ADAPT

STRUCTURAL CONCRETE SOFTWARE

ADAPT-ABI 2019

GETTING STARTED GUIDE

Copyright 2019

LIST OF CONTENTS

ADAPT-ABI 2019 Getting Started Guide

Getting Started

Thank you for choosing ADAPT-ABI 2019. This Getting Started Guide is intended to provide you with a brief overview of the capabilities of the ADAPT-ABI 2019 product. In following the tutorials below, you will learn how to model a structure and perform the necessary design steps. For more details about the program please refer to the ADAPT-ABI user manuals.
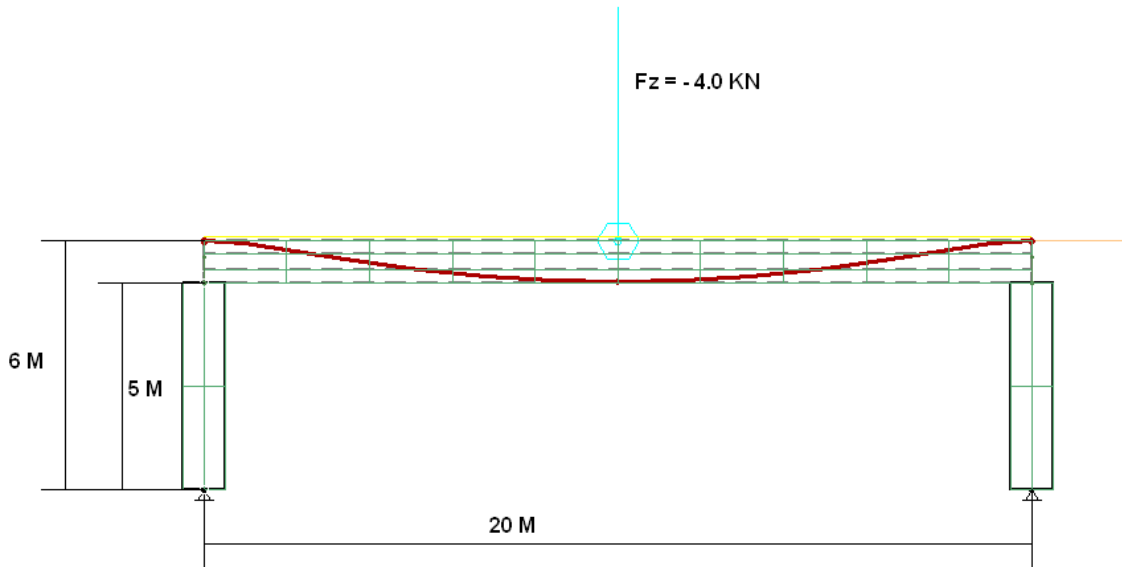
1   ADAPT-ABI 2019 USER INTERFACE
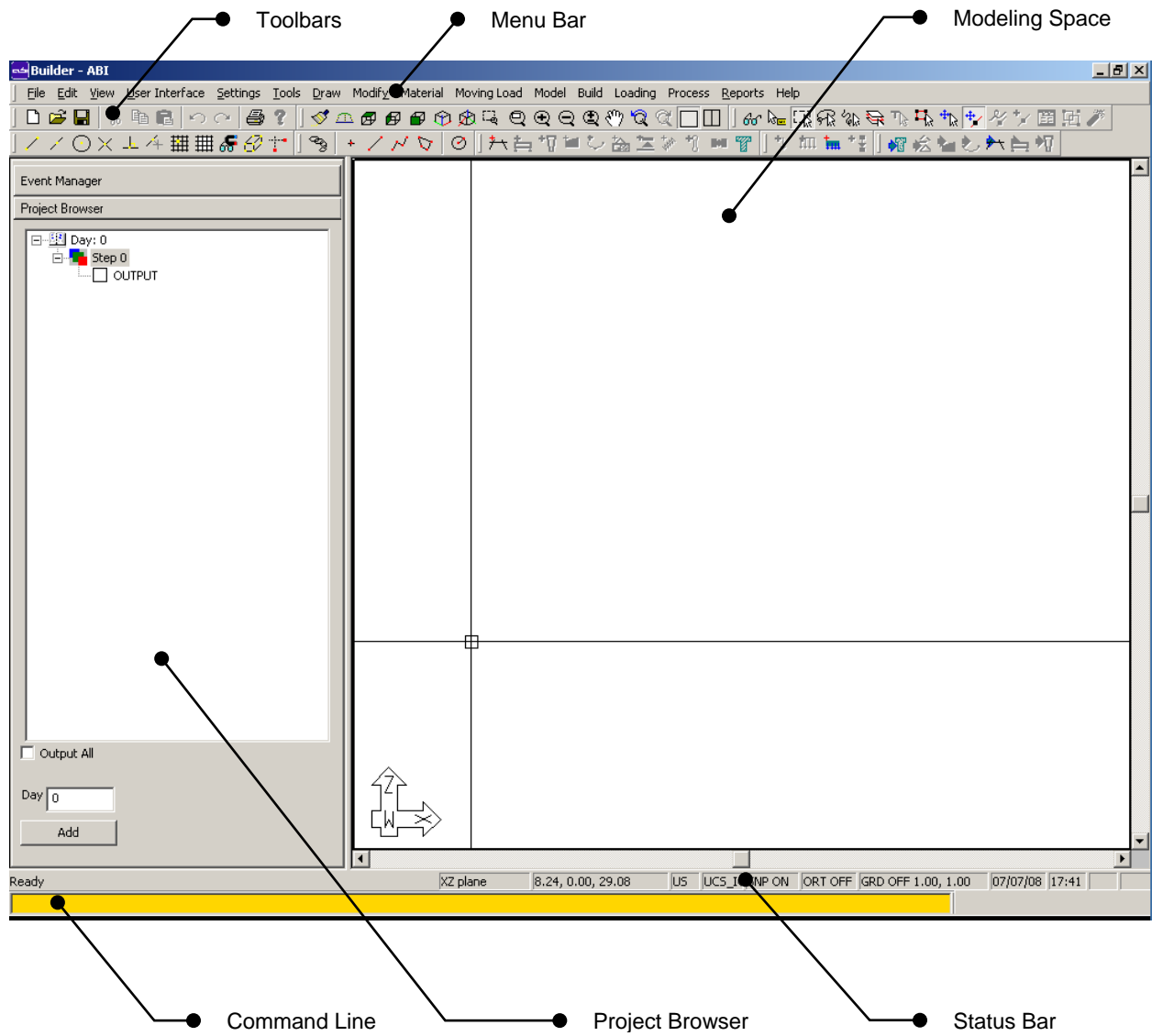
2 TUTORIAL 1: MODELING OF SINGLE SPAN BRIDGE WITH MOVING LOAD IN DIFFERENT STAGES

The objective of this tutorial is to model a single span bridge girder with moving load.

The cross section used is a single cell box girder with 1 m depth and 5 m width (**Fig. 2-1**).
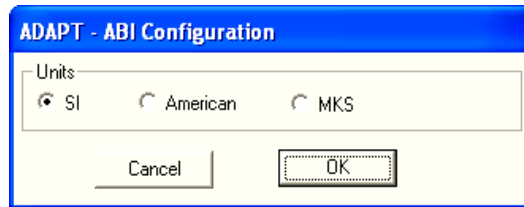

FIGURE 2-1

| Span Section	Single cell box girder cross section of 5000 mm width and 1000 mm depth.
Pier Section	Rectangular cross section of 1000 mm width.
Loadings	Point load of 4 kN will be applied at a distance of 10 meters from left end.
Moving Loads	1. ABI_MIXED 2. ABI_TRUCK
Material Properties	1. Concrete: default ACI concrete model 2. Mild Steel: default material properties
Support	Fixed supports are used at the both ends.
Tendon	200 KN stressing at both ends.
Analysis	Execute ABI and view analysis results graphically and in text format

2.1 GENERATE THE MODEL

To model and analyze a segmental bridge in ABI 2019 follow the following steps:

1. Go to *Start* menu in Windows, select *ADAPT-Builder ABI 2019* under *ADAPT Application* program group.

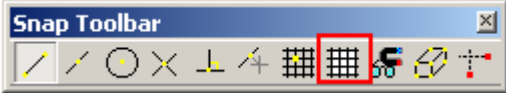2. Program will display *ADAPT-ABI Configuration* dialog box as shown in **Figure 2.1-2**.

