type, sout	<i>ic</i>]
Consider Shear Deformation Consider Warping Effect(7t Warping Check Au Mesh Size for Stilff. Calc. Offset : Center-Top Change Offset Show Calculation Results OK Ca			L'' 🕻	Include Option	s(Shear Check, Mi	nimum Web Thickness)
Consider Warping Effect(7t Warping Check OAU Mesh Size for Stiff. Caic. Offset : Center-Top Change Offset Show Calculation Results OK Ca		Consider Shear	Deformatio S	elect PSC Sectio	n	
Warping Check Au Mesh Size for Stilf. Calc. Offset : Center-Top Change Offset Show Calculation Results		Consider Warpin	ng Effect(71	:Span		Import
Offset : Center-Top Change Offset Show Calculation Results		Warping Check	O'AU	Support		
Offset : Center-Top Change Offset Show Calculation Results OK Ca		Mesh Size for S	tiff Cale			
Offset : Center-Top Change Offset Show Calculation Results OK Ca		Dimestrate for a	units terdines			S
Offset : Center-Top Change Offset Show Calculation Results OK Ca						
Change Offset Show Calculation Results OK Ca	Offset: Center-Top					A
Show Calculation Results OK Ca	Change Offset					
Show Calculation Results OK Ca						é y
Show Calculation Results OK Ca						
	Show Calculation Resul	ts OK	Ca			

Section ID 5	PSC-1CELL		
Name Support-Span	O Value O Oser	C DB DUSCH	1011
	Dimension		
	Inist		
	101	0	- 1 E
5000	102	0	-
	J03	0	-
	JI1	0	
	JI2	0	
	JI3	0	
-	JI4		
	JI5	8	
	Size-I	Import	
	Calc. Sect	tion Properties	
	Z1-Auto		1
X X	Z3-Auto		-
	S1-Auto		
	S2-Auto	8	-8
	S3-Auto		1
	y Axis Variation	Linear	
	2 Axis Variation	Lineat	
	Consider Shear D	eformation	
	Gonorper oneor o	and the second second	
	Consider Warping	geneou/in DOF)	
	Warping Check	O Auto O Use	6
	Mesh Size for Sti	ff. Calc.	m

Tapered Section composition

Structural Modeling

Model FCM Bridge using general functions of MIDAS CIVIL NX.

To perform construction stage analysis, construction stages must first be defined. In MIDAS CIVIL NX, 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.



Segment Division

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





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) <sup>^</sup>∩

Reverse Element Local (on) ↓
```



Copy the beam elements symmetrically for right half

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" 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



Change the cross-sections of the variable cross-section segments and the main

girder's end

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.

- 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.

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

Properties Tab / Tapered Section Group





Specify a tapered section group

Pier Modeling

After copying the nodes of the prestessed 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.





Copy nodes



Create piers

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.

Assign Structure Group

Figure below 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. ^O 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.



Construction sequence

By appending suffix numbers to Name, multiple

Structure Groups can be generated simultaneously.

ଜ

Generate Structure Group

Tree Menu > Group Tab

Structure Group / Define Structure Group (Right Click > New...)







Element Group Generation

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





Structure Group composition

Assign corresponding beam elements to the other remaining Structure Groups. by referring to table below.

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 doubleclicking 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)



Definition of 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

15	Select Single	(Nodes : 1)	
	Boundary Group	Name> BC_FsmLeft	
	Support Type>D	y (on), Dz (on), Rx (on) and Rz (on)	┙
•5	Select Single	(Nodes : 43)	
	Boundary Group	Name> BC_FsmRight	
	Support Type>D	y (on), Dz (on), Rx (on) and Rz (on)	₊
G	Select Window	/ (Nodes : 108 ~ 111)	
	Boundary Group	Name> BC_Pier	
	Support Type>D	-All (on) and R-All (on) ↓	



Enter boundary conditions

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)<sup>♀</sup>
2 Nodes (84, 23)<sup>♥</sup>
```

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

multiple

Soom Fit

ନ

Assign



Elastic Connection of piers and girders

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.)

Define load conditions for each load.

