## Problem description

The cable-stayed bridge shown in the figures below is to be analyzed for its dead-load static response and then to two loadings corresponding to earthquakes. The bridge has a single continuous span bearing on a central pier and the span is suspended by cables fixed at the top of two towers. The span consists of two longitudinal box girders, floor beams, a concrete deck and transverse box girders at the hanger points.
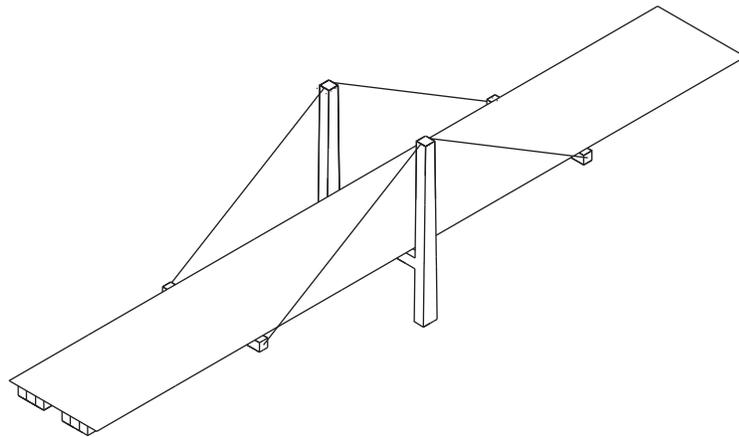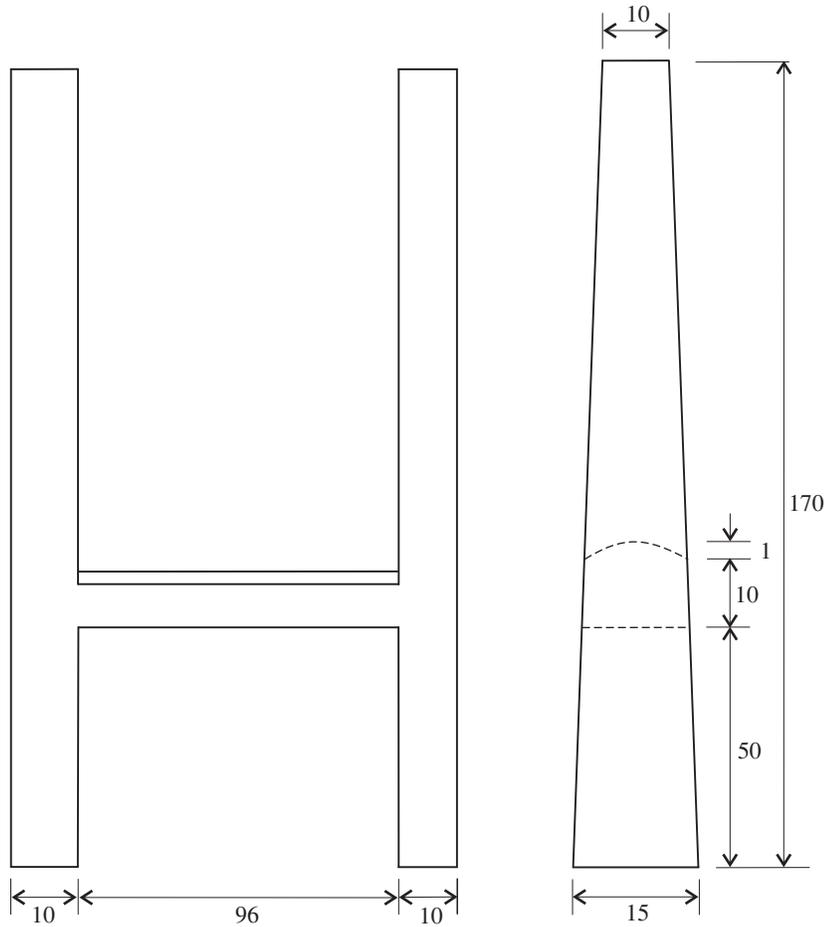

Plan view of deck:

Longitudinal girders not shown
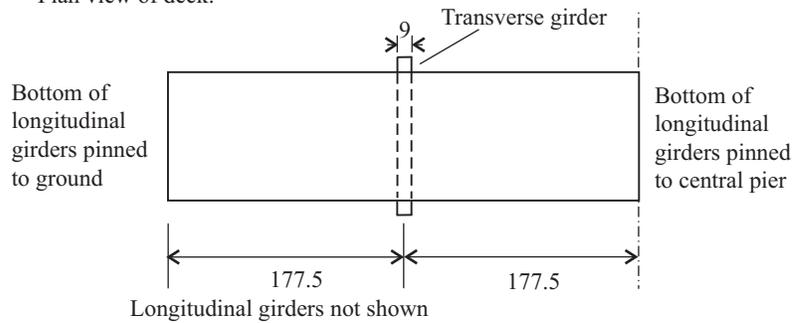
Cross-section of deck:
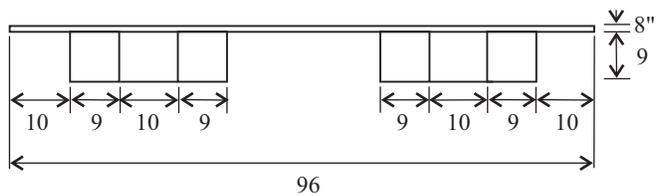
Tower dimensions:

For the purpose of analysis, the towers are considered to be rigid.

