Problem description
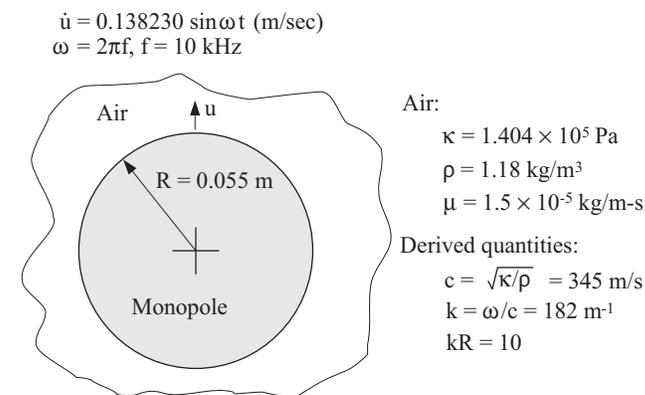
A spherical monopole vibrates sinusoidally, producing spherical waves that propagate into the surrounding air, as shown:

$\dot{u} = 0.138230 \sin \omega t$ (m/sec)
$\omega = 2\pi f,\ f = 10$ kHz

Air $\uparrow$ u

$R = 0.055$ m

Monopole

Air:
$\kappa = 1.404 \times 10^5$ Pa
$\rho = 1.18$ kg/m^3
$\mu = 1.5 \times 10^{-5}$ kg/m-s

Derived quantities:
$c = \sqrt{\kappa/\rho} = 345$ m/s
$k = \omega/c = 182$ m^{-1}
$kR = 10$

In this problem solution, we use ADINA-FSI. ADINA CFD is used to model the air close to the monopole and ADINA Structures is used to model the air further from the monopole. Although this problem could be solved with ADINA Structures alone, we choose to use ADINA-FSI in order to demonstrate the principles of fluid-structure-interaction (FSI) analysis.

An axisymmetric analysis is appropriate here.

We could have chosen to apply the monopole vibration loading in several ways. The way that we choose here is to specify the monopole velocity directly to the ADINA CFD fluid model. Notice that the initial monopole velocity is zero. This is consistent with zero velocity initial conditions throughout the fluid domain.

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

• Performing a fluid-structure-interaction (FSI) analysis
• Using ADINA Structures potential-based fluid elements
• Defining and applying skew systems
• Importing a time function into the time function dialog box
• Performing a Fourier analysis

We assume that you have worked through problems 1 to 14, 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 900 nodes version of the ADINA System does not include ADINA-FSI.

The data for the time function is stored in a separate file `prob15_tf.txt`. You need to copy the file `prob15_tf.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. Choose Edit→Memory Usage and make sure that the ADINA/AUI memory is at least 48 MB.
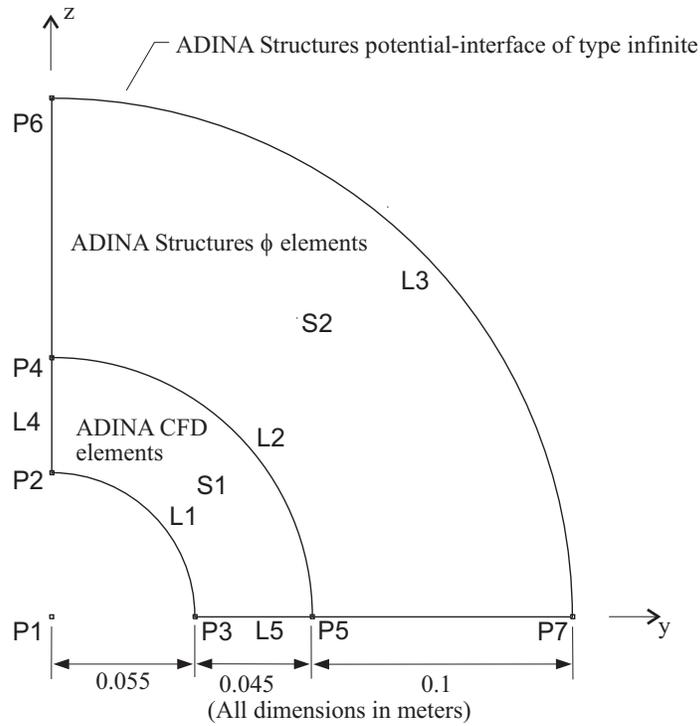
The memory allocation is required to perform the animations at the end of this example.