Load Tab >Static Loads	s Option > Static Load Cases Static Load Cases
Name (Self) ;	Type > Construction Stage Load
Name (PS) ;	Type > Construction Stage Load
Name (FT) ;	Type > Construction Stage Load
Name (WC) ;	Type > Construction Stage Load
Name (Time) ;	Type > Construction Stage Load
Name (2nd) ;	Type > Construction Stage Load

ase	All Load	Case	~	Modify
rpe escriptio	Construe	ction Stage Load (CS)	×	Delete
No	Name	Туре	Descriptio	n
1	Self	Construction Stage Load (C		
2	PS	Construction Stage Load (C		
3 FT		Construction Stage Load (C		
4	WC	Construction Stage Load (C		
5 Time		Construction Stage Load (C		
6	2nd	Construction Stage Load (C		

Define load conditions

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".

Generation "Time Loads for

Define load group for each load condition.





Load Group Definition

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

Define and Arrange Construction Stages



Construction Sequence

Define the construction stage arrangement and pier table, construction of segments 1 to 12 and construction of key segments 1 to 3 (See figure below). 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.

Define Construction Stages

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 figure below).

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



Loads at Construction Stage N

When loads are applied to the same structure with different time stages, as shown in figure below, 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.



Construction Schedule

According to the construction schedule, shown in figure above (Construction Sequence), segments in Pier 1 and Pier 2 are constructed simultaneously. Figure above 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.
- 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.



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
 - 1st day: Activate prestress, form traveler load and self-weight
 - 7th day: Activate self-weight of wet concrete (segment 1)
- 2. Construction stage 2
 - Activate segment 1
 - 1st day: Deactivate form traveler load and self-weight of wet concrete; activate form traveler load and prestress
 - 7th 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
 - 1st day: Deactivate form traveler load and self-weight of wet concrete; activate form traveler load and prestress
 - 20th day: Activate self-weight of the wet concrete (key segments 1 and 3)
- 5. Construction stage 14
 - Activate KeySeg 1 and FSM1
 - 1st 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
 - 1st 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
 - 1st 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.
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} ہـ
Name(CS15) ; Suffix(); Duration(30) Additional Steps>Day(20) ^{Add}
Name(CS16) ; Suffix(); Duration(0) Additional Steps> ^{Clear} 山
Name(CS17) ; Suffix(); Duration(10000) Additional Steps> ^{Clear} ↓

Construic	tion Stage				×	Define Construct	tion Stage		
Name	Duration	Date	Step	Result	Add	Stage			-
CS1	12	12	1	Stage,	Insert Prev	Name	h		
CS2	12	24	1	Stage,		Suffly			
CS3	12	36	1	Stage,	Insert Next	Junix			
CS4	12	48	1	Stage,	Generate	Duration	0		C day(s
CS5	12	60	1	Stage,	Chancelone	-			
CS6	12	72	1	Stage,	Modify/Show	Additional Step	25		
CS7	12	84	1	Stage,	Delete	Day :		Add	Delete
CS8	12	96	1	Stage,		10.000		and the second second	
CS9	12	108	1	Stage,		(Example: 1,	3,7,14]	Modify	Gear
CS10	12	120	1	Stage,				Sten	Dav
0511	12	132	1	Stana		Auto General	tion	ouch	Cuj
				_		Chan Mumha	0 0		
						Step Number	U v		
					Close	Generate	Steps		
					L,				
								-	
						Save Result			
						Sars neoun			
						Stage	Additio	nal Steps	
						-	1		_

Definition of overall 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. $^{\scriptsize \Theta}$

Load / Construction Stage Option / Define Construction Stage
Name>CS1 Modify/Show
Name(CS1); Duration(12)
Element tab
Group List> Pier1, Pier2 ; Activation>Age (100)
Group List>PierTable1, PierTable2
Activation>Age(15)
Boundary tab
Group List>BC_Pier
Activation>Spring/Support Position> Original (on)
Load tab
Group List>Self, PS-PierTable1, PS-PierTable2
FT-PierTable1, FT-PierTable2
Activation>Active Day> First
Group List>WC-P1Seg1, WC-P2Seg1
Activation> Active Day>7
Compose Construction Stage X