The floor beams are I beams, 20 feet apart; each I beam has a 48" × 3/8" web and 12" × 1" flanges. The effect of the floor beams is included by increasing the stiffness of the deck in the transverse direction and by increasing the density of the deck.

The flanges of the box girders are 4" thick and the webs are 2.5" thick. All box girders are capped with diaphragms 2.5" thick and diaphragms are also included at the centers of the longitudinal girders (at the central pier).

The cable is considered to have no stiffness in compression. An initial strain is applied to the cable elements to provide the necessary prestress needed to keep the deck from sagging under the dead load.

The units of the problem are: length (feet), force (kip), time (sec). Therefore the unit of mass is (kip-sec^2/ft) and the unit of density is (kip-sec^2/ft^4).

In the first earthquake analysis, the earthquake load is modeled using a response spectrum and in the second earthquake analysis, the ground displacements are applied directly to the model. The two loads are completely unrelated and describe two separate earthquakes.

The emphasis of this worked example is on the specification of analysis types (static, response spectrum and dynamic) and on the presentation of the results. Therefore, in this analysis, we have set up the model geometry and the finite element definitions in a batch file (prob26_1.in). The organization of the batch file is: definition of geometry points, definition of geometry lines, definition of geometry surfaces, definition of geometry volumes, application of fixities, specification of material properties, definition of finite elements. If you are interested in understanding the commands in the batch file, refer to the AUI Command Reference Manual for ADINA Structures.

We have also set up the response spectrum definitions in a separate batch file (prob26_1.plo). If you are interested in understanding the commands in the batch file, refer to the AUI Command Reference Manual for Display Processing.

The finite element mesh is divided into five element groups:

Group 1: Concrete deck, modeled using 8-node shell elements. The elastic orthotropic material model is used to incorporate the floor beams.

Group 2: Towers and pier, modeled using 27-node solid elements. The Young's modulus of these elements is set artificially high so that the towers do not participate in the motion.

Group 3: Longitudinal box girders and stiffeners, modeled using 8-node shell elements.

Group 4: Transverse box girders, modeled using 8-node shell elements.

Group 5: Cables, modeled using truss elements with specified initial strains and the nonlinear elastic material model. Each cable is modeled with one truss element, to avoid cable vibration modes in the response spectrum analysis.

The model is nonlinear because of the truss elements.

In this problem solution, we will demonstrate the following topics that have not been presented in previous problems:

• Reading a batch file for model definition.
• Reading a batch file for post-processing definitions.
• Calculating mass properties.
• Turning off saving of the element results.

• Restarting from a dead load analysis.

We assume that you have worked through problems 1 to 25, or have equivalent experience with the ADINA System. Therefore we will not describe every user selection or button press.

Before you begin

Please refer to the Icon Locator Tables chapter of the Primer for the locations of all of the AUI icons. Please refer to the Hints chapter of the Primer for useful hints.

This problem cannot be solved with the 900 nodes version of the ADINA System because the model has 1666 nodes.

You need to copy the files `prob26_1.in`, `prob26_2.in`, `prob26_1.plo`, `prob26_xtf.txt`, `prob26_ytf.txt` and `prob26_ztf.txt` from the folder `samples\primer` into a working directory or folder before beginning this analysis.

Invoking the AUI and choosing the finite element program

Invoke the AUI and set the Program Module drop-down list to ADINA Structures.

Dead load analysis

Reading the model geometry and finite element definitions from a batch file

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob26_1.in` and click Open. The AUI processes the commands in the batch file. You can see the log messages from the commands in the Message Window as the AUI processes the commands.

For processing speed, we have not included any graphics commands in the batch file.

After the AUI processes the last batch command, click the Mesh Plot icon to display the geometry and mesh. The graphics window should look something like the figure on the next page.
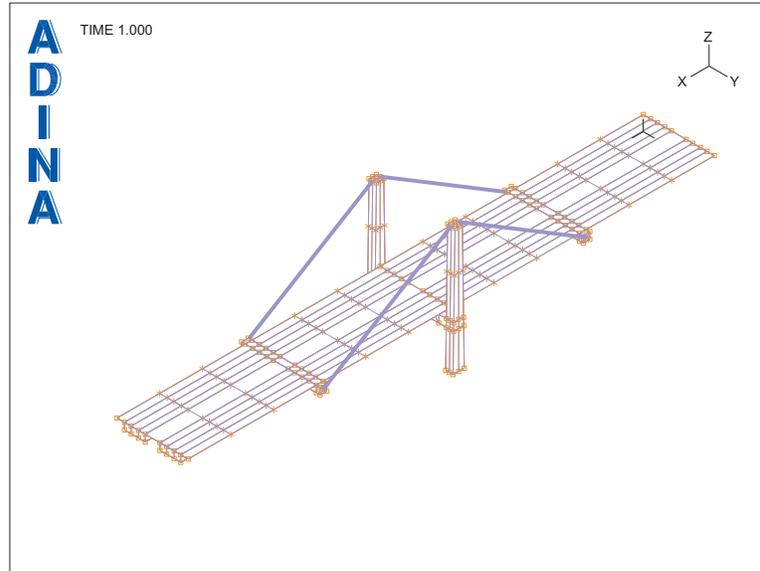
Specifying the control parameters