Defining model geometry

The top figure on the next page shows the key geometry used in defining this model.

Surface S1 will be meshed with ADINA CFD elements and surface S2 will be meshed with ADINA Structures potential-based fluid elements.

Line L3 will be used to define a potential-interface of type infinite. When the AUI generates the data file, this potential-interface will create ADINA Structures potential interface elements of type infinite.

z

ADINA Structures potential-interface of type infinite

P6

ADINA Structures ϕ elements

L3

S2

P4

L4

ADINA CFD
elements

L2

P2

S1

L1

P1

P3 L5 P5

P7

y

0.055

0.045

0.1

(All dimensions in meters)

The next figure shows the fluid-structure interface used in this problem. Notice that the fluid-structure boundary must be defined in both the ADINA Structures input and in the ADINA CFD input.

ADINA Structures: FSI boundary 1

ADINA CFD: Special boundary condition 1

L2
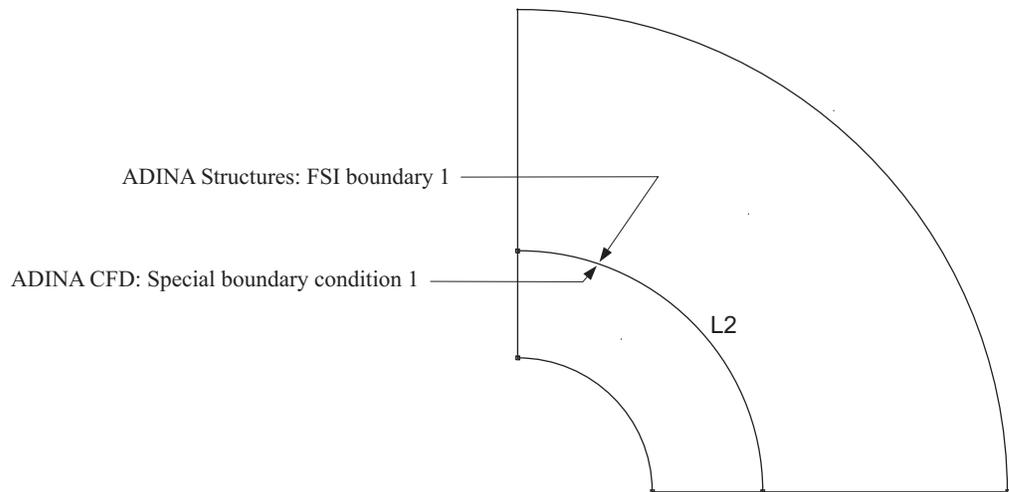
Click the Define Points icon ⊐ , enter the following information into the table (remember to leave the X1 column blank), then click OK.

Point #	X2	X3
1	0	0
2	0	0.055
3	0.055	0
4	0	0.1
5	0.1	0
6	0	0.2
7	0.2	0

Now we define the arc lines. Click the Define Lines icon and add the following lines:

Line Number	Type	Defined by	P1	P2	Center
1	Arc	P1, P2, Center	3	2	1
2	Arc	P1, P2, Center	4	5	1
3	Arc	P1, P2, Center	6	7	1

Next we add the straight lines on which we will assign boundary conditions later:

Line Number	Type	Point 1	Point 2
4	Straight	4	2
5	Straight	5	3

Click OK to close the line dialog box.