FIGURE 2.1-2

3. Select *SI* unit and click OK. ABI 2019 will appear on the screen.

4. At this point, if required, please go to the *User Interface* menu (**Fig. 2.1-3**) and turn on any or all required toolbar(s) for your operation. However, for beginners this step is not recommended.
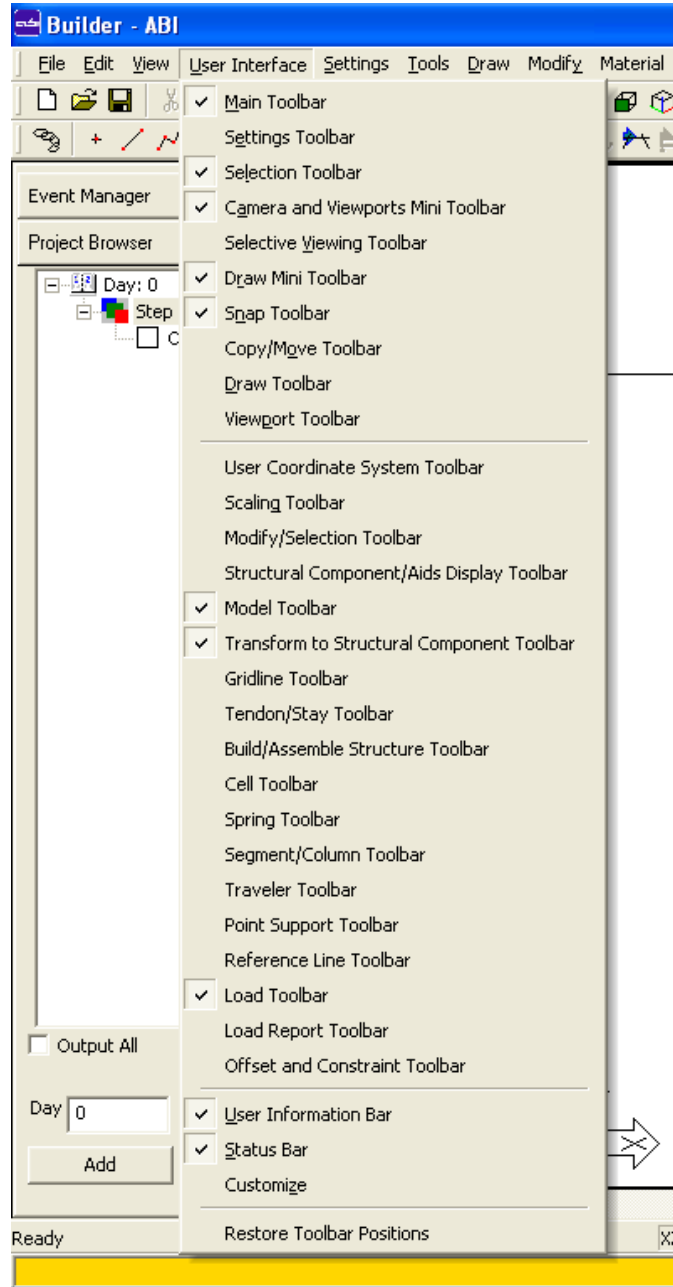
FIGURE 2.1-3

5. On the *Snap Toolbar* click on the *Grid Setting* tool and the program will display the dialog box on your screen as shown in **Figure 2.1-4.**
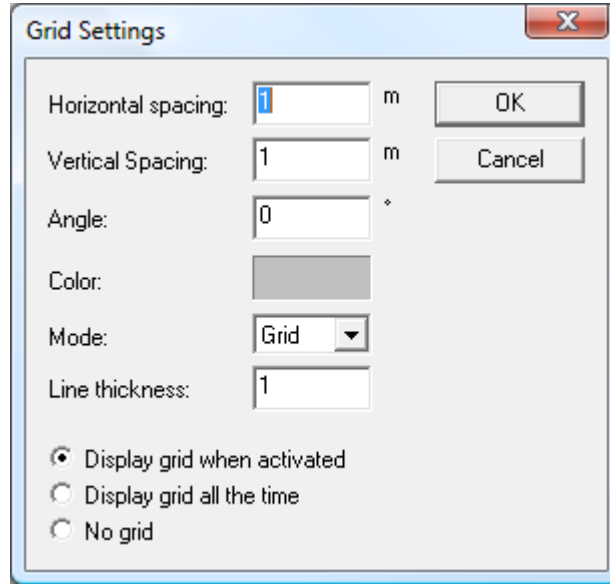
FIGURE 2.1-4

6. Let us specify 5 m for both *Horizontal Spacing* and *Z spacing*. Now click OK.

7. Then click on *Snap to Grid* ⊞ on the *Snap* toolbar to view the grid lines and create points only at the intersections of the grid lines.

8. Then click on *Display WCS* ⊥ on the *Setting* toolbar to view the WCS on your screen.

9. Click on *Create a Reference Line* ⊼ icon on the *Model* toolbar (**Fig. 2.1-5**).

FIGURE 2.1-5

10. Enter first point and last point at a distance of 40 m in X direction, then Press 'C' from the keyboard. Program will display the reference line on the screen as shown in **Figure 2.1-6.**

8

FIGURE 2.1-9

15. In *Section Manager* go to *Properties* and select *Single cell girder*. Put Bf = 5000 mm, B = 2000 mm, t = 300 mm, Hf = 300 mm, H= 1000 mm, a = 600 mm. Then click on the green check mark. Again, go to *Properties* and select *Rectangular*. Put width b = 1000mm, height h = 1000 mm and click on *Add*. Then press 'OK' to save sections in section manager.

16. Double click on the segment and the program will display the dialog box named *Segment* on your screen as shown in **Figure 2.1-10.** In that box go to *Section* tab, select *Section 1* for Starting Cross Section and Ending Cross Section. Then click on the green check mark and exit the property box.
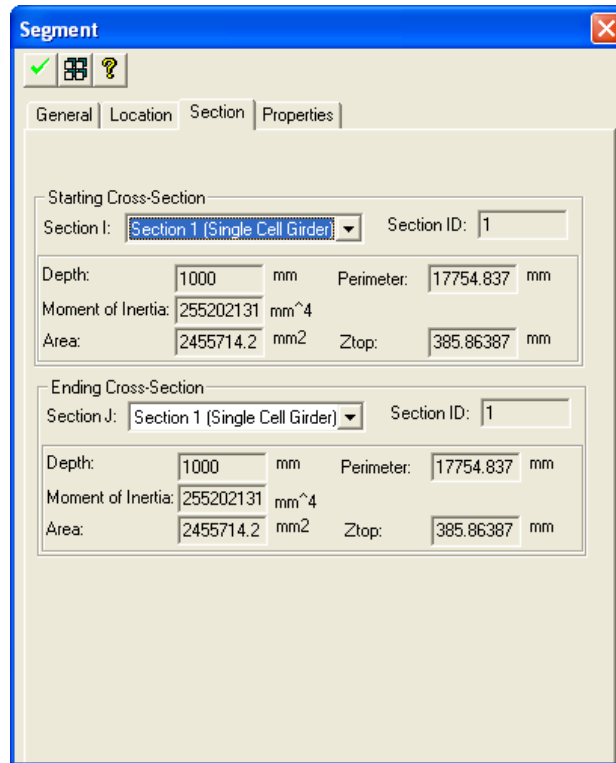
FIGURE 2.1-10

17. Click on *Create/Identify a Column or Pier* on the *Model* toolbar.

18. Enter most left end and most right end of the segment respectively to create two columns of 5 m height.

19. Double click on the 1st column and the program will display the dialog box named "*Pier/Column*" on your screen, same as shown in **Figure 2.1-11**. In the '*Section*' Tab, select 'Section 2' for both Starting Cross Section and Ending Cross Sections respectively. Then click on the green check mark and exit. Follow the same steps for 2nd column as well.
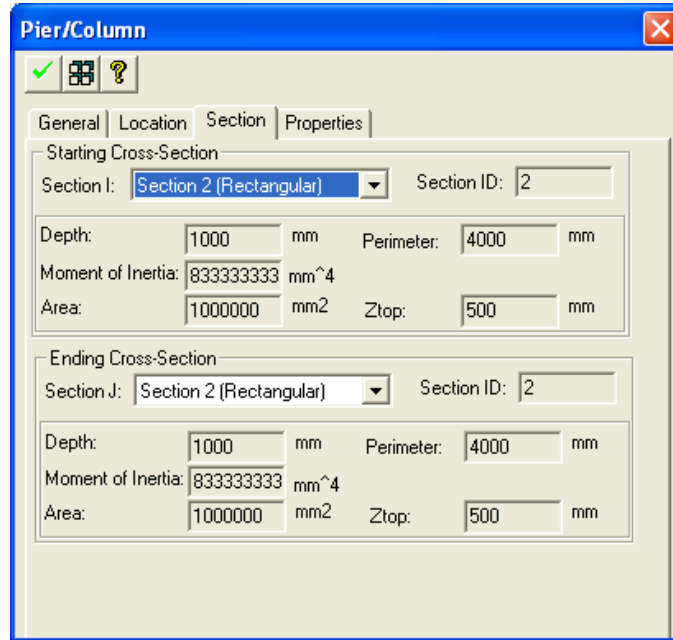
FIGURE 2.1-11

20. Click *Concrete* under the *Material* menu bar. Program will display the dialog box as shown below. Select *ACI 1978* on the *Concrete Model* tab (**Fig. 2.1-12**).
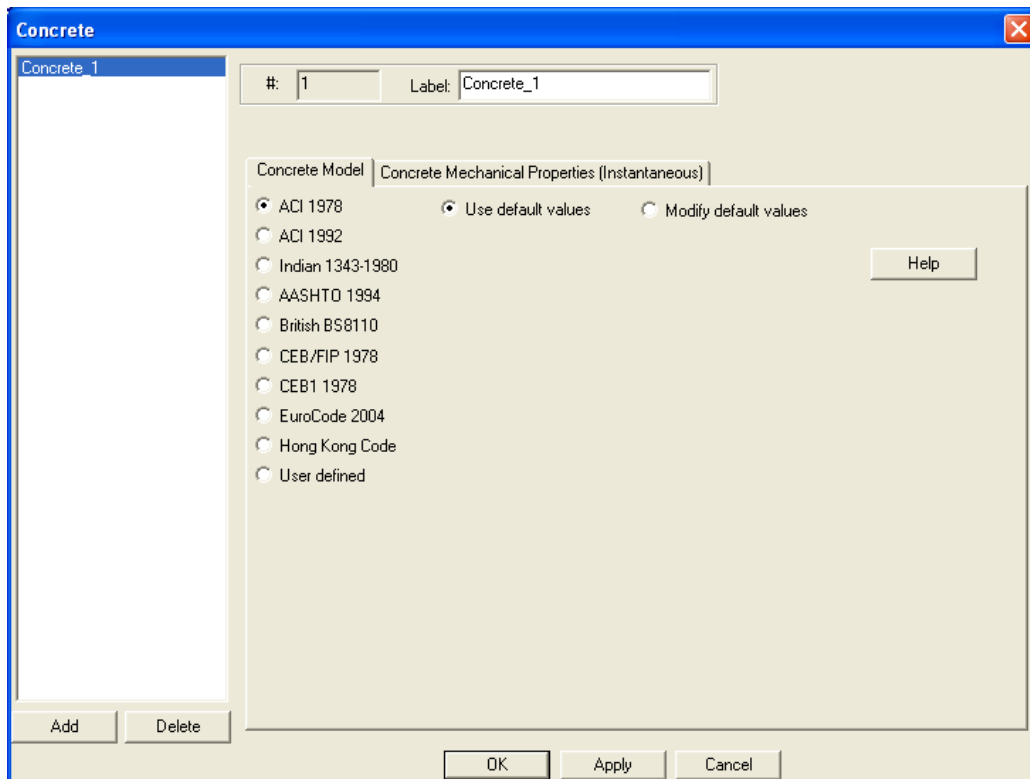
FIGURE 2.1-12

21. Click on *Create a Point Support* ⤒ on the *Model* toolbar and enter the point position of support by clicking on the end points of each column. It is best to turn off the grid and to select the *Snap to Endpoint* ╱ option instead.