Let's verify some of the control parameters. The Analysis Type field should be set to Statics. Choose Control→Heading, verify that the problem heading is "Problem 26: Static analysis of cable-stayed bridge" and click Cancel to close the dialog box. Choose Control→ Time Function, verify that the time function is constant with unit value and click Cancel to close the dialog box.

Let's activate calculation of the mass properties. Click the Element Groups icon , then, for each of the five element groups, set the "Calculate Mass Properties" field to Yes and then click Save. (Click the Advanced tab to view the "Calculate Mass Properties" field.) Click OK to close the dialog box.

Let's select the iteration tolerances. Choose Control→Solution Process, click the Tolerances... button, set the Convergence Criteria to Energy and Force, set the Reference Force to 1, set the Reference Moment to 1 and click OK twice to close both dialog boxes.

Specifying the load

The load is the dead weight of the structure. First we need to define a mass-proportional load, then we need to apply it to the model. Click the Apply Load icon , set the Load Type to Mass Proportional and click the Define... button to the right of the Load Number field. In the Define Mass-Proportional Loading dialog box, add load number 1, set the Magnitude to 32.2 and click OK. In the Apply Load dialog box, in the first row of the table, set the Time

Function to 1 and click OK.

Specifying the cable initial strains

We need to specify the initial strain in the cable elements. This strain prestresses the cable elements, providing a cable tension force to prevent the deck from sagging.

We have already determined the magnitude of the required initial strain to be 3.52E-3. Choose Model→Element Properties→Truss. In the first four rows of the table, set the Initial Strain to 3.52E-3 and click OK.

Generating the data file, running ADINA Structures, loading the porthole file

Click the Save icon and save the database to file `prob26`. Click the Data File/ Solution icon , set the file name to `prob26_1a`, make sure that the Run Solution button is checked and click Save. When ADINA Structures is finished, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon and open porthole file `prob26_1a`.

Displaying mass properties

To display the bridge mass properties, choose List→Info→Mass. The total mass is 1.08932E+03 (kip-mass). Because the unit is kip-mass, you have to multiply by 32.2 to compute the weight. Notice that the centroid is at the actual center of the bridge; this is a good check on the model. Click Close to close the dialog box.

Displaying the deformed mesh

We will display the original mesh and magnify the deformations. Click the Show Original Mesh icon and the Scale Displacements icon . The graphics window should look something like the figure on the next page.

Listing the cable stresses and maximum deck displacements

To list the cable stresses, choose List→Value List→Zone, set the Zone Name to EG5, set Variable 1 to (Stress:STRESS-RR) and click Apply. The stress for time 0.0 is 1.01376E+04 and the stress for time 1.0 (which is what we are interested in) is 9.75382E+03 (kip/ft**2). Click Close to close the dialog box.

To determine the maximum displacement, choose List→Extreme Values→Zone, set Variable 1 to (Displacement:Z-DISPLACEMENT) and click Apply. The AUI displays the maximum displacement = -1.02377E-01 (feet). Click Close to close the dialog box.

TIME 1.000   DISP MAG 692.8
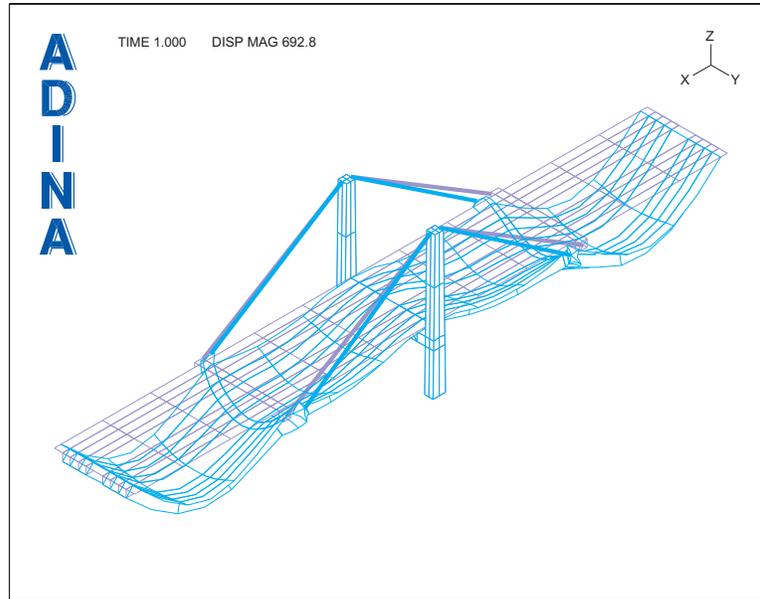
Response spectrum analysis

In this analysis, we subject the bridge model to an earthquake load described by a response spectrum. The bridge model is preloaded by its weight before the earthquake load is applied.