Finally we define the surfaces. Click the Define Surfaces icon and enter the following surfaces:

Surface Number	Type	Point 1	Point 2	Point 3	Point 4
1	Vertex	5	4	2	3
2	Vertex	7	6	4	5

Click OK to close the dialog box. The graphics window should look something like the figure on the next page.
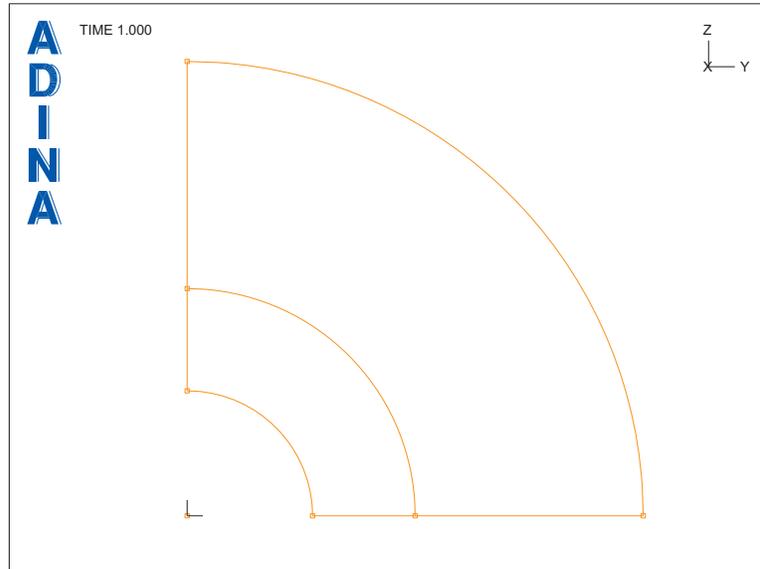
Defining subdivision data

We will use a 10×20 mesh for the ADINA CFD fluid region and a 10×60 mesh for the

ADINA Structures fluid region. Click the Subdivide Surfaces icon , select surface number 1, set the Number of Subdivisions in the u- and v-directions to 10 and 20 respectively, then click Save. Now select surface number 2, set the Number of Subdivisions in the u- and v-directions to 10 and 60 respectively, then click OK.

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

ADINA Structures: Defining model control data

Problem heading: Choose Control→Heading, enter the heading "Problem 15: 10 kHz monopole, ADINA Structures input" and click OK.

Analysis type: Set the Analysis Type drop-down list to Dynamics-Implicit.

FSI analysis: Set the Multiphysics Coupling drop-down list to "with CFD".
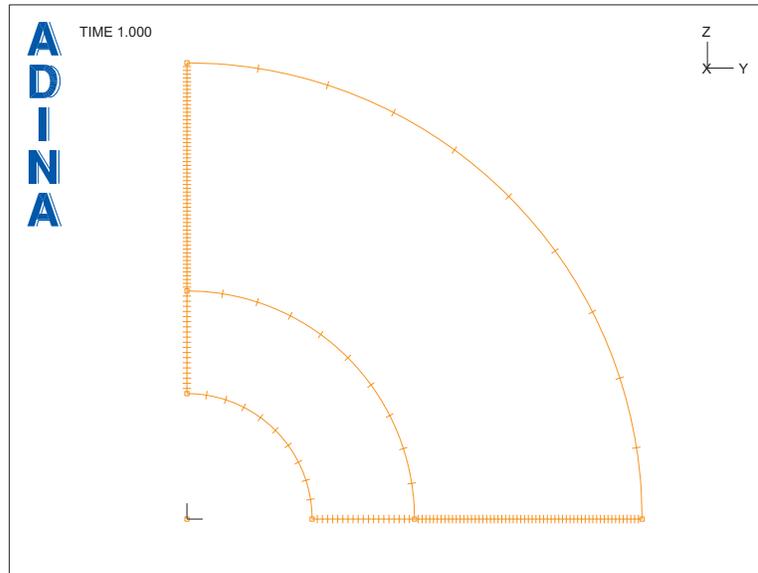
ADINA Structures: Defining the fluid-structure boundaries

In this problem, a fluid-structure boundary is used at the interface between the ADINA Structures and ADINA CFD domains. Choose Model→Boundary Conditions→ FSI Boundary, add Boundary Number 1, enter 2 in the first row of the table and click OK.