Stage				Additional S	Reps				
Stage	CS1		*	Day 0		Add	Delete		
Name	CSI			{ Example	1, 3, 7, 14	Modify	Clear		
Duration	12		etay(u)	Auto Gene	eration	Step	Day		
Service Contraction				Step Num	ber 0 🗘	1	7		
Store		Autoliticatial Sheet		Gene	erate Steps	1			
a oraște		C HOURING SEE							
_		Current Stage Information				-			
Flament	Boundary	Load							
aroup List			Activation		Deactivation				
PS-P15eg1									
/S-P1Seg2 /S-P1Seg3		1	Active Day First	 ✓ day(s) 	Inactive Day	First	 day(s) 		
PS-P15eg4 PS-P15eg5			Group List		Croup List				
PS-P15eg6 PS-P1Seg7		1	Name Day	1	Name	Day			
PS-P1Seg8			Self First		2022203				
PS-P1Seg10			PS-PierTable1 First		1				
PS-P15eg12		Element Boundary	Load						
PS-P2Seg1 PS-P2Seg2		Group List		Activation			Deactivation		
PS-P25eg3 PS-P2Seg4		S_FsmLeft		Support	/ Spring Position				
PS-P2Seg5 PS-P2Sep6		S-EsmRight EL_Pier		Origin	o Defo	rmed			
PS-P2Seg7 PS-P2Seg8		EL_FsmLeft EL_FsmRight		Group List	0		Group List		
PS-P2Seg9 pc.prcamm		1.2.2		Name	Positio		Name		
				S_Pier	Origina		127034		
Load increme	enta oteps								
			Element Boundary	Ioat					
			Group List			Activition		Deactivation	
			PS-P1Seg1 PS-P1Seg2		1	Active Da	First y davis	inactive Dev	First w day
			PS-P1Seg3 PS-P1Seg4						
			PS-P1Seg5 PS-P1Seq6			Group Lis	a.	Group List	
			PS-P1Seg7 PS-P1Sep8		1	Name	Day	Name	Day
		I and incompanying During I	PS-P15eg9 PS-R15ea10			Self PS-Piert	First able1 First		
		Coad incremental Steps i	PS-PtSeg11			PS-Pierl	able2 First		
			PS-P15eg12 PS-P2Seg1			FT-PierT	able1 First		
			PS-P2Seg2 PS-P2Seg3			WC-PIS	egt 7		
			PS-P2Seg4 PS-P2Seg5			WC-P25	egt 7		
			PS-P2Seg6 PS-P2Seg7						
			PS-P2Seg8			-		and the second s	
			PS-P25ega			Add	Modity Deleter	Add	Modity Leten
			Load Incremental Steps fi	or Material No	intinear Analysis	6			

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:

Command or Da	ta •STAGE	V Insert Comm	and Insert Data	Delete Data
NAME-NAME, DURATI STEP-DAY1, DAY2, AELEM-GROUP1, AGE DELEM-GROUP1, RED ABNDR-BGROUP1, PO DBNDR-BGROUP1, DA ALOAD-LGROUP1, DA NAME-CS1, 12, YES STEP-7 AELEM-Pier1, 100, ABNDR-S Pier, ORI	ON, BSAVESTAGE, 1, GROUP2, AGE2, 1571, GROUP2, AGE2, 1571, GROUP2, POS ROUP2, Y1, LGROUP2, DAY Y1, LGROUP2, DAY Y25, NO, 5 Pier2, 100, Pie GINAL	DIST2, 2,	Table2, 15	<pre>P ; line ; line ; line ; line ; line ; line ; line ; line</pre>

MCT Command Shell

As shown in figure above, 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 (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)	Add

ee menu	Task Pane			
óde Eleme	ent Bourn	dary Mas	6 Load	
elf Weight			-	
Load Case N	larne :			
Self		~	8	
Load Group	Name			
Default		~	8	
Serf Weight	Pactor			
		Marx Marx		
x 0.1				
v 0				
4				
.oad Case Sett	X Y 0 0	Z Grou -1 Setf	D.	
Dpenation Arts	Modify	Deteta		

Enter self-weight

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

Image: Stage>CS1

Image: Solution Static Loads Option/ Nodal Loads

Image: Select Single (Node : 21)

Load Case Name>FT ; Load Group Name>FT-PierTable1

Options>Add ; FZ (-800), MY (-2000)

Image: Select Single (Node : 29)