22. Double click on 1st support and the program will display the property box *Point Support* as shown in **Figure 2.1-13**. On the *General* tab select *Fixed* under *Degree freedom* and then click on the green check mark and exit. Follow the same steps for 2nd point support as well.
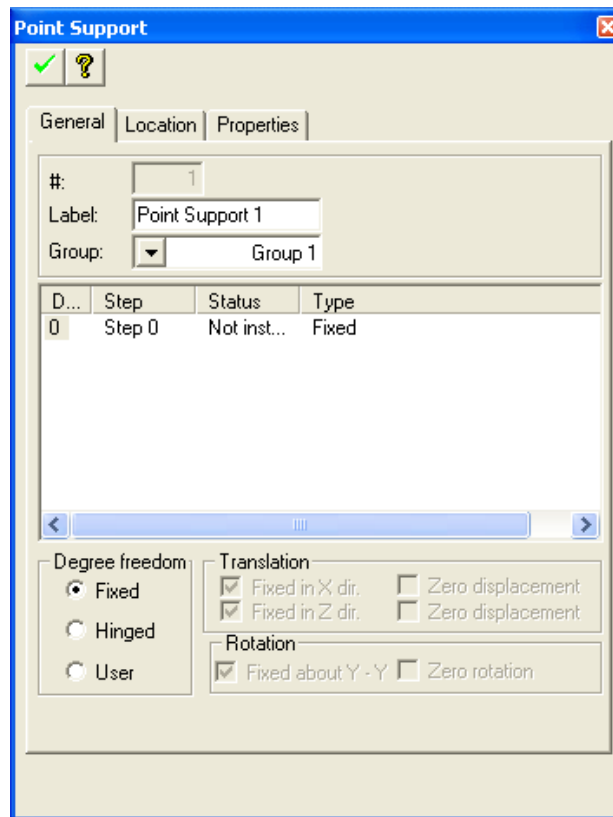
FIGURE 2.1-13

23. Click on the *Create a Tendon* menu under the *Model* toolbar. Enter the Left End and Right End points of segment respectively and Press 'C' on your keyboard. Program will display the tendon in the structure as shown in **Figure 2.1-14.**
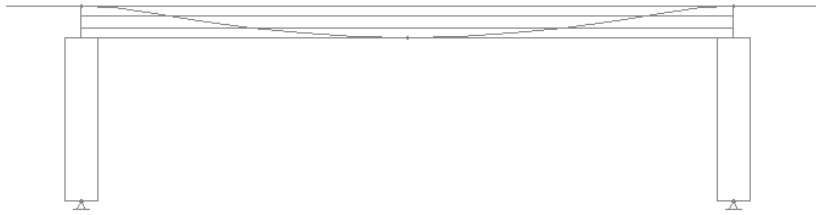
FIGURE 2.1-14

2.2 DEFINE THE LOADING

24. Click *Load Case Library* (**Fig. 2.2-1**) under the *Loading* menu bar. The Program will display the dialog box as shown below. Click the *Add* button. Highlight "Load_Case_1" and insert "DL" into the *Label* filed. Then click OK.
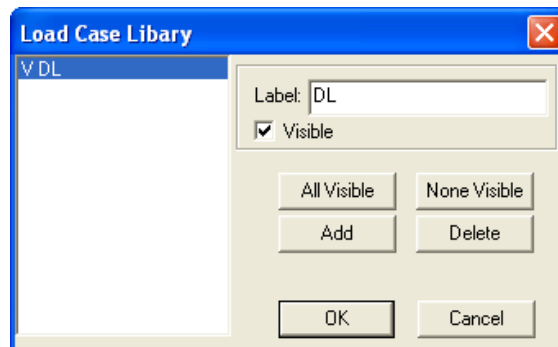
FIGURE 2.2-1

25. Click on *Create Point Load* ⬇ on the *Load* toolbar. Enter the point at which to apply the point load at a distance of 10 meter from left end.

FIGURE 2.2-2

26. Double Click on the point load and the program will display the property box named *Point Load* as shown below. On the *General* tab put the value Fz = - 4.000 KN and change the load case name as 'DL' (**Fig. 2.2-3**). Click on the green check mark and exit.
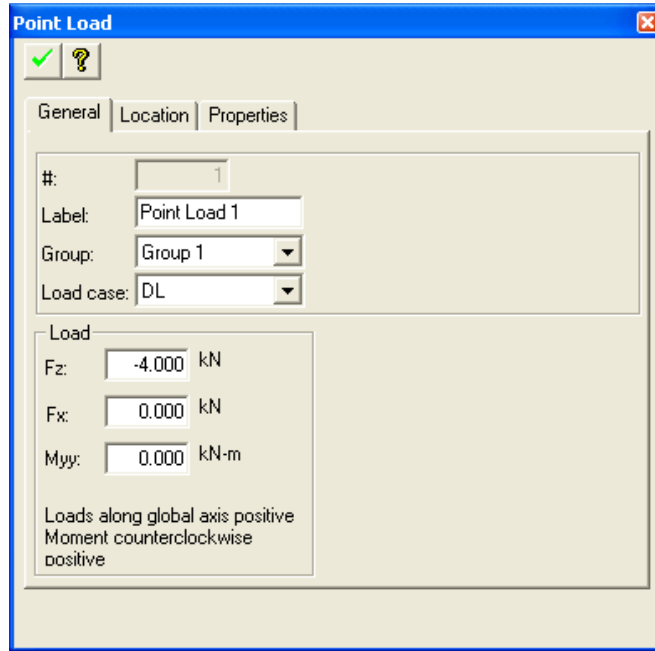
FIGURE 2.2-3

27. Click on *Add Selfweight* on the *Load* toolbar. Put the value Gz = -1.00 (**Fig. 2.2-4**). Press *OK*.

FIGURE 2.2-4

28. Go to the *Travel Path* option under *Moving Load* menu. Select the left end of the span as the 1st point and the right end of the span as the 2nd point of the travel path. This way you create a travel path along the entire span.

29. Go to the *Moving Load Library* option under *Moving Load* menu. Observe all the moving load trains that are available in the existing library. You will be using *ABI_MIXED* and *ABI_TRUCK* in the following step.

30. Go to the *Load Case* option on the *Moving Load* menu and the program will display the dialog box named *Moving Load Cases* on your screen, same as shown in **Figure 2.2-5**.

Select *ABI_MIXED* under *Load trains*. Select *Travel_Path_1* in the *Travel paths* box. Click on *Both* under *Factors and travel directions*. Click the ADD button in the

Combination List. Click the *ADD* button in *Moving load cases* to create the load case called "Case1".
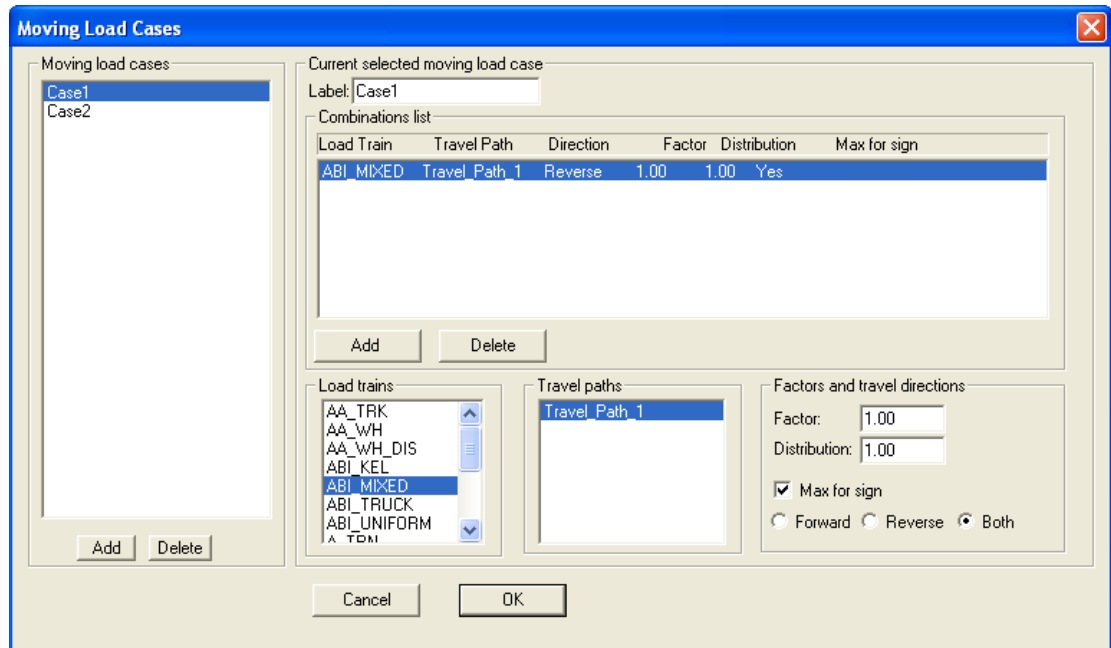
FIGURE 2.2-5

31. To create a 2nd load case, first create "Case2" by clicking on *Add* under *Moving load cases*. Then highlight "Case2", select *ABI_TRUCK* under *Load trains*, select *Travel_Path_1* under *Travel paths* box, and then click on *Both* under *Factors and travel directions*. Click the *ADD* button in *Combinations list*.