ADINA Structures: Defining the material

Click the Manage Materials icon **M** and click the Potential-based Fluid button. In the Define Potential-based Fluid Material dialog box, add material 1, set the Bulk Modulus to 1.404E5, the Density to 1.18 and click OK. Click Close to close the Manage Material Definitions dialog box.

**ADINA Structures: Defining the potential-interface**

Choose Model→Boundary Conditions→Potential Interface, add Potential Interface Number 1, set the Type to "Fluid – Infinite Region", set the Boundary Type to Spherical, set the Radius of Boundary to 0.2, enter 3 in the first row of the table and click OK.

ADINA Structures: Defining the fluid elements

Element group: Click the Element Groups icon ⬚, add element group 1, set the Type to 2-D Fluid, make sure that the Element Sub-Type is Axisymmetric, make sure that the Formulation is set to Linear Potential-Based Element and click OK.

Element generation: Click the Mesh Surfaces icon [icon], set the "Nodes per Element" to 4, enter 2 in the first row of the table and click OK.

The graphics window should look something like this:
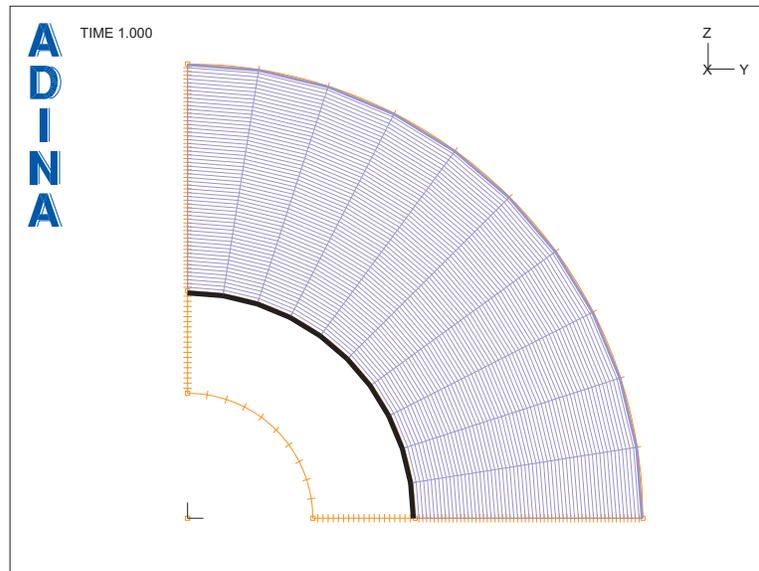
ADINA Structures: Generating the data file

Click the Data File/Solution icon [icon], set the file name to `prob15_a`, uncheck the Run Solution button (we don't want to run ADINA Structures by itself) and click Save.

Notice that the AUI writes "Model completion information for potential-based elements" into the Log Window. This is because the AUI completes the potential-based model when it generates the ADINA Structures data file. In this case the AUI does the following:

  Creates 10 ADINA-CFD interface elements (these are potential-based interface elements between the ADINA Structures and the ADINA CFD models)

  Generates and assigns skew systems to some of the nodes on the boundary between the ADINA Structures and ADINA CFD models (9 nodes).

  Adds fixities to the nodes on the boundary between the ADINA Structures and ADINA CFD models in the direction of zero stiffness. The zero stiffness direction is tangential to the boundary (the AUI assigns the skew systems so that one direction is tangential to the boundary).

The AUI also notes that there are 120 uncovered element sides (sides without any interface). These sides are treated as symmetric boundary conditions. Also there are two nodes that have both a free normal and structural normal. These nodes are on the intersection of the ADINA Structures / ADINA CFD boundary and the symmetric boundaries.

ADINA CFD: Defining model control data

Finite element program: Set the Program Module drop-down list to ADINA CFD.

Problem heading: Choose Control→Heading, enter the heading "Problem 15: 10 kHz monopole, ADINA CFD input" and click OK.

Analysis type: Set the Analysis Type drop-down list to Transient. Then click the Analysis Options icon , set the Integration Method to Composite and click OK.