Load Case Name>FT ; Load Group Name>FT-PierTable1

Options>Add ; FZ (-800), MY (2000)

Image: Select Single (Node : 71)

Load Case Name>FT ; Load Group Name>FT-PierTable2

Options>Add ; FZ (-800), MY (-2000)

Image: Select Single (Node : 63)

Load Case Name>FT ; Load Group Name>FT-PierTable2

Options>Add ; FZ (-800), MY (-2000)
```

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

The form traveler load is defined according to the construction stages using the same procedure given in stage CS1. $^{\rm G}$



Enter temporary load

- 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 figure below.

Stage>Base 🖗

Properties Tab/ Tapered Group

Name>1stspan Convert to Tapered Section..

New Start Section Number (101) $^{
m eta}$ \downarrow

Properties Tab / Section Properties



Tapered group converted to a cross section

Calculate the self-weight of each segment using the Bill of Material function. In figure below, 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) لم

					Bill of Materia	4			
					Select BOM o	utputs			
					Beam-Tru: Beam-Tru: Beam-Tru: BOM by M	ss Element BC ss Element BC ss Element BC aterial	M type1 M type2 M type3		
					Output Opti	ons			
					Insert for	m feed symbo	l at each outp	out end	
					File Name	C:\Users\D	ELL\AppData	Local\Temp\5 F	CM Ge
AS/Text Editor ~ [5 FCM	General NX.bon	n]						ок	
Edit View Window	u Helo	-9		ļ	5				2 4
	XXX 1	KN XXX KN XXX	XXX X XXXXXXXX VERSION	CX XXX 1 XXX V 5.4.5	XX XX XX XXXXX	XXX XXX /Civil			
******	COPYRIGHT	(C) SINCE 19 ALI	89. MIDAS RIGETS R	3 Informati ESERVED.	on Technolog	y Co.,Ltd.	*****		
******	COPYRIGHT	(C) SINCE 15 ALL BII	89. MIDAS RIGEIS R	S Informati RESERVED.	on Technolog	y Co.,Itd.	*******		
BEAN & TRUSS BON	COPYRIGHT	(C) SINCE 15 ALL BII ID, SECT NAP	E9. HIDAS RIGETS R L OF MATE	3 Informati MESERVED. MINING M	on Technolog	y Co.,Itd.	.: ка, т		
BEAR 5 TRUSS BOR SECT SECTION	COPYRIGHT I TYPEI SECT	(C) SINCE 15 MI BII ID, SECT NAP MATTERIJ ID	<pre>09. MIDAS RIGETS R L OF MATE L OF MATERI L NAME</pre>	Informati MESERVED.	LENGTH	y Cs., Ltd. Unit System PAINT INNER	s : kB , т area outer	REIGHT	
BEAH & TRUSS BOH	COPYRIGHT	(C) SINCE 15 MII BII ID, SECT NAP ID, SECT NAP ID ID I Grad	<pre>k9, MIDAS RIGETS R L OF MATE L. MATERI L. NAME k6 C5000 k6 C5000</pre>	Informati RESERVED. REAL LENSITY	LEDGTH	Unit System PAINT INNER 9.9160+02	AREA OUTER 1.9010+03	WEIGHT	
BEAN & TRUSS BOM SECT SECTI ID 2 Span 3 Support - 5	COPYRIGHT I TYPEI SECT	(C) SINCE 15 ALI BII ID, SECT NAP ID 1 Grad 1 Grad 1 Grad 1 Grad	<pre>k9, MIDRS RIGETS R L OF MATE L K NAME k k KAME k c SC000 k c SC000 k c SC000 k c SC000 k c SC000</pre>) Informati HESENVED. HIRL HAL DENSITY 2.3560+01 2.3560+01 2.3560+01	LENGTH 6.300e+01 1.7200e+01 4.750e+00	y Cs., Ltd. Unit System PAINT INNER 9.5156+00 2.7796+02 7.4656+01	н з k2 , л ОUTER 1.9010+03 4.552e+02 1.330e+03	WEIGHT 1.260e+04 4.522e+03 5.522e+03	