In ADINA Structures, this type of analysis is performed using the restart feature. First the static deformations of the model are determined (we did this in the last section). These deformations are used as initial conditions for the frequency and modal participation factor calculations of the restart ADINA Structures run.

Specifying the control parameters

In order to specify the parameters required for the restart ADINA Structures run, we need to update the model. Set the Program Module drop-down list to ADINA Structures (you can discard all changes and continue). Choose database file `prob26` from the recent file list near the bottom of the Files menu.

To change the title, choose Control→Heading, change the problem heading to "Problem 26: Response spectrum analysis of cable-stayed bridge", then click OK.

To specify a restart analysis, choose Control→Solution Process, set the "Analysis Mode" to "Restart Run" and click OK.

To request initial calculations required for response spectrum analysis, set the Analysis Type to Modal Participation Factors and click the Analysis Options icon . Now we enter the number of mode shapes to calculate and some parameters that control the frequency analysis. Click the Settings... button, set the "Number of Frequencies/Mode Shapes" to 15 and the "Max. Number of Iterations per Eigenpair" to 40. Now click the Settings… button, check the "Perform Sturm Sequence Check" button and click OK twice to close the "Bathe Subspace or Lanczos Iteration Settings" dialog box and the "Frequencies (Modes)" dialog box. In the Modal Participation Factors dialog box, set the Number of Modes to Use to 15, verify that the Type of Excitation Load is set to Ground Motion and click OK.

In order to reduce the size of the porthole file, we will turn off saving of the element stresses. Choose Control→Porthole→Volume, uncheck the Individual Element Results button and click OK.

Generating the data file, running ADINA Structures, loading the porthole file

Click the Save icon to save the database file. Click the Data File/Solution icon , set the file name to `prob26_1b`, make sure that the Run Solution button is checked and click Save.

The AUI opens a window in which you specify the restart file from the first analysis. Enter restart file `prob26_1a` and click Copy.

When ADINA Structures is finished, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon and open porthole file `prob26_1b`.

The graphics window should look something like the figure on the next page.

Viewing the frequency solution

We will obtain a listing of the natural frequencies, modal participation factors and modal masses. Choose List→Info→MPF and examine the first table. The first several frequencies should be 1.33568 (Hz), 1.49623, 2.99607. The second table gives the modal masses for each frequency and direction, the third table gives the percent modal masses, the fourth table gives the accumulated modal masses (the accumulated modal mass for mode n is the sum of the modal masses for modes 1 to n) and the fifth table gives the percent accumulated modal masses. From the third table, we learn that mode 10 accounts for 48.84 % of the total mass for motions in the y direction. From the fifth table, we learn that our model includes 58.48% of the total mass for motions in the y direction, and 65.07% of the total mass for motions in the z direction. Click Close to close the dialog box.
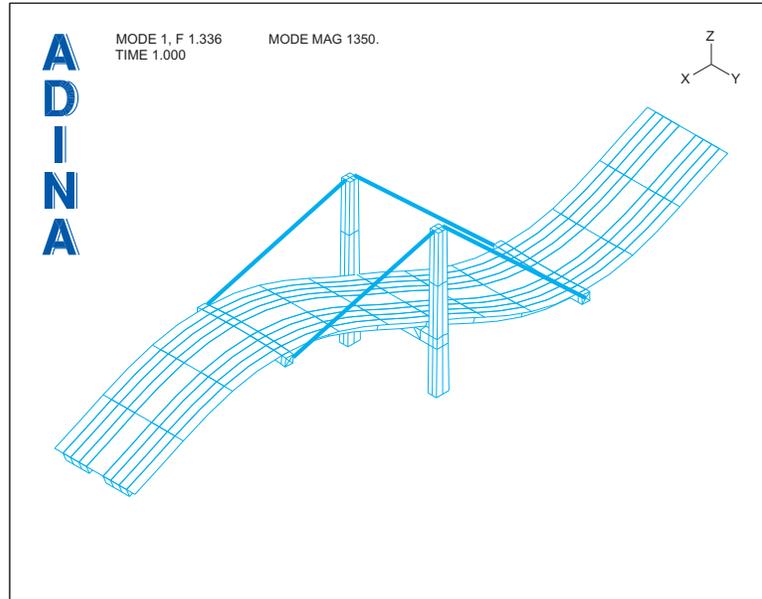
Now we will animate the mode shape. Click the Movie Mode Shape icon ![icon]. The AUI shows the model moving through the mode shape. After the AUI is finished, choose Display→Animate, set the Number of Cycles to 5 and click OK to animate the motion. Click the Refresh icon ![icon] to clear the animation.