The program will display both load cases in the list named *Moving Load Cases* on your screen same as shown in **Figure 2.2-6**.
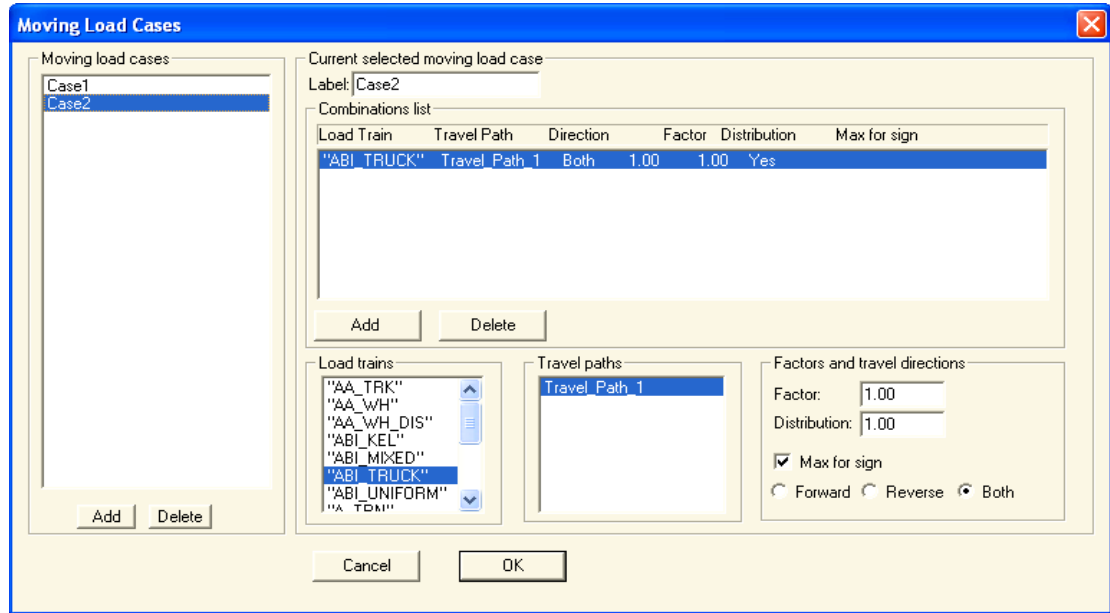
FIGURE 2.2-6

32. Click *Load Combinations / Envelopes* in the *Loading* menu. Set the factors for *DL* and *Case1* to 1 and click on *Create Load Combination*. Set the factors for *DL* and *Case2* to 1 and click on *Save As Load Combination*. Next, set the factors for *Case1* and *Case2* to 1 and click on *Save As Load Combination*. The program will display all three load combinations in the dialog box as shown in **Figure 2.2-7**. Then click *OK*.

The "Base_load" load case includes all loads applied to the structure in the construction phase, excluding prestressing and long-term effects.

The "Current" load case includes all loads applied to the structure in the construction phase, including prestressing and long-term effects.
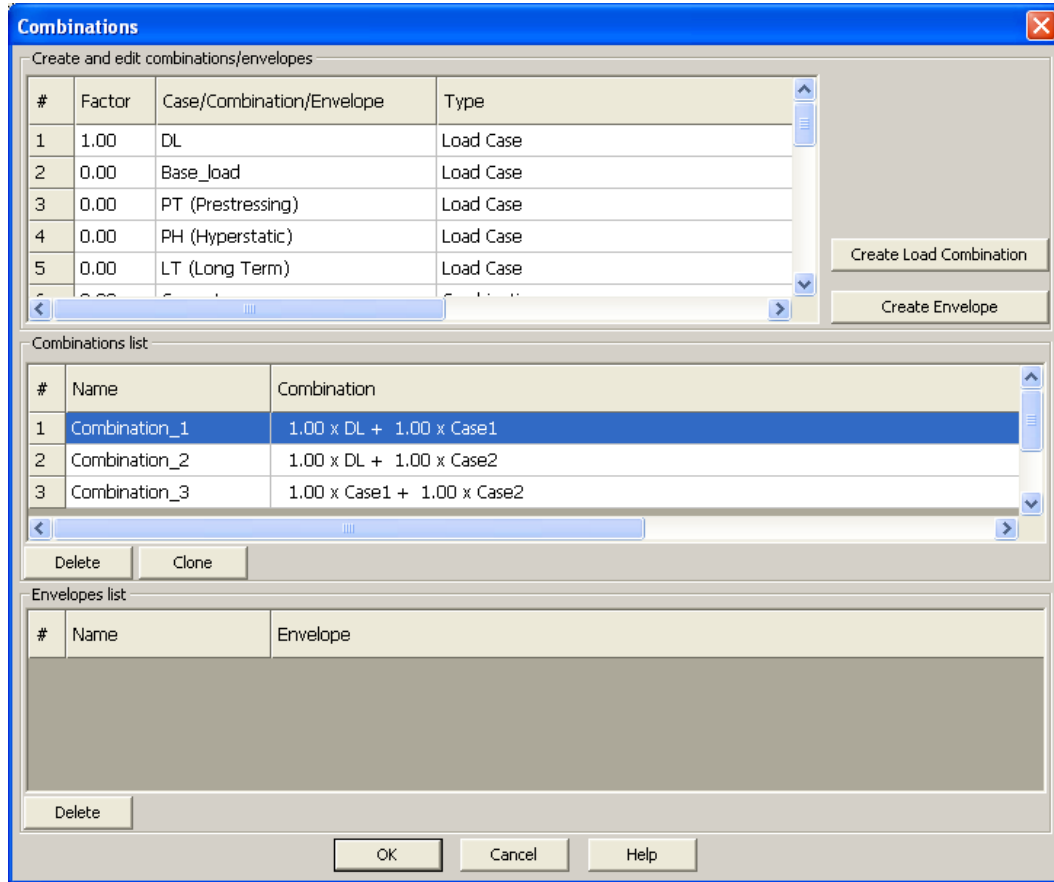
FIGURE 2.2-7

2.3 DEFINE CONSTRUCTION STAGES

33. Click *Create Mesh* in the *Proc*ess menu. The program will display the dialog box as shown in **Figure 2.3-1**. Insert a value of 2 m for *Suggested cell size*. Then click *OK*.
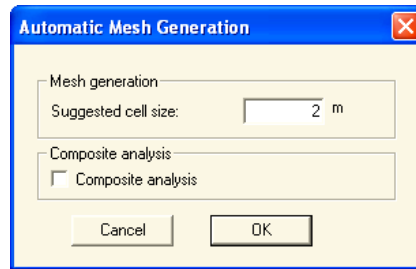
FIGURE 2.3-1

34. In the Project Browser, create the following days: 1, 10, 20 and 30. Simply enter "1" into the *Day* field and hit *Add*. This creates a "Day: 1" in the *Project Browser's* tree structure including a "Step_1". Notice that "Step_1" is editable once created since in most cases you would want to rename the steps. Create days 10, 20, and 30 accordingly. If the Project Browser is not visible, click on *Project Browser* in the *View* menu. Check the *Output All* checkmark to receive output for every step in the Result Viewer.

FIGURE 2.3-2

19
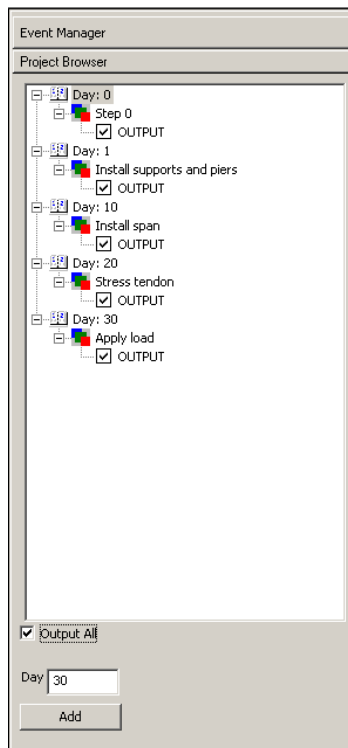
35. Select "Step_1," right-click on it and select *Rename*. Rename this step to "Install supports and piers".

36. In the same way, rename "Step_2" to "Install span", "Step_3" to "Stress tendon" and "Step_4" to "Apply load." The Project Browser will then appear as shown in **Figure 2.3-2**.

37. Select both supports and piers, right-click and select *Install at step*. Use Ctrl key on your keyboard to select multiple structures at a time.

38. Select "Install span" in the *Project Browser*. Select all span segments, then right click on your mouse and select *Install at step*. You will notice that the selected segments are now drawn with a thicker line.

39. Select "Stress tendon" in the *Project Browser*. Double-click on the tendon to open its property box and select the *Stressing* tab. Ensure that in the *Stressing history* "Day 20" is highlighted, then select *Jack Ratio* from the drop-down list on the bottom and set the value to *0.80* (**Fig. 2.3-3**). Confirm by clicking on the green checkmark and close the property box.