BEAN & TRUSS BOM SECT SECTI ID 2 Span 3 Support-S 7 Support-S	COPYRIGHT I TYPEI SECT OR NAME	(C) SINCE 15 MLI BII ID, SECT NAP ID ID ID I Grad I Grad I Grad I Grad I Grad	<pre>k9, MIDAS RIGHTS R L OF MATE L KAME ke C5000 ke C5000 ke C5000 ke C5000</pre>	3 Informati ESERVED. INFORMATIONI INFORMATIONI INFORMAT	con Technolog LERSTH 6.300e+01 1.2009+01 4.759e+00 4.759e+00 4.759e+00	y Co., Ltd. Unit System EXINT 10028 9.9160402 2.7796402 7.4696401 7.4699401 7.4699401	ала и кала и ка	WEIGHT 1.260e+04 4.520e+03 9.5245+03 9.5245+02 9.5245+02	
BEAN & TRUSS BOH SECT SECTI 1D 2 Span 3 Support- 6 Support- 9 Support-9	COPYRIGHT I TYPEI SECT CO RAME	(C) SINCE 15 ALL BIL ID, SECT NAP ID ID, SECT NAP ID ID I Grad I Grad I Grad I Grad I Grad I Grad I Grad	<pre>k9, MIDAS RIGHTS R L OF MATE HE, MATERI L HAME ke C5000 ke C5000 ke C5000 ke C5000 ke C5000 ke C5000</pre>	3 Informati ESENVED. (RIAL 	LENSTH 6.300e+01 1.709e+01 4.750e+00 4.750e+00 4.750e+00 4.750e+00 4.750e+00 4.750e+00	y Cs., Ltd. Unit System PAINT INNER 9,3150402 2,7738402 7,659401 7,639401 7,639401 7,639401 7,639402	AREA OUTER 1,901+03 4,552+02 1,430±02 1,430±02 1,430±02 1,430±02 1,430±02	MEIGHT 1.360e+04 5.320e+02 5.826e+02 5.846e+02 5.846e+02 1.020e+03	
BEAH & TRUSS BOH SECT SECTI 10 2 Span 3 Support-S 9 Support-S 9 Support-S 10 Support-S 10 Support-S	COPYRIGHT I TYPEI SECT COD RAME Joan Joan Joan Joan	(C) SINCE 15 ALL BII ID, SECT NAP ID ID ID IG Grad I Grad	<pre>k9, MIDAS RIGETS R L OF MATE L L KAME k6 C5000 k6 C5000 k6 C5000 k6 C5000 k6 C5000 k6 C5000</pre>	3 Informati ESERVED. TRIAL DENSITY 2.356e+01 3.556e+01 3.556e	LERSTH 6.300e+01 1.200e+01 1.200e+01 4.750e+00 4.750e+00 4.750e+00 4.750e+00 4.750e+00 4.750e+00	y Co., Ltd. Unit System PAINT INEER 9.9156402 2.775402 7.659401 7.559401 7.559401 7.592401	а: kH , л OUTER 1.901ею 1.432+02 1.432+02 1.452+02 1.452+02 1.452+02	MEIGH: 1.360+04 4.328+03 5.438+02 5.458+02 1.0226+03 1.0226+03 1.0226+03 1.0726+03	
* BEAH & TRUSS BOH ID 2 Span 3 Support-9 7 Support-9 9 Support-9 10 Support-9 11 Support-9 11 Support-9	COPYRIGHT I TYPEI SECT COB RAME Joan Dan Dan Dan Dan Dan Dan Dan Dan	(C) SINCE 15 ALL BIL ID, SECT NRP 1D 1 Grad 1 Grad 1 Grad 1 Grad 1 Grad 1 Grad 1 Grad	<pre>#89. MIDAS RIGHTS R L OF MATE E, MATERI # C5000 # C5000 # C5000 # C5000 # C5000 # C5000 # C5000 # C5000 # C5000</pre>	3 Information ESERVED. Information Infinite Infi	LERASTH 6.300e+01 1.2008471 4.7509400 4.7509400 4.7509400 4.7509400 4.7509400 4.7509400 4.7509400	y Co., Ltd. Unit System PAINT INUER 9.9156+02 2.7756+02 7.405+01 7.405+01 7.405+01 7.405+01 7.405+01 7.405+01 7.405+01 8.445+01 8.445+01 8.455+010000000000000000000000000000000000	AREA OUTER 1. 5010403 4. 552402 1. 43040402 1. 43040402 1. 4520402 1. 4520402 1. 4520402	WIIGHT 1.360e+04 4.321e+03 5.421e+02 5.421e+02 1.021e+03 1.0	
* BEAN & TRUSS BOM ID 2 Span 3 Support 6 Support-9 9 Support-9 10 Support-9 11 Support-9 11 Support-9 13 Support-9	COPYRIGHT	(C) SINCE 15 ALL SIL ID, SECT NAP ID ID ID I Grad I	<pre>89. MIDAS RIGETS R L OF MATE L E, MATERI HAME HE C5000 HE C5000</pre>	3 Informati HESERVED. HELL HELL DENSITY 2.3560+01 3.3560+01 3.3500+000+000+000+000+000+000+000+000+000	LEMSTH 	vy Cs., Ltd.	ат кала и кал и кала и кал и кала и кал и кала и кал	MEIGHT 1.3600+04 4.522+03 5.524+02 5.8464+02 5.8464+02 1.022+03 1.022+03 1.022+03 1.022+03 1.032+03 1.032+03 1.032+03	