You can use the Next Solution icon ▶ , Previous Solution icon ◀ , First Solution icon ⏮ and Last Solution icon ⏭ to view the other modes.
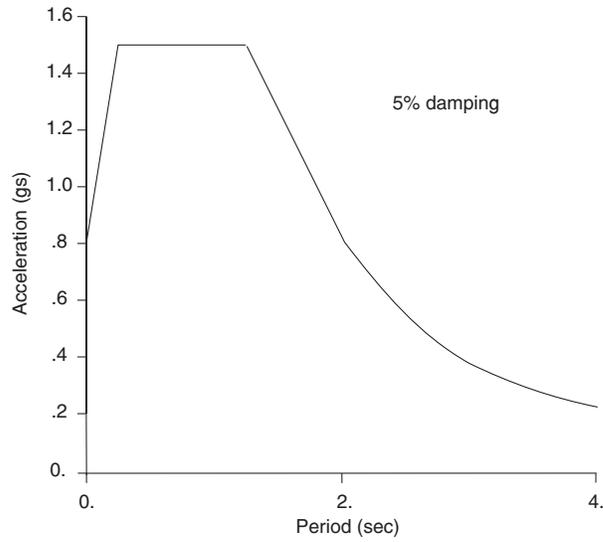
Specifying the response spectrum

We now assume that an earthquake causes ground motions in the y direction of the model.

The response spectrum describing the earthquake load is shown in the figure on the next page.

Now we need to enter the response spectrum into the AUI. We have prepared all of the necessary commands in batch file `prob26_1.plo`. (See problem 12 for an example in which we define a response spectrum using the dialog boxes.)

Choose File→Open Batch, navigate to the working folder or directory, select file `prob26_1` and click Open.

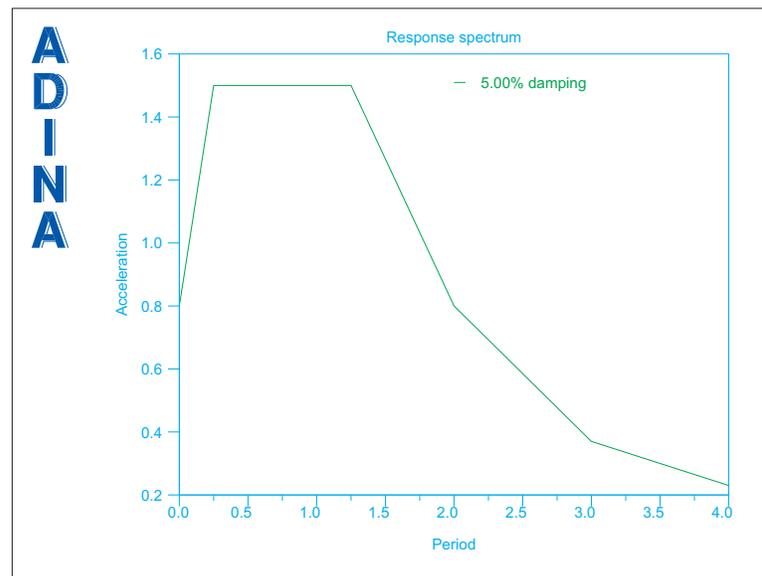Now we graph the response spectrum (which we defined with name ARS in the batch file).

Click the Clear icon ![CLEAR]. Choose Graph→Response Spectrum, select response spectrum ARS and click OK.

The graphics window should look something like this:

Determining the maximum response

Now we use the response spectrum to compute the maximum responses. Choose
List→Extreme Values→Zone and click the Single Response button in the Response Option
box. Notice that response DEFAULT is displayed. Click the … button to the right of the
Response field to view the definition of response DEFAULT. Response DEFAULT is a
response of type response-spectrum (the commands in the batch file defined this response).
Click Cancel to close the Response dialog box.

Since the DEFAULT response corresponds to a response spectrum analysis, the AUI will use
the response spectrum calculations when evaluating the variables. For the variables, enter
(Displacement:Y-DISPLACEMENT), (Velocity:Y-VELOCITY), (Acceleration:
Y-ACCELERATION). Then click Apply. The AUI computes the maximum y-displacement
as 4.27844E-02 (feet), the maximum y-velocity as 1.44463 (ft/sec) and the maximum y-
acceleration as 4.91908E+01 (ft/sec**2). Click Close to close the dialog box.

The first time a response is used, information about the response is printed in the message
window. To view this information, increase the size of the message window, then use the
message window scrollbar. The AUI prints information about the response, then prints a
table giving the mode number, its period, the damping, the modal participation factor and the
spectrum value. You may find this information useful when evaluating the response spectrum
calculations.

Dynamic analysis

In this analysis, we subject the bridge model to an earthquake load described by prescribed
displacements. We assume that the motion of all supports can be represented by the same
prescribed displacements. The bridge model is preloaded by its weight before the earthquake
load is applied.