FSI analysis: Set the Multiphysics Coupling drop-down list to 'with Structures', then click the Coupling Options icon to the right of this list. Set the FSI Solution Coupling to Direct and click OK.

Flow assumptions: Choose Model→Flow Assumptions, set the Flow Dimension to 2D (in YZ Plane), uncheck the "Includes Heat Transfer" button, set the "Flow Type" to Slightly Compressible and click OK.

Time steps: Choose Control→Time Step, edit the first row of the table to be 150, 5.0E-6 and click OK.

ADINA CFD: Defining the boundary conditions

Fluid-structure boundary conditions: Click the Special Boundary Conditions icon , add special boundary condition 1, set the Type to Fluid-Structure Interface and make sure that the Fluid-Structure Boundary # is 1. Enter 2 in the first row of the Line # table. Click Save to save special boundary condition 1.

Wall boundary condition: Add special boundary condition 2 and set the Type to Wall. Set the Slip Condition to Yes. Enter 4 and 5 in the first two rows of the Line # table, then click OK to close the dialog box.

When you click the Boundary Plot icon , the graphics window should look something like the figure on the next page.
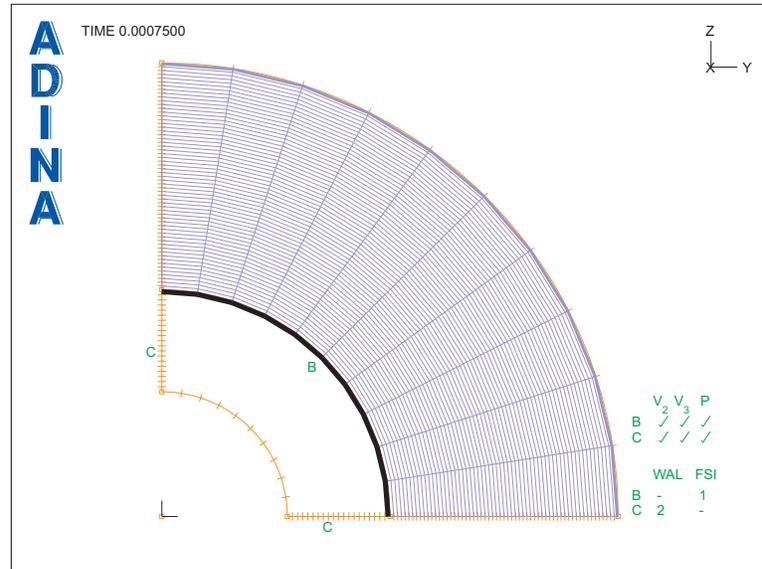
Skew systems: In preparation for prescribing the velocities, we need to define skew systems for the nodes on which we will prescribe velocities. At each node on line 1, the skew system directions are chosen to be normal and tangential to the line.

Choose Model→Skew Systems→Apply and click the Define... button. In the Define Skew Coordinate System dialog box, add skew system 1, set "Defined by" to Normal and click OK. In the Apply Skew Coordinate System dialog box, make sure that the Skew System # is 1 and set the Type to Edge/Line. In the first row of the table, set the Edge/Line # to 1, then click Apply. Notice that the value under the "Normal Direction" column is "Aligned with Axis 'C'" and that the value under the "Tangential Direction" column is "Aligned with Axis 'B'". Click OK to close the dialog box.

Prescribed velocities: Click the Apply Load icon , make sure that the Load Type is set to Velocity and click the Define... button to the right of the Load Number field. In the Define Velocity dialog box, add velocity 1, set the Y Prescribed Value to 0.0, the Z Prescribed Value to 0.138230 and click OK. In the Apply Load dialog box, set the "Apply to" field to Line, then, in the first row of the table, set the Line # to 1. Click OK to close the Apply Load dialog box.

Because the nodes on line 1 have skew systems, the Y direction load is actually applied in the B direction (tangential direction) and the Z direction load is actually applied in the C direction (normal direction).

Time function: Choose Control→Time Function, clear the table, import file `prob15_tf.txt` and click OK. `Prob15_tf.txt` contains a sinusoidal time function with unit amplitude and frequency of 10 kHz.