40. You can now view the construction sequence in greater detail and adjust with the help of the *Event Manager*. Go to *Event* in the *Build* menu to open the *Event Manager*. In the left list box, you will find the days and steps you created. The list box on the far right is a pre-compiled list of all the events that you specified so far. For example, on day 1 you specified that the supports be installed as well as the piers (note that the piers consist of segments and hence, segments are installed).
The *Solve* command at the beginning of each day moves the solution over the time interval from the period at the end of the previous solution to the day number specified.

41. On day 10, you also want to apply the selfweight of the segment. To add this to the *Event Manager*, you select *Day 10*, select from the middle list box *Apply Selfweight* which is part of the *Loading* tree, and then click on the arrow that points to the right between the middle and the very right list box. Notice in the right list box that the command *Apply Selfweight* is now the last command on day 10. Highlight this command and click the *Move up* button once to move it between *Install Restraint* and *Solve* as shown in **Figure 2.3-4**.

42. If you wish to see the results (e.g., moments, stresses) in the result viewer for a specific day, you can simply add the *Output* command at the end of each day. Therefore, select any day in the very left list box, then select the *Output* command from the middle list box that lists all possible operations and click on the arrow that points to the right between the middle and the very right list box. This will add the *Output* command as the last command for the selected day in the right list box. For this example, ensure that you have one Output command at the end of each day. Your *Event Manager* will then look like shown in **Figure 2.3-4**.
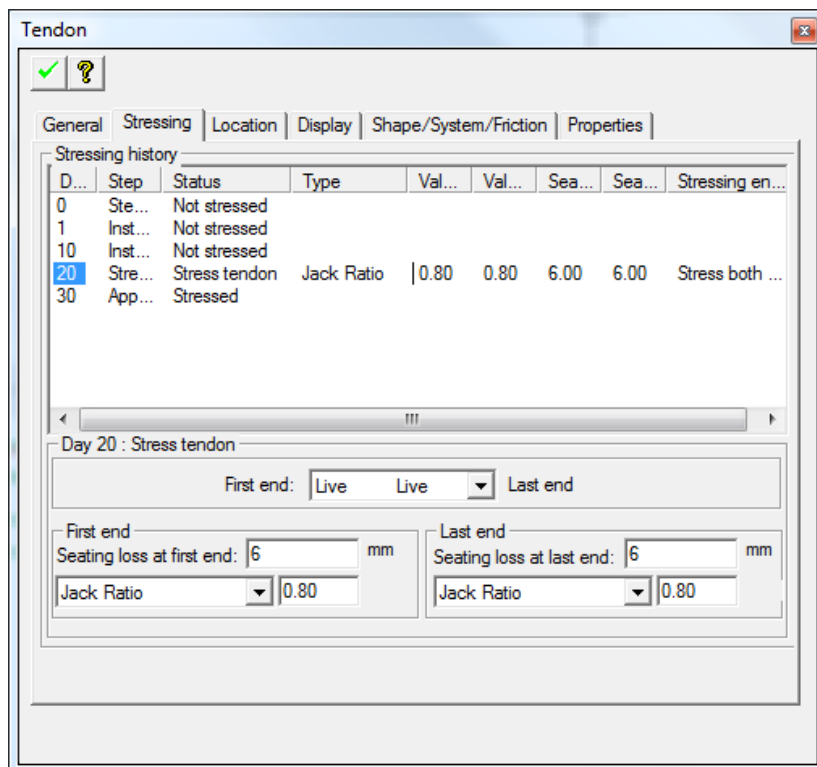
FIGURE 2.3-3
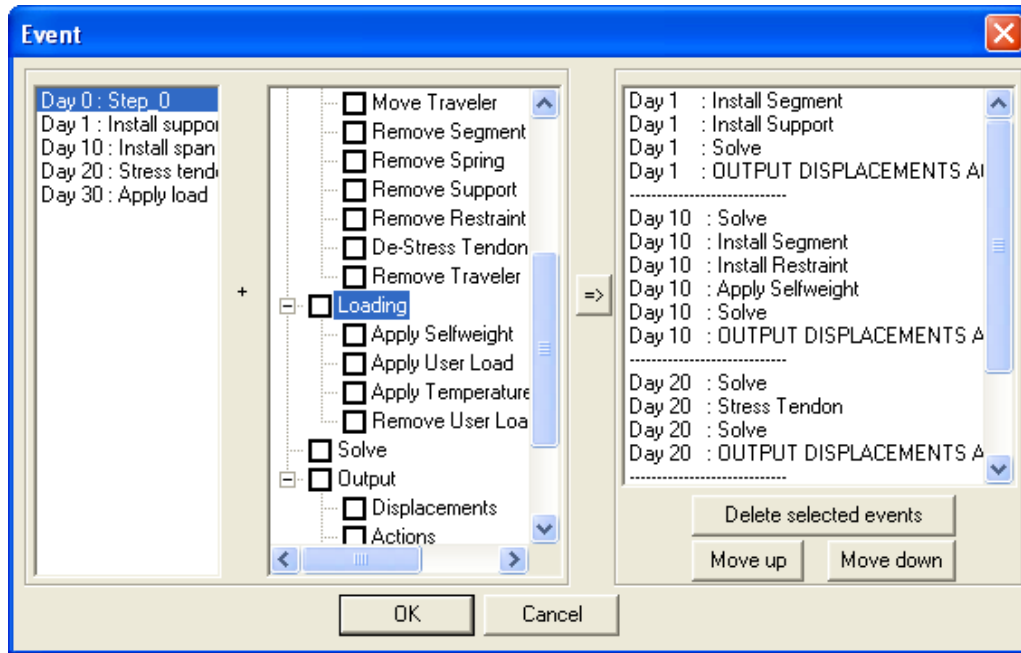
FIGURE 2.3-4
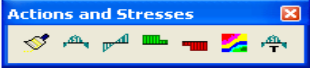
2.4   ANALYZE THE STRUCTURE

43. Select *Generate Input Data* in the *Process* menu.

44. Select *Generate Input Data for Moving Load* in the P*rocess* menu.

45. Select *Execute* in the *Process* menu.

46. Select *Execute Moving Load* in the *Process* menu.

2.5   REVIEW THE RESULTS GRAPHICALLY

47. Click on *View Results* in the *Process* menu to open the result viewer.

2.5.1   Bending Moment.

48. Click on *Bending Moment* 🔼 in the *Action and Stresses*
toolbar. Program will display the bending moment diagram as shown in **Figure 2.5-1**
for Stage 4 which you can select either from the drop-down box or you can walk
through the stages with the help of the vertical blue arrows.
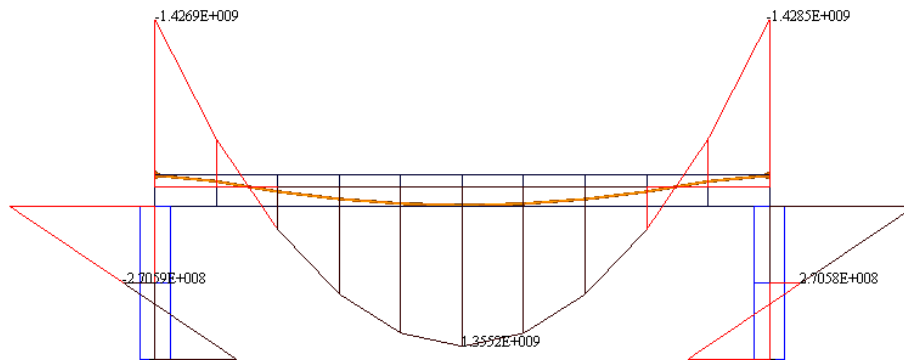
FIGURE 2.5-1

49. You can view the bending moment for a load combination. Go to '*CC*' in the Right-
Side Post Construction Combo Box (**Fig. 2.5-2**) and observe the moment diagram
(**Fig. 2.5-3**).
In the same manner, you can display shear forces, axial forces, pre-stressing forces,
stresses, and torsion moments. Simply select the desired option from the *Action and
Stresses* toolbar.

FIGURE 2.5-2

FIGURE 2.5-3

3   TUTORIAL 2: MODELING OF FOUR SPAN BRIDGE WITH THREE MOVING LOAD PATHS AND BULB-T SECTION

This tutorial requires that you have successfully completed tutorial 1.

The objective of this tutorial is to model a four span bridge and three load paths with different load patterns.
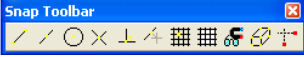
The cross section used is a Bulb-T with 50 inches depth and 36 inches width (**Fig. 3-1**).