In ADINA Structures, this type of analysis is performed using the restart feature. First the
static deformations of the model are determined (we did this in one of the previous analyses
above). These deformations are used as initial conditions for dynamic analysis. Unfortunately
we cannot use the results of the previous analysis because in the previous analysis fixities
were used at the pier and ground. We have to replace these fixities by zero prescribed
displacements and rerun the static analysis first, as we show in detail below.

Because of the relatively large size of this model, we will run the model for only 10 time
steps, simply to demonstrate how to perform this type of analysis. Of course, this means that
the results will be relatively uninteresting, but if you have a fast computer, you can run the
model for more steps by modifying the procedures given here.

Preload: Specifying the control parameters

In order to specify the parameters required for the static ADINA Structures run, we need to update the ADINA Structures model. Set the Program Module drop-down list to ADINA Structures (you can discard all changes and continue). Choose database file `prob26` from the recent file list near the bottom of the File menu.

To change the title, choose Control→Heading, change the problem heading to "Problem 26: Static (preload) analysis of cable-stayed bridge", and click OK.

To select the analysis type, set the Analysis Type to Statics. Choose Control→Time Step, verify that one time step with time increment 1.0 is entered, then click Cancel.

Since this is not a restart analysis, choose Control→Solution Process, set the "Analysis Mode" to "New Run" and click OK.

Preload: Specifying the ground displacements

The ground motion is described by the three components of displacement. Each component is described by its own time function, time function 2 for the X component, time function 3 for the Y component and time function 4 for the Z component.

Since we are prescribing the ground displacements directly, we must remove the fixities that we used in the previous analyses. In the Model Tree, click the + next to the 'Fixity' text, then highlight both of the Fixity rows, right-click and choose Delete.

Now we enter the time functions. For the static analysis, we will assign time function values of 0.0 to each of the new time functions. Choose Control→Time Function, add time function 2, enter 0.0, 0.0 for the first row and 1.0, 0.0 for the second row, then click Save. Copy this time function to time function 3 as follows: click the Copy… button and click OK. Then copy time function 3 to time function 4 in a similar way and click OK to close the dialog box.

Now we define displacement loads. Click the Apply Load icon , set the Load Type to Displacement and click the Define... button to the right of the Load Number field. In the Define Displacement dialog box, add displacement number 1, set the X Prescribed Value of Translation to 1, then click Save. Now add displacement number 2, set the Y Prescribed Value of Translation to 1 and click Save. Finally add displacement number 3, set the Z Prescribed Value of Translation to 1, then click OK to close the dialog box. Click OK to close the Apply Load dialog box.

Now we apply these loads to the model. The supports are modeled with geometry lines 1 to 12 (for the attachments of the box girders to ground) and with geometry surfaces 401 to 404 and 501 to 504 (for piers). Therefore we have 20 load applications for each load direction for a total of 61 load applications in the model (remember that we are also applying the dead load). Because the applications of the loads are tedious and error-prone when using the

dialog boxes, instead we have entered the equivalent commands into the batch file
`prob26_2.in`.

Choose File→Open Batch, navigate to the working folder or directory, select the file
`prob26_2.in` and click Open.

After the AUI processes the commands, you can examine the load applications using the
Model Tree.

Preload: Generating the data file, running ADINA Structures, loading the porthole file

Click the Save icon and save the database to file `prob26`. Click the Data File/Solution
icon , set the file name to `prob26_2a`, make sure that the Run Solution button is checked
and click Save. When ADINA Structures is finished, close all open dialog boxes. Set the
Program Module drop-down list to Post-Processing (you can discard all changes), click the
Open icon and open porthole file `prob26_2a`.

The solution should be exactly the same as was computed in the previous static analysis. You
can verify this by following the instructions given above for listing the maximum deck
displacements.

Dynamic analysis: Specifying the control parameters

In order to specify the parameters required for the restart ADINA Structures run, we need to
update the ADINA Structures model. Set the Program Module drop-down list to ADINA
Structures (you can discard all changes and continue). Choose database file `prob26` from the
recent file list near the bottom of the File menu.

To change the title, choose Control→Heading, change the problem to "Problem 26: Dynamic
analysis of cable-stayed bridge", then click OK.

To select the analysis type, set the Analysis Type to Dynamics-Implicit. Click the Analysis
Options icon , verify that the Bathe method is being used, then click Close.

To specify a restart analysis, choose Control→Solution Process, set the "Analysis Mode" to
"Restart Run" and click OK.

To enter the time step size and the number of steps, choose Control→Time Step, set the first
row of the table to 10, 0.01, then click OK.

We will model the damping in the bridge using Rayleigh damping with constants chosen to give 5% damping for modes 1 and 15. Choose Control→Analysis Assumptions→ Rayleigh Damping, set the Default Alpha to 0.7225 , the Default Beta to 1.67E-3, then click OK.