To plot the load, click the Load Plot icon ![icon]. Unfortunately, at the end of the solution, the load has zero amplitude. So, click the First Solution icon ![icon], then click the Next Solution icon ![icon] several times until the displayed time is 2.500E-05. Click the Load Plot icon ![icon] twice to replot the loads. The graphics window should look something like this:
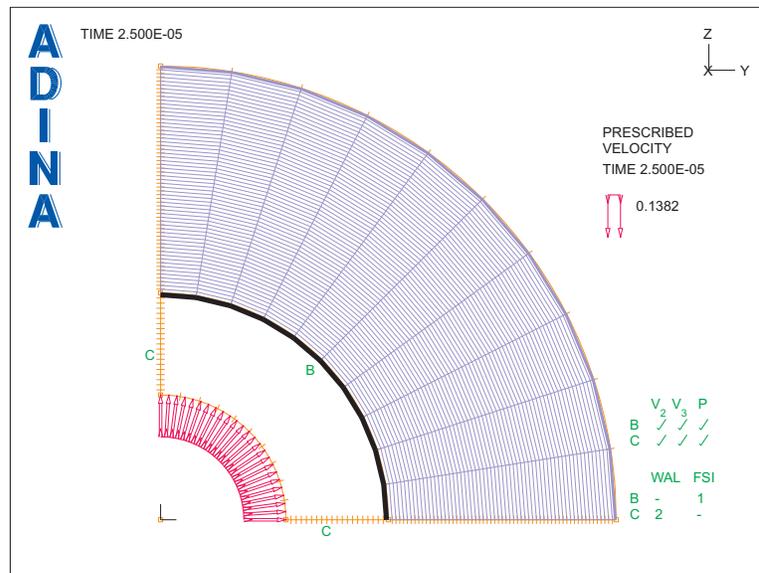

ADINA CFD: Defining the material

Click the Manage Materials icon ![M icon] and click the Laminar button. In the Define Laminar Material dialog box, add material 1, set the Viscosity to 1.5E-5, the Density to 1.18, the Fluid Bulk Modulus to 1.404E5 and click OK. Click Close to close the Manage Material Definitions dialog box.

ADINA CFD: Defining the elements

Element group: Click the Element Groups icon ![icon], add group 1, make sure that the Type is 2-D Fluid and the Sub-Type is Axisymmetric, then click OK.

Element generation: Click the Mesh Surfaces icon ![icon], enter 1 in the first row of the table and click OK. The graphics window should look something like this:
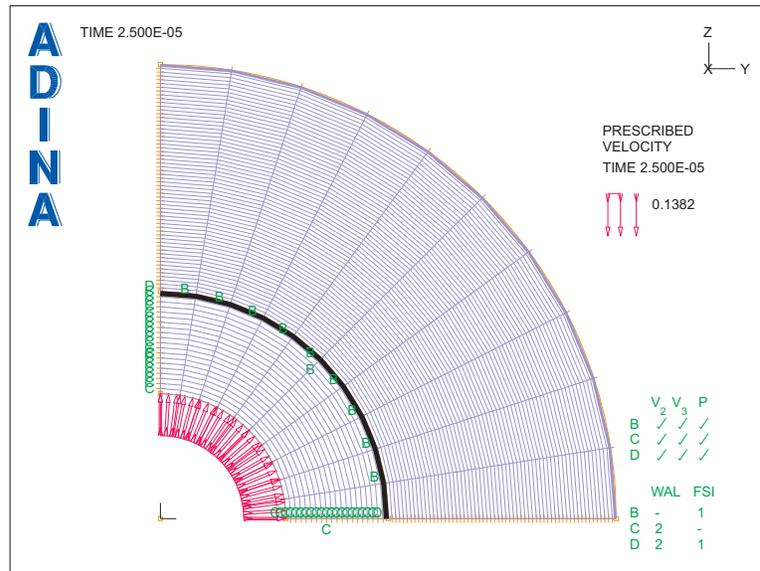
ADINA CFD: Generating the data file

Click the Data File/Solution icon ![icon], enter file name `prob15_f`, uncheck the Run Solution button (we don't want to run ADINA CFD by itself) and click Save.