FIGURE 3-1

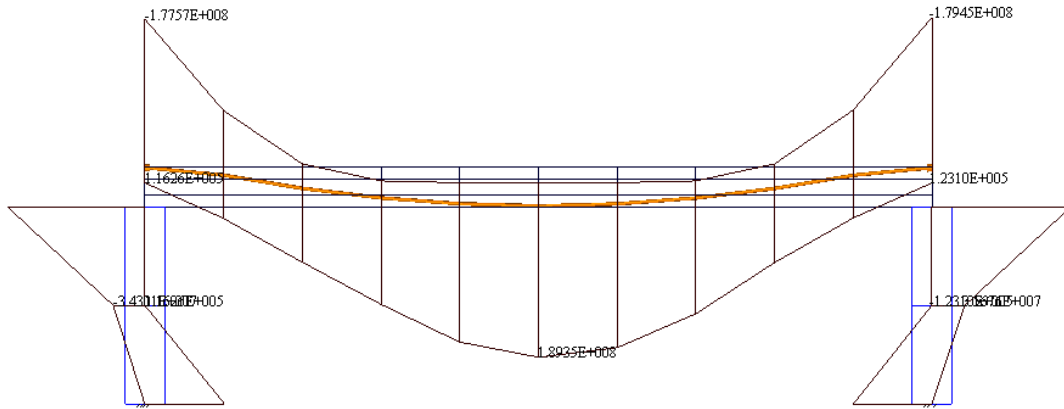| Span Section	Bulb-T cross section of 50 inches depth and 36 inches width.
Pier Section	Rectangular cross section 36 x 24 inches.
Moving Loads	1) HS20_14: HS-20 truck with 14 ft distance between axle 2 and 3
	2) HS20_16: HS-20 truck with 16 ft distance between axle 2 and 3
	3) HS20_18: HS-20 truck with 18 ft distance between axle 2 and 3
	4) HS20_20: HS-20 truck with 20 ft distance between axle 2 and 3
	5) HS20_22: HS-20 truck with 22 ft distance between axle 2 and 3
	6) HS20_24: HS-20 truck with 24 ft distance between axle 2 and 3
	7) HS20_26: HS-20 truck with 26 ft distance between axle 2 and 3
	8) HS20_28: HS-20 truck with 28 ft distance between axle 2 and 3
	9) HS20_30: HS-20 truck with 30 ft distance between axle 2 and 3
	10) MILTRK: Military truck with axle distance of 4 ft and 24 kips of concentrated load
Material Properties	• Concrete: default ACI concrete model • Mild Steel: default material properties
Support	Fixed supports are used at all ends.
Analysis	Execute ABI and Moving Load

3.1 GENERATE THE MODEL

1. Open ABI 2019 and select the American unit system in the ADAPT-ABI Configuration window.

2. On the *Camera and Viewports Toolbar*, click on *Front View*.

3. On the *Snap Toolbar* [toolbar image], click on *Grid Settings* and define 10 ft for both, *Horizontal spacing* and *Vertical spacing*. Then click on *Snap to Grid,* which is located on the same toolbar.

4. Ensure that your *Model Toolbar* [toolbar image] is visible, click on *Create a Reference Line* and draw a horizontal reference line that is 200 ft long.

5. In the *Section Manager*, you define the cross sections needed for this model. Click on the *Define Sections* button on the *Model Toolbar* to open the *Section Manager*.

6. The default shape is *Rectangular*. Enter 36 inches for the width and 24 inches for the height and click the green checkmark to propagate the dimension changes. Double-click on the *Label* field of your *Rectangular* cross section, change the label to 36x24 and hit *Enter* on your keyboard. Keep your *Section Manager* open.

7. Next, you define your *BulbT* cross section. Click on the *Add* button to add one more line in your cross-section list. Select *BulbT-Section* from the drop-down list and change the dimension for *H1* to 50 inches. Confirm the change with the green check mark. Double-click on the *Label* field of your *BulbT* cross section, change the label to *BulbT-50* and hit *Enter* on your keyboard. You have now successfully created a rectangular and a BulbT cross section (**Fig. 3.1-1**). You can close the *Section Manager* by clicking on *OK*.
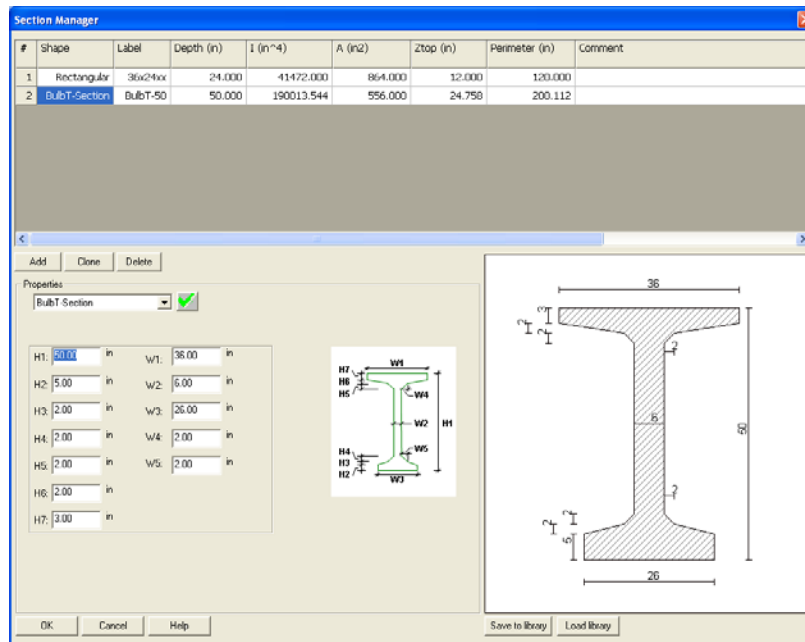

FIGURE 3.1-1

8. To model the structure, you have to be in the *Developed View*. Therefore, click on the *Developed View* button ⬛ on the *Camera and Viewports Toolbar*.

9. First, you will create the four spans. On the *Snap Toolbar*, click on *Snap to Endpoint*. Click on the *Create a Segment* button ⬛ on the *Model Toolbar* and snap to the left endpoint of the reference line and then to the right endpoint of the reference line. Make a right-click and select *Exit* from the menu. You now have a segment that is 200 feet long.

10. Now you need to ensure that spans have the correct cross section. Double-click on the segment and its property box opens (**Fig. 3.1-2**). Go to the *Section* tab and select *BulbT-50* from the *Section I* and the *Section J* drop down list. Confirm with the green check mark and close the property box. The cross section of your four spans is the BulbT-50 that you defined earlier in the *Section Manager*.
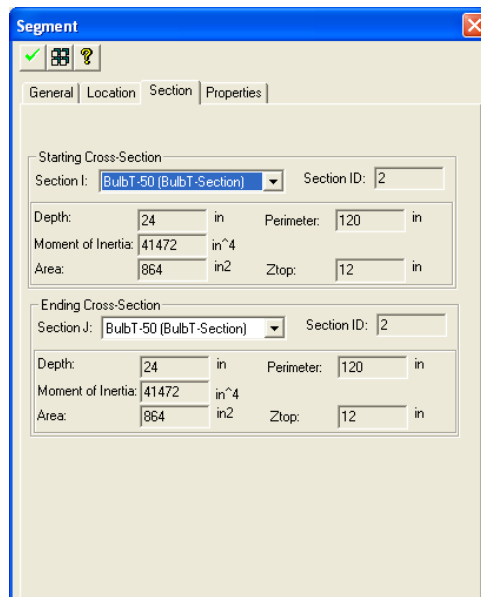
FIGURE 3.1-2

11. Next, you will create the five piers with equal distances of 50 feet. Ensure that none of the snapping options on the *Snap Toolbar* are chosen. Click the *Create Column / Pier* button ⬛ on the *Model Toolbar* and click on the left end of the reference line which is also the left end of your girder. Open the property box of the pier, go to the *Section* tab and select *36x24* for *Section I* and *Section J*. Confirm with the green check mark and close the property box. You have now created the first out of five piers beneath your girder.
Click 50 feet (5 cells on the grid) right next to the left beginning of the girder to place the second pier and create three more piers in the same manner. After all the piers are created, make a right click and select *Exit* from the menu.

12. The piers are 10 feet tall by default. To make them taller, you can click on the *Grid Settings* button on the *Snap Toolbar* and change the *Horizontal* and *Vertical spacing* to 5 feet. Then click *OK* and you will see that the grid cell size changes to 5 x 5 ft. Now drag the lower ends of your piers one by one to the next lower grid vertex.

13. You need to model the point supports on the bottom of the piers. Click on *Create a point support* button on the *Model Toolbar* and click on center point of the lower end of each pier to create a point support at those locations. Once the fifth point support is created, make a right click and select *Exit* from the menu.

14. The point supports are by default modeled as hinged point supports. You want to change them to fixed point supports. Therefore, click on the *Select by Type* button on the *Selection Toolbar*. This opens the *Select by Type* window. Select *Point Support* from the list and click *OK*. This highlights all five point supports that you have modeled. Now select *Modify Item Properties* from the *Modify* menu which opens the *Modify Item Properties* window. Then select the *Point Support* tab, check the checkbox on that tab and select *Fixed* in the *Degree freedom* group frame. Confirm with *OK*. If you open any point support property box you will notice on the *General* tab that the boundary condition is now changed to *Fixed*.

3.2 MESH AND DEFINE CONSTRUCTION STAGES

15. Click *Create Mesh* in the *Process* menu. The program will display the dialog box as shown in **Figure 2.3-1**. Insert a value of 3 ft. for *Suggested cell size*. Then click *OK*.

16. You may want to define some construction stages according to what is described in **Chapter 2.3**. Or you can simply select *Day 0* in the *Project browser*, select the entire structure in the modeling space then right click on your mouse and select *Install at step.* This will install the entire structure theoretically in one day.

3.3 DEFINE TRAVEL PATHS WITH SECTORS

17. In the next step, you will model the first out of three travel paths consisting of four sectors over each span. Go to the *Moving Load* menu and select *Travel Path*. Click on the grid vertex at the left start of your girder where also your reference line starts to define the start point of your travel path. Move along the reference line and click on the grid vertex that is on the reference line above the second pier. Keep moving along the reference line and click on the grid vertices above the third, fourth and fifth pier. After you have clicked above the fifth pier, hit *C* and then *Esc* on your keyboard to finish the operation. You have now successfully created the first travel path that consists of four sectors, one sector over each span.

18. Double click on the travel path to open its property box, put *PATHALL* into the *Label* field, confirm with the green check mark and close the property box.