Dynamic analysis: Specifying the ground displacements

In the dynamic analysis, we replace the time functions used for the prescribed displacements in the static analysis with time functions that describe the base motion. The points of each time function curve are stored in files `prob26_xtf.txt`, `prob26_ytf.txt` and `prob26_ztf.txt`. Choose Control→Time Function, select time function 2, click the Clear button, click the Import… button, enter the filename `prob26_xtf.txt` and click OK. The time function curve table displays the time function that you just loaded. Click Save, then alter time functions 3 and 4 using files `prob26_ytf.txt` and `prob26_ztf.txt` in the same way. Click OK to close the dialog box.

Dynamic analysis: Generating the data file, running ADINA Structures, loading the porthole file

Click the Save icon to save the database file. Click the Data File/Solution icon , set the file name to `prob26_2b`, make sure that the Run Solution button is checked and click Save.

The AUI opens a window in which you specify the restart file from the first analysis. Enter restart file `prob26_2a` and click Copy.

When ADINA Structures is finished, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon and open porthole file `prob26_2b`.

Dynamic analysis: Beginning post-processing

To see which solutions have been computed, choose List→Info→Response and examine the listing. You should see that 11 sets of displacements, velocities and accelerations have been loaded. Click Close to close the dialog box.

To check the load application, click the Load Plot icon . The AUI displays the prescribed displacements as arrows. Use the Pick icon and the mouse to rotate the mesh plot so that you are looking at the underside of the bridge. The graphics window should look something like the figure on the next page.
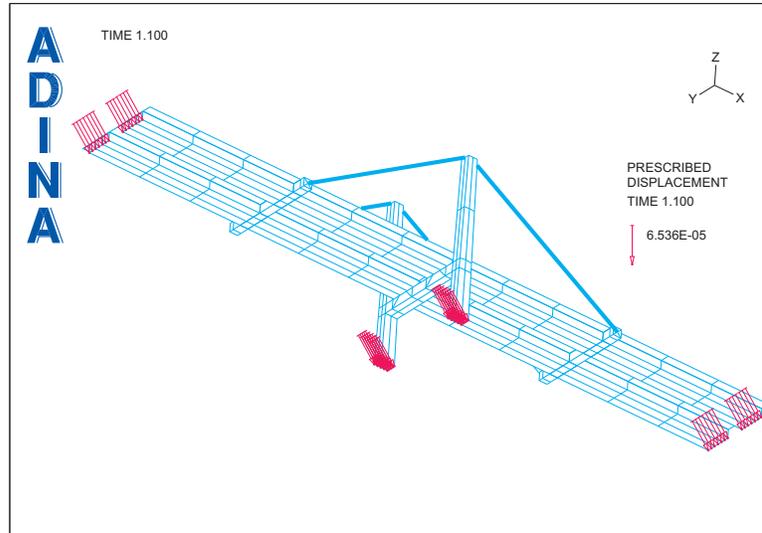
Dynamic analysis: Graphing the solution

We will create a time history graph showing the displacement of one of the points where the cable is attached to the span. This point corresponds to node 1662. Choose Definitions→ Model Point→Node, define point N1662 as node 1662, then click OK. Then click the Clear icon ![CLEAR], choose Graph→Response Curve (Model Point), set the Y Variable to (Displacement:Z-DISPLACEMENT), verify that the model point is N1662, then click OK.

This displacement is, of course, the absolute displacement of the point. In order to obtain a graph of the relative displacement of the point (relative to the ground), we need to create a combination point in which the AUI subtracts the result at the ground from the result at the point. Choose Definitions→Model Point (Combination) →Node, add point N1662R, then, in the first row of the table, enter 1662 and 1.0 for the Node Number and Weight and in the second row of the table enter 826 and -1.0 for the Node Number and Weight. (Node 826 is one of the ground nodes.) Then click OK.

Now choose Graph→Response Curve (Model Point), set the Y Variable to (Displacement: Z-DISPLACEMENT), set the Model Point to N1662R, set the Plot Name to PREVIOUS, then click OK.

Now we change the legends of the graph curves. Choose Graph→Modify, set the Action to "Modify the Curve Depiction", click the P button and highlight the green curve (the one with the circles), then click the … button to the right of the Curve Depiction field. In the Curve Depiction dialog box, uncheck the Display Curve Symbol button, and, in the Legend Attributes box, set the Type to Custom, enter "Absolute displacement" in the first row of the Legend table and click OK. Click Apply to plot the updated curve.

In a similar way, remove the curve symbols and set the curve legend to "Relative displacement to ground" for the red curve (the one with the triangles).