BEAM & TRUSS BOM SECT SECTI 1D 2 Span 3 Support-9 9 Support-9 10 Support-9 11 Support-9 13 Support-9 13 Support-9 14 S	COPYRIGHT I TYPEI SECT CO RAME CO RAME Joan Joan Joan Joan Joan Joan Joan	(C) SINCE 15 ALL BIL ID, SECT NEP MATERIJ ID ID ID I Grad I Grad	<pre>469. MIDAS RIGHTS R L OF MATE E. NATERI E. NATERI E. CS000 E CS000 E CS000 E CS000 E CS000 E CS000 E CS000 E CS000 E CS000</pre>	3 Information Inf	LEINTH 6.300e401 1.2009401 4.7509400 4.7509400 4.7509400 4.7509400 4.7509400 4.7509400 4.7509400 4.7509400 4.7509400	y Cs., Ltd.	и: kB , л ОUTER ОUTER 1.901е+03 4.652e+02 1.430e+02 1.430e+02 1.430e+02 1.532e+02 1.532e+02 1.532e+02 1.539e+02	MEIGHT 1.360+04 4.520+02 9.830+02 1.020+03 1.073+03 1.020+03 1.200+03 1.300+03 1.300+03	
BEAH & TRUSS BOH SECT SECTI 10 2 Span 3 Support-9 9 Support-9 9 Support-9 10 Support-9 11 Support-9 13 Support-9 13 Support-9 13 Support-9 15 Su	COPYRIGHT I TYPEI SECT COD RAME Open pan pan pan pan pan pan pan pan pan pa	ID, SECT NEW ID, SECT NEW ID, SECT NEW ID, SECT NEW IG Grad I G I G I G I G I G I G I G I G I G I G	 KIDAS RIGETS R RIGETS R RIATERI L L OF MATERI E, NATERI E, NATERI E, NAME E, CS000 E,	3 Information ESERVED. TRIAL DENSITY 2.356e+01 3.356e+01 3.3	LEERSTH 300e+01 200e+01 200e+01 200e+00 750e+0	y Co., Ltd.	AREA OUTER 1. 501e+03 4. 552e+02 1. 430e+02 1. 430e+02 1. 430e+02 1. 430e+02 1. 430e+02 1. 430e+02 1. 430e+02 1. 552e+03 1. 552e+03 1. 552e+03 1. 556e+03 1. 556e+03	MEIGH: 1.360+04 4.328+02 5.428+02 5.468+02 1.078+03 1.028+03 1.388+08 1.488+088+08 1.488+088+08 1.488+088+088+088+088+088+088+088+088+088+	
BEAH & TRUSS BOH SECT SECTI ID 2 Span 3 Support-S 7 Support-S 9 Support-S 10 Support-S 11 Support-S 13 Support-S 14 Support-S 15 Support-S 15 Support-S 16 Support-S 16 Support-S 16 Support-S 17 Support-S 16 Support-S 17 Support-S 16 Support-S 17 Support-S 16 Support-S 17 Support-S 16 Support-S 17 Support-S 17 Support-S 18 Support-S 19 Support-S 10 Su	COPYRIGHT COPYRIGHT I TYPEI SECT OR RAME Joan Joan Joan Joan Joan Joan Joan Joan	(C) SINCE 15 ALL BILL ID, SECT NAP 10 10 1 Grad 1 G	 KIDAS RIGHTS R RIGHTS R RATE L DF MATE IE, MATERI IE, MATERI IE, MATERI IE, MATERI IE, C5000 IE C50	3 Informatia HESERVED. ALL UENSITY 2.3560+01 3.3560+01 3.3500+000+000+000+000+000+000+000+000+000	LERASTH 6.300e+01 1.2009+01 4.750e+00 4.	y Cs., Ltd. Whit System PAINT INNER 9,9150402 2,7459401 7,7694401 7,7694401 7,7694401 7,7694401 7,7694401 8,2458401 8,2458401 8,2458401 9,25944000000000000000000000000000000	ABEA OUTER OUTER 000000 1.4010+03 4.522+02 1.432+02 1.432+02 1.432+02 1.522+02 1.522+02 1.522+02 1.532+02 1.532+02 1.532+02 1.552+02 1.552+02	MXIGHT 1.360e+04 4.328+03 5.328+02 5.488e+02 1.075e+03 1.038e+03 1.338e+03 1.388e+03 1.488e+03 1.488e+03 1.488e+03 1.488e+03 1.488e+03 1.488e+03 1.488e+03 1.488e+03 1.488e+03 1.488e+03 1.488e+03 1.488e+03 1.488e+03 1.488e+03 1.488e+03 1.484	
