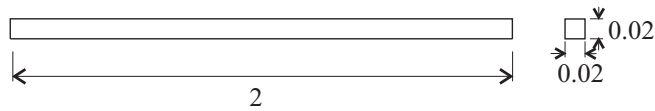## Problem description

In this problem, we apply the component mode synthesis (CMS) technique to a simple beam model.

All lengths in meters.
$E = 2.07 \times 10^{11}$ N/m^2
$\rho = 7800$ kg/m^3

The beam is a free-free beam.

For simplicity, we consider only motions of the beam model in the xy plane. Therefore each beam node has three degrees of freedom: the x translation, the y translation and the z rotation.

Because the beam model that we use has 6 nodes, the total number of degrees of freedom n is 18.

In the CMS method, the degrees of freedom are divided into boundary degrees of freedom and interior degrees of freedom. In this problem, we will assign the degrees of freedom at the two end-nodes of the beam model to be boundary degrees of freedom. Since there are 6 degrees of freedom total at the two end-nodes, the number of static constraint modes k is 6. We will also set the number of fixed interface dynamic vibration modes q to 3. The resulting size of the reduced matrices $\mathbf{K}_r, \mathbf{M}_r$ is $(k+q) \times (k+q) = 9 \times 9$.

As part of the CMS solution, we will also obtain approximations for the first 8 natural frequencies and mode shapes.

For a review of the CMS technique, see Section 11.18 of the ADINA Structures Theory and Modeling guide.

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

• Setting up a component mode synthesis analysis

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 can be solved with the 900 nodes version of the ADINA System.

Much of the input for this problem is stored in the following file: `prob60_1.in`. You need to copy this file from the folder `samples\primer` into a working directory or folder before beginning this analysis.

Invoking the AUI and choosing the finite element program

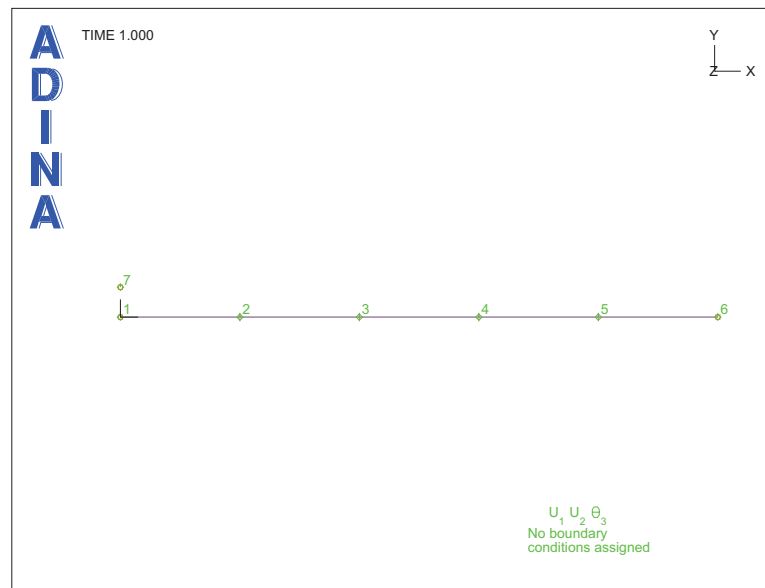Invoke the AUI and set the Program Module drop-down list to ADINA Structures.

Defining the model

Model definition

We have prepared a batch file (`prob60_1.in`) that defines the following items:

> ► Problem heading
> ► Selection of master degrees of freedom, so that the beam vibrates in the xy plane.
> ► Geometry points and lines
> ► Cross-section
> ► Element group 1, which is a Hermitian beam element group. The element group contains 5 equally spaced beam elements.

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob60_1.in` and click Open. The graphics window should look something like this:

Specifying the analysis options for the CMS method

Set the Analysis Type drop-down list to Frequencies/Modes and click the Analysis Options icon ![a] . Set the "Solution Method" to "Component Mode Synthesis (Subspace)", set the "Number of Frequencies/Mode Shapes" to 8, set the "Number of Mode Shapes to be Printed" to 8, check the "Allow Rigid Body Mode" button and set the "# of Fixed Interface Modes to 3". Then click the Boundary DOFs... button. In the Boundary DOFs for Component Mode Synthesis dialog box, fill in the table as follows, then click OK twice to close both dialog boxes.

Node #	Deg. of Freedom
1	X-Translation
1	Y-Translation
1	Z-Rotation
6	X-Translation
6	Y-Translation
6	Z-Rotation

Generating the data file, running ADINA Structures

Click the Save icon ![save] and save the database to file `prob60`. Click the Data File/Solution icon ![icon], set the file name to `prob60`, make sure that the Run Solution button is checked and click Save.