Running ADINA-FSI

Click the Save icon ![icon] and save the database to file `prob15`. Choose Solution→ Run ADINA-FSI, click the Start button, select file `prob15_f`, then hold down the Ctrl key and select file `prob15_a`. The File name field should display both file names in quotes. Then click Start. When ADINA-FSI finishes, close all open dialog boxes.

Post-processing strategy

ADINA-FSI creates two porthole files, one for ADINA Structures and one for ADINA CFD. We post-process the results for both models simultaneously.

Set the Program Module drop-down list to Post-Processing (you can discard all changes).

Click the Open icon ![icon] and open porthole file `prob15_a`. Then click the Open icon ![icon] and open porthole file `prob15_f`.

We compare the finite element solution with the following analytical solution. The analytical solution for the pressure amplitude as a function of the distance from the center of the monopole is

$$p(r) = \frac{R}{r} p(R), \quad p(R) = \frac{\omega \dot{u} \rho R}{\sqrt{(kR)^2 + 1}}$$

where r is the distance from the center of the monopole. Accordingly we expect that, at a given time, a plot of $r \cdot p(r)$ versus r is sinusoidal with an amplitude of

$R \cdot p(R) = 3.084$ N/m and period of $\lambda = \dfrac{c}{f} = 3.45 \times 10^{-2}$ m .

Displaying the waves

We need to define variables corresponding to $r \cdot p(r)$. First we define a variable for r, then we define two variables for $r \cdot p(r)$, one variable for the ADINA CFD pressures and one variable for the ADINA Structures pressures.

Choose Definitions→Variable→Resultant, add Resultant name R, define it as

```
SQRT(<Y-COORDINATE>**2 + <Z-COORDINATE>**2)
```

and click Save (hint, you can type the resultant in upper, lower or mixed upper and lower case). Now add Resultant name R_P_S, define it as

```
R * FE_PRESSURE
```

and click Save. (We use FE_PRESSURE to access the pressures directly calculated by the ADINA Structures potential-based fluid elements.), Finally add Resultant name R_P_CFD, define it as

```
R * NODAL_PRESSURE
```

and click OK. (We use NODAL_PRESSURE to access the pressures calculated by the ADINA CFD fluid elements.)

Now we present these variables as band plots. Click the Model Outline icon ![icon], then click the Create Band Plot icon ![icon], set the Band Plot Variable to (User Defined: R_P_CFD) and click Apply, then set the Band Plot Variable to (User Defined: R_P_S) and click OK.

The graphics window should look something like the figure on the next page. The plot shows the spherical nature of the waves, with maximum amplitude and wavelength comparable to the solution given above.
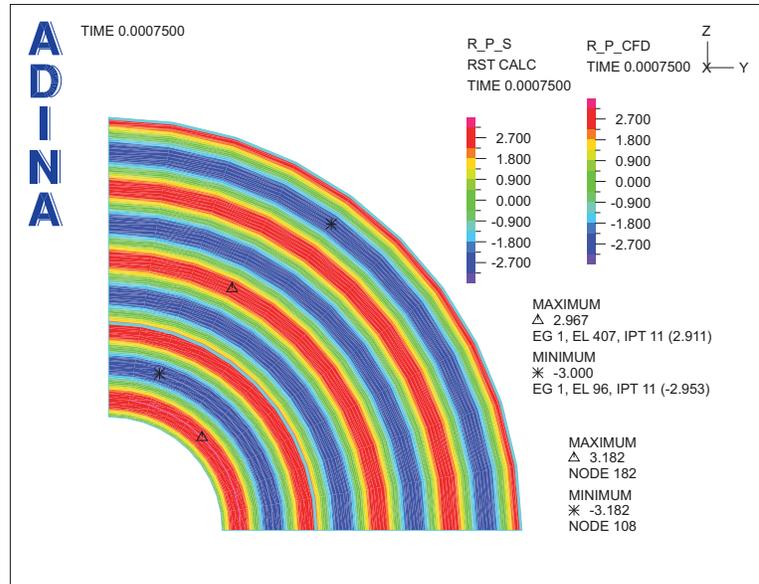
To create an animation, click the Movie Load Step icon [icon]. When the movie shoot is finished, click the Animate icon [icon]. The animation shows the outwards motion of the waves. The absorption of the waves by the infinite potential-based interface elements is also apparent.