* BEAN & TRUSS BOH ID 2 Span 3 Support 6 Support 9 Support 10 Support 11 Support 13 Support 13 Support 13 Support 13 Support 14 Support 15 Supp	COPYRIGHT I TYPEL SECT OR'RAME Span Span Span Span Span Span Span Span	(C) SINCE 15 ALL SIL ID, SECT NAP ID ID ID I Grad I	K69. MIDAS RIGHTS R RIGHTS R L OF MATE KIAME IE, MATERI IE, MATERI IE C5000	3 Informati HIRL	LENGTH LENGTH 6.300e401 1.2009401 1.2009401 4.75094000 4.75094000 4.75094000 4.75094000 4.75094000 4.75094000 4.7509400000000000000000000000000000000000	y Cs., Ltd.	ат kt2 , л оттек отт	MEIGHT 1.3600+04 4.520+03 5.520+03 5.520+03 1.020+03 1.020+03 1.020+03 1.020+03 1.030+03 1.030+03 1.030+03 1.030+03 1.030+03 1.030+03 1.030+03 1.030+04 1.020+04 1.000+0	
BEAN & TRUSS BOH SECT SECTI 1D 2 Span 3 Support-9 9 Support-9 10 Support-9 11 Support-9 11 Support-9 12 Buecorr-9 13 Support-9 14 Support-9 15 Support-9 15 Support-9 15 Support-9 16 Support-9 17 Support-9 16 Support-9 17 Support-9 18 Support-9 19 Support-9 19 Support-9 10 S	COPYRIGHT I TYPEL SECT COF RAME COF RAME COF RAME COF RAME Span Span Span Span Span Span Span Span	(C) SINCE 15 ALL BIL ID, SECT NAP NATERIA ID ID I Grad I G	 KIDAS RIGETS R RIGETS R RIGETS R R COF MATE COO COO<td>Information Infor</td><td>LENSTH 6.300e401 1.2009401 4.75094000 4.75094000 4.75094000 4.7509400000000000000000000000000000000000</td><td>y Co., Ltd.</td><td>ас kd , т ОUTER ОUTER 1. 5010+03 4. 652e+02 1. 634e+02 1. 634e+02 1. 632e+02 1. 632e+02 1. 632e+02 1. 632e+02 1. 552e+02 1. 552e+02 1. 552e+02 1. 552e+02</td><td>MEIGHT 1.360e+04 4.520e+03 5.820e+02 5.880e+02 1.020e+03 1.030e+03 1.030e+03 1.640e+03 1.640e+03 1.640e+03 1.640e+03 1.640e+03 1.640e+03 1.640e+03 1.640e+03</td><td></td>	Information Infor	LENSTH 6.300e401 1.2009401 4.75094000 4.75094000 4.75094000 4.7509400000000000000000000000000000000000	y Co., Ltd.	ас kd , т ОUTER ОUTER 1. 5010+03 4. 652e+02 1. 634e+02 1. 634e+02 1. 632e+02 1. 632e+02 1. 632e+02 1. 632e+02 1. 552e+02 1. 552e+02 1. 552e+02 1. 552e+02	MEIGHT 1.360e+04 4.520e+03 5.820e+02 5.880e+02 1.020e+03 1.030e+03 1.030e+03 1.640e+03 1.640e+03 1.640e+03 1.640e+03 1.640e+03 1.640e+03 1.640e+03 1.640e+03	

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 selfweight 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 (-173.0), MY (-173.0*2.5)
Select Single (Node : 29)
Load Case Name>WC ; Load Group Name>WC-P1Seg1
Options>Add ; FZ (-173.0), MY (173.0*2.5)
Select Single (Node : 71)
Load Case Name>WC ; Load Group Name>WC-P2Seg1
Options>Add ; FZ (-173.0), MY (-173.0*2.5)
Select Single (Node: 63)
Load Case Name>WC ; Load Group Name>WC-P2Seg1
Options>Add ; FZ(-173.0), MY(173.0*2.5)



Enter self-weight of wet concrete



Enter 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.

Tendon Name		Тор				
Tendon Ty	pe		Internal(Post-Tensio	n) ~		
Material		3	3- Tendon	~		
Total Tendon Area		0.0026353	m²			
Duct Diame	ster		0.103	m	×	
🛛 Relaxatio	on Coefficient		Magura 🗸	45 🗸	Tendon Area	
Name				~ 📖	Strand Diameter	15.2mm(0.6")
Ultimate SI	trength		1.86326e+06	kN/m ²	Number of Strands	19
Yield Stren	igth		1.56906e+06	kN/m ²		1
Curvature I	Friction Factor		0.2		ОК	Cancel
Wobble Fri	ction Factor		0.001	1/m		
External Ca	able Moment Ma	gnifier	0	kN/m²		
Anchorag	e Slip(Draw in)		Bond Type			
Begin	0.006	m	O Bonded			
End	0.006	m	Unbonded			

Enter tendon characteristics



Tendon Arrangement



Tendon Arrangements for the Side Span



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 figure above.