When ADINA Structures is finished, the Message Window should look something like this:

```
Initializing ... Stage 1
Allocating 32.0 MB of memory ...
Initializing ... Stage 2
Initial ADINA memory allocation  -      32.0 mb  (or       4.0 mw)
Starting Solution Process ...
Input phase...
Assemblage of linear matrices.
Calculate and store the load vector
Memory used by the in-core sparse solver. . :      0.2 mb  (or       0.0 mw)
Total memory used by the program. . . . . . :     32.2 mb  (or       4.0 mw)
Factorization completed.
Starting calculation of constraint modes...
Step number =      1   step size =  1.0000000E+00  time =  1.0000000E+00
Step number =      2   step size =  1.0000000E+00  time =  2.0000000E+00
Step number =      3   step size =  1.0000000E+00  time =  3.0000000E+00
Step number =      4   step size =  1.0000000E+00  time =  4.0000000E+00
Step number =      5   step size =  1.0000000E+00  time =  5.0000000E+00
Step number =      6   step size =  1.0000000E+00  time =  6.0000000E+00
Allocated memory without sparse solver . . . . :     32.0 mb (       4.0 mw)
Memory used without sparse solver. . . . . . . :      0.1 mb (       0.0 mw)
Starting calculation of fixed interface vibration modes...
Iteration number =   1     Number of initially converged eigenvalues =      0
Iteration number =   2     Number of initially converged eigenvalues =      0
```

```
Final number of fixed vibration interface modes =    3
Allocated memory without sparse solver . . . . :      32.0 mb (       4.0 mw)
Memory used without sparse solver. . . . . . :        0.1 mb (       0.0 mw)
Starting calculation of eigenvalues for reduced system matrices
Iteration number =    1     Number of initially converged eigenvalues =      0
Iteration number =    1     Number of initially converged eigenvalues =      0
Iteration number =    2     Number of initially converged eigenvalues =      0
Iteration number =    3     Number of initially converged eigenvalues =      4
Final number of converged eigenvalues =     9
```

ADINA Structures has performed several operations:

1) Computed 6 static steps. Each static step corresponds to one static constraint mode (column in Φ_c). There are 6 constraint modes, because there are 6 boundary degrees of freedom.

2) Computed the natural frequencies and mode shapes of the fixed interface dynamic vibration modes (columns in Φ_n). There are 3 fixed interface dynamic vibration modes.

3) Computed the reduced matrices $\mathbf{K}_r$, $\mathbf{M}_r$. These are 9×9 matrices.

4) Computed approximations to the first 8 natural frequencies and mode shapes of the original analysis model.

Examining the reduced matrices

The reduced matrices are written to file `prob60.out4`. Use a text editor to open this file. The first few lines of the file should look something like the printout at the top of the next page. These are the nonzero entries of matrix $\mathbf{K}_r$.

The remaining lines of the file should look something like the printout at the bottom of the next page. These are the nonzero entries of matrix $\mathbf{M}_r$.

File prob60.out4, beginning:

```
     9        9        6        2KXX     1P,5E16.9
     1        1        6
 0.414000000E+08 0.000000000E+00 0.000000000E+00-0.414000000E+08 0.000000000E+00
 0.000000000E+00
     2        1        6
 0.000000000E+00 0.414000000E+04 0.414000000E+04 0.000000000E+00-0.414000000E+04
 0.414000000E+04
     3        1        6
 0.000000000E+00 0.414000000E+04 0.552000000E+04 0.000000000E+00-0.414000000E+04
 0.276000000E+04
     4        1        6
-0.414000000E+08 0.000000000E+00 0.000000000E+00 0.414000000E+08 0.000000000E+00
 0.000000000E+00
     5        1        6
 0.000000000E+00-0.414000000E+04-0.414000000E+04 0.000000000E+00
 0.414000000E+04
```

```
-0.414000000E+04
        6          1          6
 0.000000000E+00 0.414000000E+04 0.276000000E+04 0.000000000E+00-0.414000000E+04
 0.552000000E+04
        7          7          1
 0.277029313E+05
        8          8          1
 0.211892170E+06
        9          9          1
 0.830018452E+06
       10          1          1
 0.100000000E+01
```