The graphics window should look something like this:
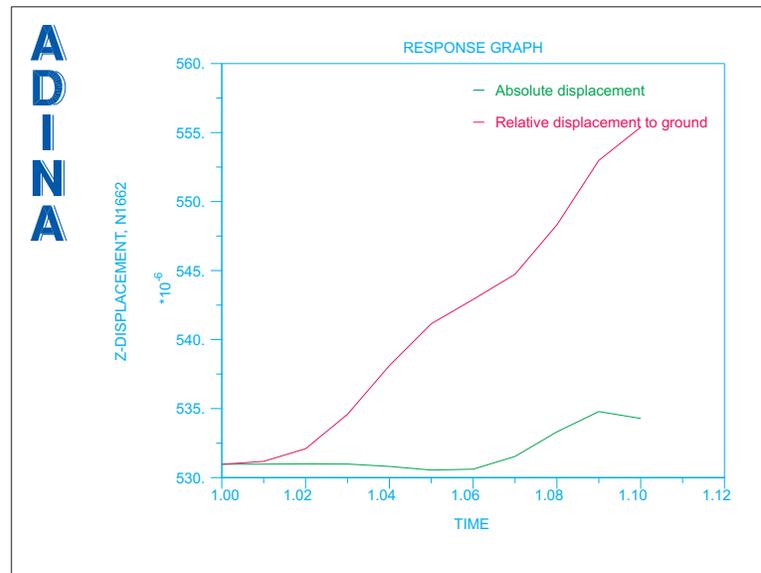
Dynamic analysis: Plotting the deformations due to dynamic effects

To plot the mesh with magnified displacements, click the Clear icon ▣, then the Scale Displacements icon ▣. We don't see the displacements due to the ground motions because they are smaller than the displacements due to the mass-proportional loads.

So we plot the mesh, magnifying just the dynamic displacements. Specifically, we set the reference time for the mesh plot to time 1.0. Click the Modify Mesh Plot icon ▣ and click the Model Depiction… button. In the Model Depiction dialog box, set the "Option for Plotting Original Mesh" to "Use Configuration at Reference Time" and set the "Reference Time for Original Mesh" to 1.0. Also set the "Defined by" field to Max. Displacement and the "Max. Displacement" to 10. Click OK twice to close both dialog boxes.

The graphics window should look something like the figure on the next page. We see that the bridge has translated and that one of the bridge modes has been triggered by the dynamic ground motion.
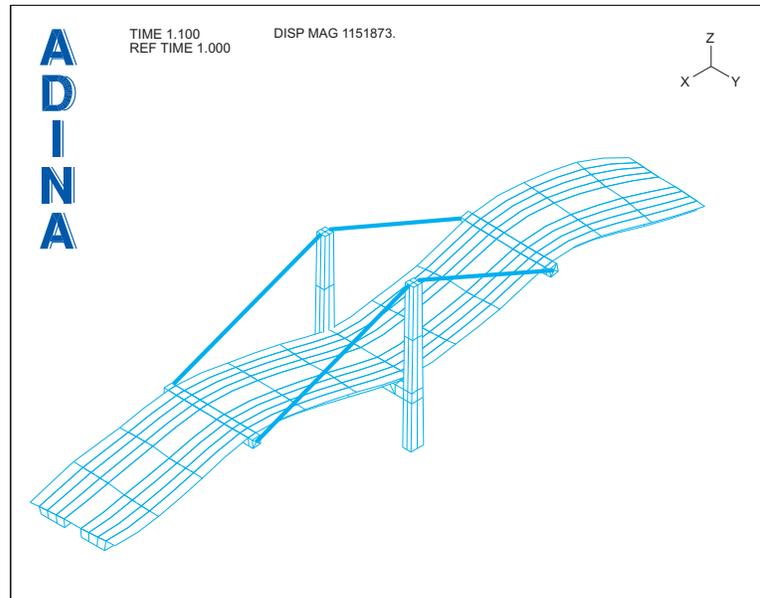
You can use the Previous Solution icon ◀ and Next Solution icon ▶ to examine the solutions for the other solution times. You can also show the solutions as an animation.

Dynamic analysis: Listing the deformations due to dynamic effects

We will list the displacements due to the ground motions. This can be done by subtracting the static displacements from the total displacements. In the AUI, this task is accomplished by defining a response of type response-combination.

First we need to name the response associated with the static deformations. Choose Definitions→Response, add response STATIC, set the Solution Time to 1.0, then click Save. Next we need to name the response associated with the total deformations. Add response TOTAL, set the Solution Time to 'Latest' and click Save.

Now we can define the response combination. Select Response Name DEFAULT_RESPONSE-COMBINATION, then enter TOTAL, 1.0 in the first row of the table and STATIC, -1.0 in the second row of the table, then click OK. Now, whenever this response is used in another dialog box, the AUI will perform the response combination.

For example, to determine the maximum dynamic displacement for this solution time, choose List→Extreme Values→Zone, set the Response Option to Single Response, set the Response to DEFAULT_RESPONSE-COMBINATION, set Variable 1 to (Displacement: X-DISPLACEMENT), Variable 2 to (Displacement:Y-DISPLACEMENT), Variable 3 to (Displacement: Z-DISPLACEMENT) and click Apply. You should obtain the maximum x

displacement = 5.98514E-05 (feet), the maximum y displacement = 2.54000E-05 (feet), the maximum z displacement = -2.43645E-05 (feet).

Exiting the AUI: Choose File→Exit to exit the AUI. You can discard all changes.