19. Create a second and third travel path in the same manner.

20. Double click on the second travel path to open its property box and you will see that the four sectors are listed on the *General* tab in the *Skipping option* group frame. For this travel path, you want to skip the even sectors, therefore click on the *Skip even sectors* button. Put *ODD_SPNS* into the *Label* field, confirm with the green check mark and close the property box.

21. For the third travel path, you want to skip the odd sectors. Double click on the third travel path to open its property box and click on the *Skip odd sectors* button. Put *EVN_SPNS* into the *Label* field, confirm with the green check mark and close the property box.

3.4 DEFINE MOVING LOAD CASES

22. Open the *Moving Load Library* through the *Moving Load* menu (**Fig. 3.3-1**) and you will notice that the required *HS20* trucks are already specified in the *AASHTO* load train library. The only truck that you need to define is the *Military* (MILTRK) load train.
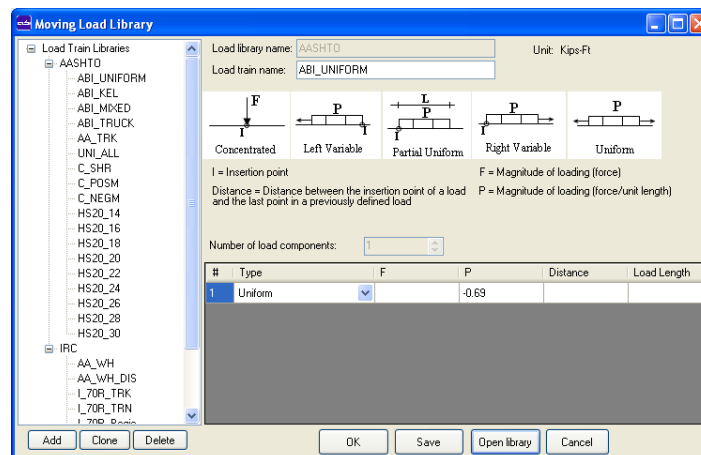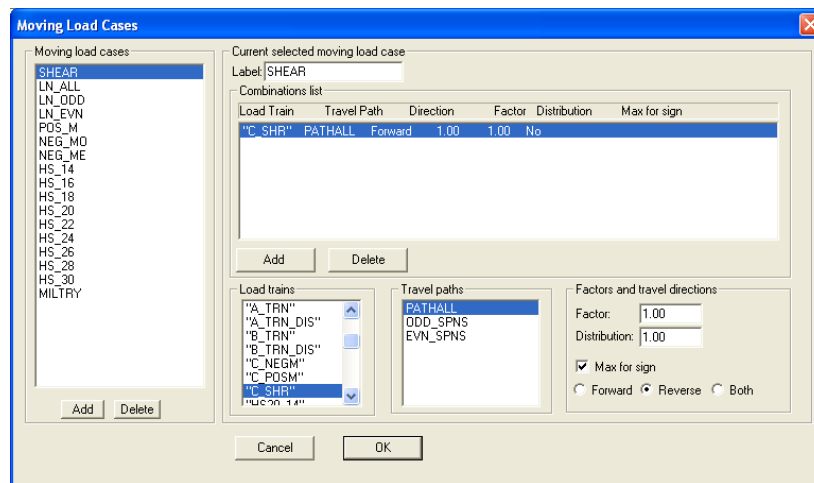

FIGURE 3.3-1

30

23. Click on one of the HS20 trucks and click on the *Add* button on the bottom of the page to add a new load train. In the *Load train name* field, enter *MILTRK* as the name.

24. The load train has two axles which are modeled as two concentrated loads of 24 kips each and a distance of 4 feet between the two axles. Select *Concentrated* from the drop down box under *Type* and type *-24* into the *F* field. Then increase the *Number of load components* to two which will give you two concentrated loads of 24 kips. Since the distance is defined as the space between the last and the current load, you have to put *4* into the *Distance* field of the second concentrated load.

25. Click *OK* to confirm and to close the *Moving Load Library*.

26. You need to associate the individual load trains to load paths. Open the *Load Case* in the *Moving Load* menu to open the *Moving Load Cases* window.
Select *C_SHR* in the *Load trains* group frame and select *PATHALL* in the *Travel paths* group frame. Leave the values for *Factor* and *Distribution* as *1.00*, ensure that the *Max for sign* check box is not selected and select *Forward*. Click on *Add* above the *Load trains* group frame to generate the combination. Click on *Add* below the *Moving load cases* group frame to create a case. This case will be labeled automatically *Case1*, you can go to the *Label* field and rename it to *SHEAR* (**Fig. 3.3-2 shows all moving load cases that have to be created**).

FIGURE 3.3-2

27. To generate the next moving load case, click on Add below the Moving load cases group frame, rename it to *LN_ALL*, select the load train *UNI_ALL* and the travel path *PATHALL*, check the check box for *Max for sign* and click on *ADD* above the *Load trains* group frame.

28. In the same manner, create the following moving load cases settings provided in Table 3.3-1. The factor and distribution factors are always set to 1.00.

Load case	Load train	Travel path	Direction	Max for sign
LN_ODD	UNI_ALL	ODD_SPNS	Forward	Unchecked
LN_EVN	UNI_ALL	EVN_SPNS	Forward	Unchecked
POS_M	C_POSM	PATHALL	Forward	Unchecked
NEG_MO	C_NEGM	ODD_SPNS	Forward	Unchecked
NEG_ME	C_NEGM	EVN_SPNS	Forward	Unchecked
HS_14	HS20_14	PATHALL	Forward	Checked
HS_16	HS20_16	PATHALL	Forward	Checked
HS_18	HS20_18	PATHALL	Forward	Checked
HS_20	HS20_20	PATHALL	Forward	Checked
HS_22	HS20_22	PATHALL	Forward	Checked
HS_24	HS20_24	PATHALL	Forward	Checked
HS_26	HS20_26	PATHALL	Forward	Checked
HS_28	HS20_28	PATHALL	Forward	Checked
HS_30	HS20_30	PATHALL	Forward	Checked
MILTRK	MILTRK	PATHALL	Forward	Checked

29. Confirm with *OK* to close the *Moving Load Cases* window.

3.5   DEFINE LOAD COMBINATIONS AND ENVELOPES

30. Go to the *Loading* menu and click on *Load Combinations / Envelopes* to open the *Load Combinations / Envelopes* window. In the *Create and edit combinations/envelopes* group frame you find a list of all load cases, load combinations and envelopes. Enter *1.00* as the *Factor* for the load cases *NEG_MO* and *NEG_ME* and click on the *Create Load Combination* button. You will notice that a load combination was created in the *Combinations* list. Highlight the name and rename the combination to *C_NEGM* and hit *Enter* on your keyboard.

31. In the same manner, create the following load combinations:

Name	Combination
SHR	1.00 * SHEAR + 1.00 * LN_ENV
LN_NEG	1.00 * C_NEGM + 1.00 * LN_ENV
LN_POS	1.00 * LN_EVN + 1.00 * POS_M

32. You will want to create five envelopes in this window as well. In the top list, highlight *LN_ALL*, *LN_ODD*, and *LN_EVN* using your *Ctrl* button on your keyboard. Click the *Create Envelope* button to create the first envelope. Highlight its name and rename it to *LN_ENV* and hit *Enter* on your keyboard.

33. In the same manner, create the following envelopes:

| Name	Envelope
HS20EN	HS_14, HS_16, HS_18, HS_20, HS_22, HS_24, HS_26, HS_28, HS_30
MAX_M	LANE, HS20EN
MAX_V	SHR, HS20EN
LANE	LN_NEG, LN_POS
MIL_M	MAX_M, MILTRK
MIL_V	MAX_V, MILTRK

34. Click on *OK* to confirm and to close the *Load Combination / Envelope* window.

3.6 ANALYZE THE STRUCTURE

35. Select Generate Input Data in the Process menu.

36. Select Generate Input Data for Moving Load in the Process menu.

37. Select Execute in the Process menu.

38. Select Execute Moving Load in the Process menu.

3.7 REVIEW THE RESULTS GRAPHICALLY

39. Select View Results in the Process menu and activate the post-construction drop down box by clicking on the yellow arrow pointing to the right.

40. Select Max_M and click on the Bending Moment button 🛆 to review your moment envelope (**Fig. 3.6-1**).
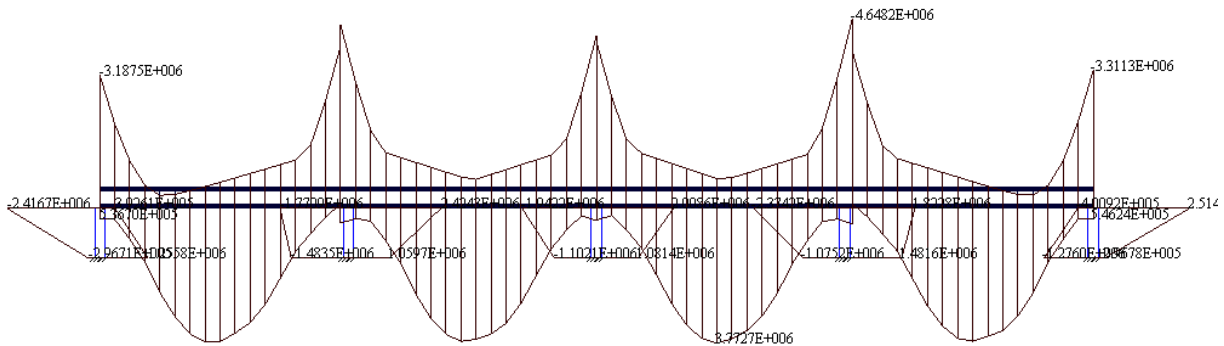
FIGURE 3.6-1