File prob60.out4, end:

```
        9          9          6       2MXX       1P,5E16.9
        1          1          9
 0.208000000E+01 0.000000000E+00 0.000000000E+00 0.104000000E+01 0.000000000E+00
 0.000000000E+00-0.943766599E-22-0.848861047E-22-0.420545185E-22
        2          1          9
 0.000000000E+00 0.231777669E+01 0.653724686E+00 0.000000000E+00 0.802223314E+00
-0.386275314E+00 0.103767381E+01 0.636310159E+00 0.452186936E+00
        3          1          9
 0.000000000E+00 0.653724686E+00 0.237742019E+00 0.000000000E+00 0.386275314E+00
-0.178292648E+00 0.446614411E+00 0.162024226E+00 0.825873176E-01
        4          1          9
 0.104000000E+01 0.000000000E+00 0.000000000E+00 0.208000000E+01 0.000000000E+00
 0.000000000E+00 0.943852467E-22 0.848122487E-22 0.424031546E-22
        5          1          9
 0.000000000E+00 0.802223314E+00 0.386275314E+00 0.000000000E+00 0.231777669E+01
-0.653724686E+00 0.103767381E+01-0.636310159E+00 0.452186936E+00
        6          1          9
 0.000000000E+00-0.386275314E+00-0.178292648E+00
0.000000000E+00-0.653724686E+00
 0.237742019E+00-0.446614411E+00 0.162024226E+00-0.825873176E-01
        7          1          7
-0.943766599E-22 0.103767381E+01 0.446614411E+00 0.943852467E-22 0.103767381E+01
-0.446614411E+00 0.100000000E+01
        8          1          8
-0.848861047E-22 0.636310159E+00 0.162024226E+00 0.848122487E-22-0.636310159E+00
 0.162024226E+00 0.000000000E+00 0.100000000E+01
        9          1          9
-0.420545185E-22 0.452186936E+00 0.825873176E-01 0.424031546E-22 0.452186936E+00
-0.825873176E-01 0.000000000E+00 0.000000000E+00 0.100000000E+01
       10          1          1
 0.100000000E+01
```
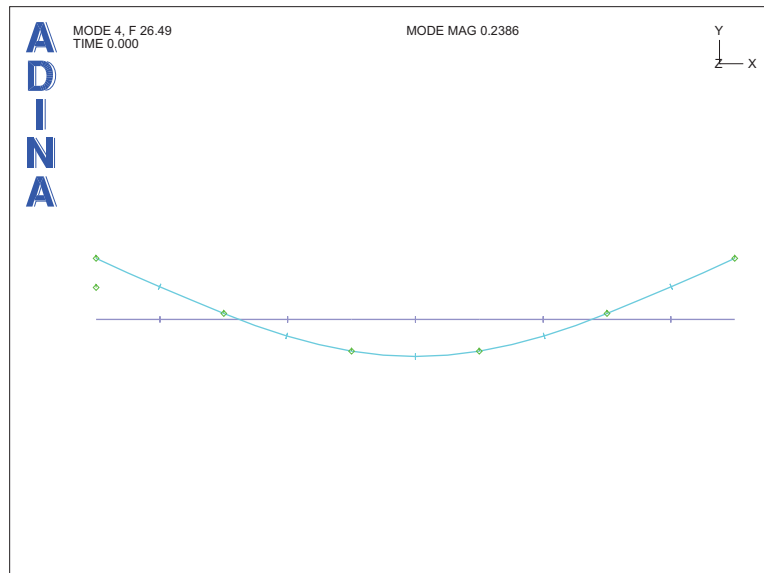
Examining the approximate natural frequencies and mode shapes of the original analysis model

Now 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 `prob60`.

Click the Next Solution icon ▶ three times to display mode 4, then click the Modify Mesh Plot icon 🖼, click the Element Depiction... button, check the Display Beam Cross Section field, click the Advanced tab, set the # Segments for Neutral Axis to 4 and click OK twice to close both dialog boxes. When you click the Node Symbols icon ⋮ and the Show Original Mesh icon ▦, the graphics window should look something like this:

Click the Next Solution ▶ and Previous Solution ◀ icons to examine the other modes. The natural frequencies should be as given in the second column of the following table:

Mode number	Approximate frequency using CMS method (Hz)	Frequency using subspace method (Hz)
1	~0	~0
2	~0	~0
3	~0	~0
4	26.49	26.48
5	73.41	73.18
6	144.6	144.3
7	434.6	238.6
8	694.6	392.4

For comparison, we have computed the first nine natural frequencies and mode shapes of the original beam model (using the subspace method), and the results are in the third column of the above table. It is seen that modes 1-6 are in good agreement, but that modes 7-8 are not in good agreement.

The accuracy of the natural frequencies and mode shapes computed using the CMS method can be improved by using more fixed interface dynamic vibration modes. Recall that in the above analysis, we are using only 3 fixed interface dynamic vibration modes.

Exiting the AUI: Choose File→Exit to exit the AUI (you can discard all changes).

This page intentionally left blank.