```
Tree Menu > Group>Structure Group>PierTable1>Active
Load Tab / (Temp./ Prestress) Option / Tendon Profile
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 )
<math>3>x ( 12 ), y ( 0 ), z ( -0.15 ), fix (on) , Ry ( 0 ), Rz ( 0 )
<math>4>x ( 14 ), y ( 0 ), z ( -0.3 ), fix (off)
Tendon Shape>Straight
Profile Insertion Point ( 78, -3.09, 0 )
X Axis Direction>X \downarrow
```



Tendon Profile Definition

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





Copy Tendon Profile

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

Define each of the tendon profiles using the same procedure. $^{igodoldsymbol{\Theta}}$

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 D Tendon Tendon>P1TC2L, P1TC2R Selected Tendons Stress Value>Stress ; 1st Jacking>Begin Begin (1330000); End (0) Grouting : after (1) Load Case Name>PS ; Load Group Name>PS-PierTable2 Selected Tendons>P1TC2L, P1TC2R D Tendon Tendon>P2TC1L, P2TC1R Delected Tendons Stress Value>Stress ; 1st Jacking>Begin Begin (1330000); End (0) Grouting : after (1) Load Case Name>PS ; Load Group Name>PS-PierTable2 Selected Tendons>P2TC1L, P2TC1R Definition Tendon>P2TC2L, P2TC2R Selected Tendons Stress Value>Stress ; 1st Jacking>Begin Begin (1330000); End (0) Grouting : after (1)

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

- 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".



1330000

101

CO MARKS



•

a +)+ +(0) [(00) +

- The prestress may be defined more easily using the MCT Command Shell. The MCT command for prestress is "*TDN-PRESTRESS".
- Apply prestress at each construction stage using the same procedure. $^{\ensuremath{\Theta}}$

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

Load Group Name>TimeLoad

Options>Add

Time Loads (60)
```



Enter Time Load

Perform Structural Analysis

We will now perform structural analysis.



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