41. Select Max_V and click on the Shear button 🛆 to review your shear envelope (**Fig. 3.6-2**).
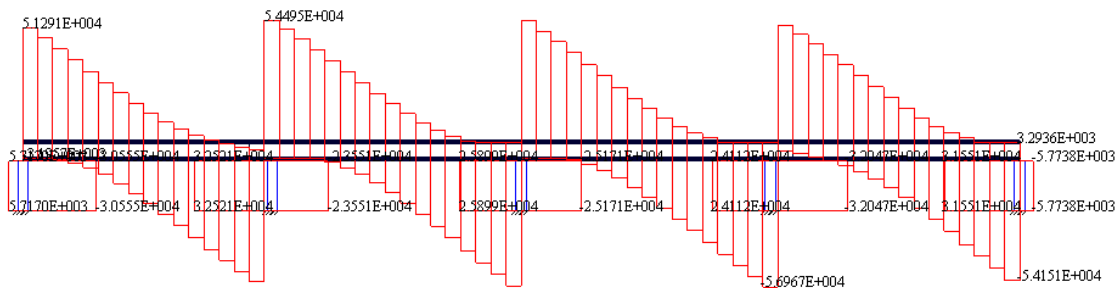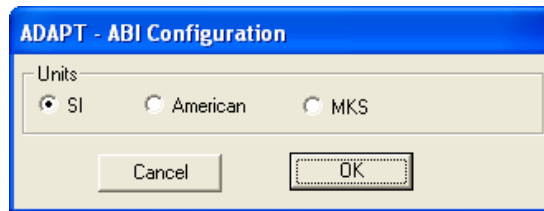

FIGURE 3.6-2

4   TUTORIAL 3: IMPORTING AN ABI v4 INPUT FILE

This tutorial shows you how to import an input file that follows the ADAPT-ABI version 4.xx syntax into ABI 2019.

1. Start ABI 2019 and select any unit in the ADAPT-ABI Configuration window (**Fig. 3-1**).
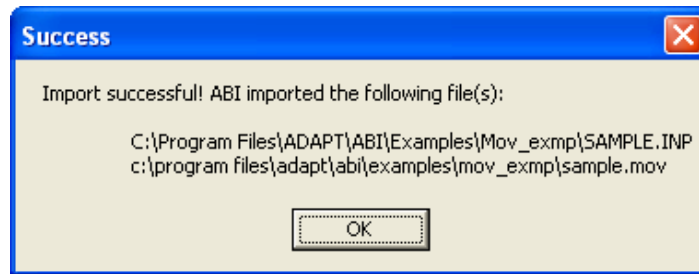

FIGURE 3-1

2. Go to the File menu and select the submenu item Generic ADAPT File in the Import submenu. This allows you to import input files that follow the ADAPT-ABI v4 syntax.

3. In the Open window (**Fig. 3-2**), browse to the location of the input file, select it and hit Open. For this tutorial, we use SAMPLE.INP which is one of the examples that come with ABI v4.5.
    In case a moving load file is related to the input file you import (i.e., a moving load file with the same file name exists in the folder your input file is located), this moving load file is automatically imported. In our case, SAMPLE.INP does have a SAMPLE.MOV related to it and which will get imported as well.
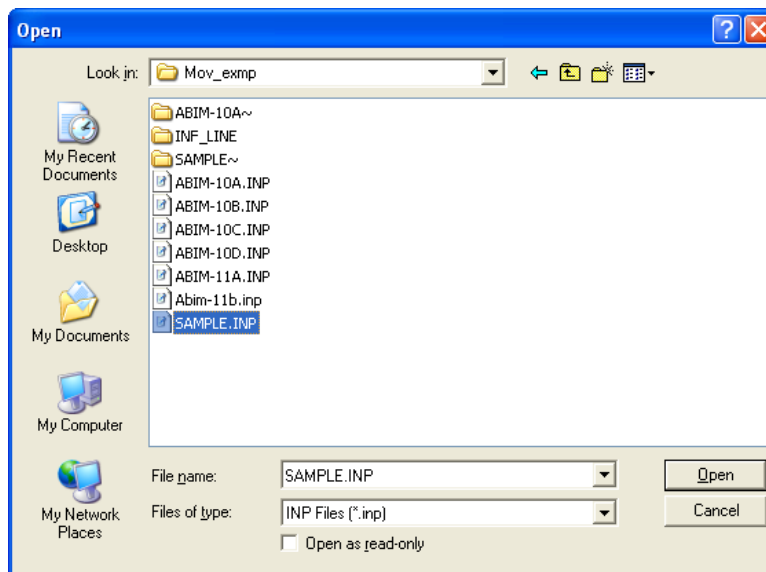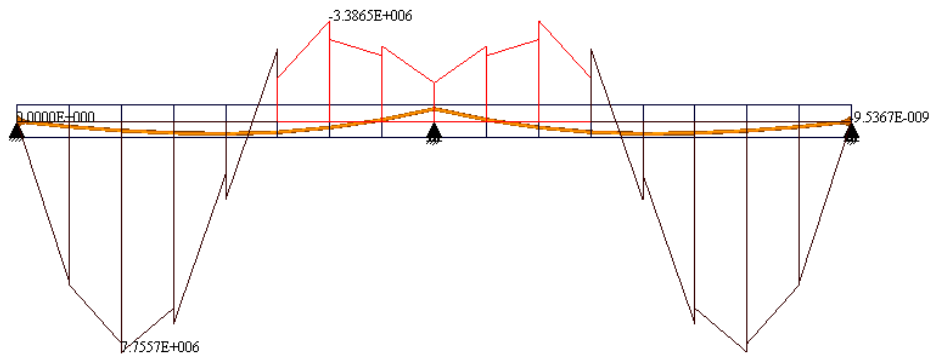

FIGURE 3-2

4. Once the import of your file(s) was successful, a confirmation message appears, which confirms the files and includes the file paths that were imported (**Fig. 3-3**). Click on OK.

Success

Import successful! ABI imported the following file(s):

C:\Program Files\ADAPT\ABI\Examples\Mov_exmp\SAMPLE.INP
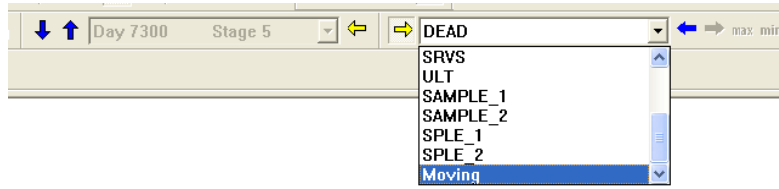c:\program files\adapt\abi\examples\mov_exmp\sample.mov

OK

FIGURE 3-3

5. The ABI interface turns off all modeling-related options that are not needed for a v4 file. You can now validate and run this model or open up a different model.

6. Select Execute in the Process menu.

7. Select Execute Moving Load in the Process menu.

8. Click on View Results in the Process menu to open the result viewer.

9. Click on Bending Moment in the Action and Stresses toolbar. Program will display the bending moment diagram as shown in **Figure 3-4**. for Stage 5, which you can select either from the drop down box or you can walk through with the help of the vertical blue arrows.
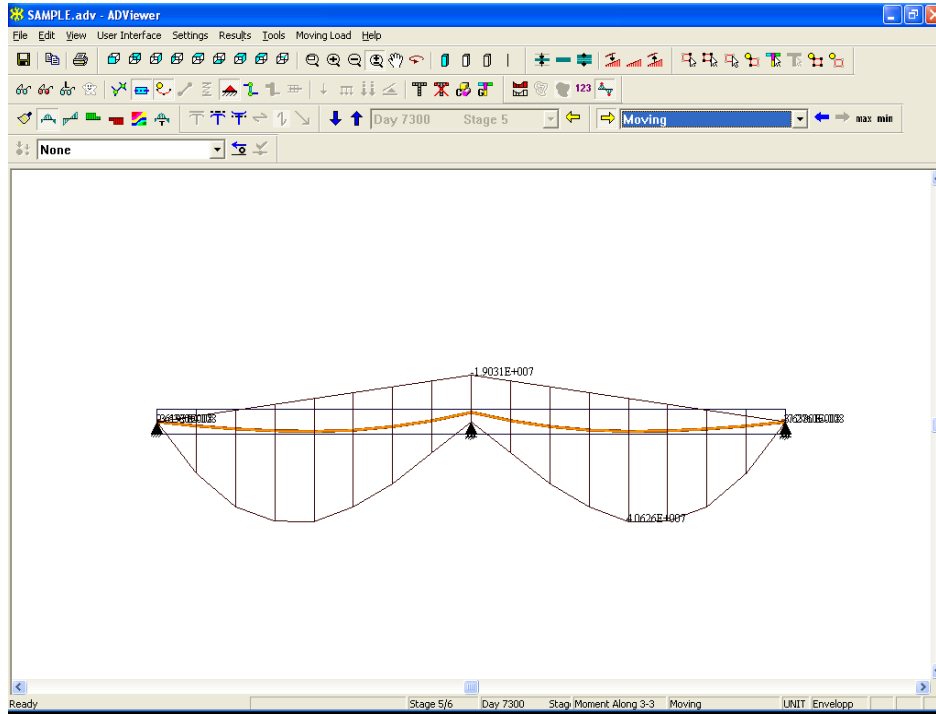If you compare the results with ABI v4.5 you will notice that you will receive exactly the same answer.

FIGURE 3-4

10. You can view the bending moment for a load combination or load case. Go to 'Moving' in the Right-Side Post Construction Combo Box (**Fig. 3.-5**) and observe the moment diagram (**Fig. 3.-6**).

FIGURE 3-5

FIGURE 3-6