Performing a Fourier analysis of the pressure time history

Let=s examine the solution at one point in the model in more detail. We choose to examine the pressure from the ADINA Structures (solid) model at point P5 (at r = 0.1 m). First we need to determine the node number corresponding to this point. Click the Clear icon [icon], then click the Node Symbols icon [icon] to display the mesh with nodal symbols. To show just the ADINA Structures model, expand the Zone entry in the Model Tree, right-click on 1. ADINA and choose Display. The graphics window should look something like the figure on the next page.

Now click the Query icon [icon] and click on the model to determine the node number of the indicated node. (You may need to enlarge the model to make it easier to select the node.) This node number should be 661.
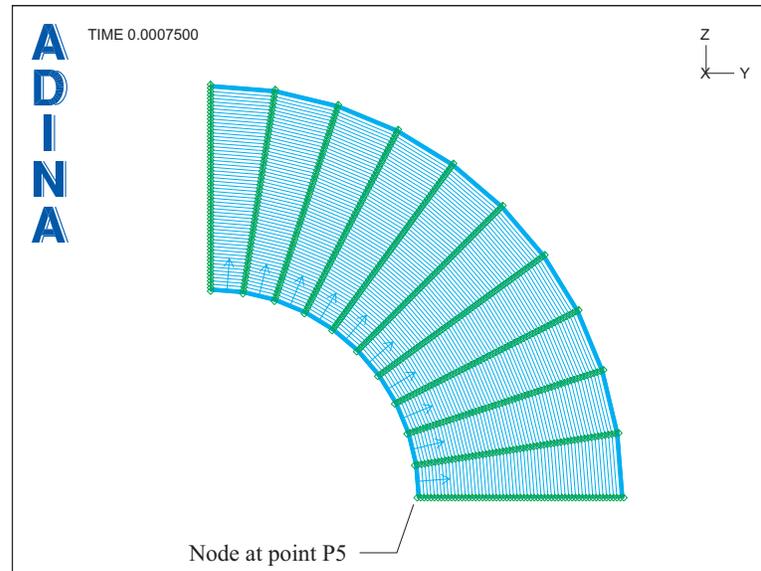
Before we can create the model point, we must change the current finite element program to ADINA Structures. Set the FE Model (not the Program Module!) drop-down list to ADINA Structures.

Create a model point corresponding to node 661 as follows: choose Definitions→ Model Point→Node, add node point name N661, set the Node Number to 661 and click OK.

Let=s plot the pressure at node 661 as a function of time. Click the Clear icon ![CLEAR], then choose Graph→Response Curve (Model Point). In the Define Response Curve (Model Point) dialog box, set the Y Variable to (Stress:FE_PRESSURE), set the Y Smoothing Technique to AVERAGED and click OK. The graphics window should look something like the top figure on the next page.

Now let=s do a Fourier analysis of the steady-state response. From the graph, we are close to steady-state by time 2.5×10^{-4}, so we choose as a time interval 2.5×10^{-4} to 7.5×10^{-4}. Click the Clear icon ![CLEAR], then choose Graph→Fourier Analysis. To set the time interval, click the ... button to the right of the Response Range field, make sure that the Response Range name is DEFAULT, set the Start Time field to 2.5E-4 and click OK to close the Define Response Range Depiction dialog box. In the Fourier Analysis dialog box, set the Variable to (Stress:FE_PRESSURE), set the Smoothing Technique to AVERAGED and click OK. The graphics window should look something like the bottom figure on the next page.

The Fourier analysis is consistent with the nearly sinusoidal time history.
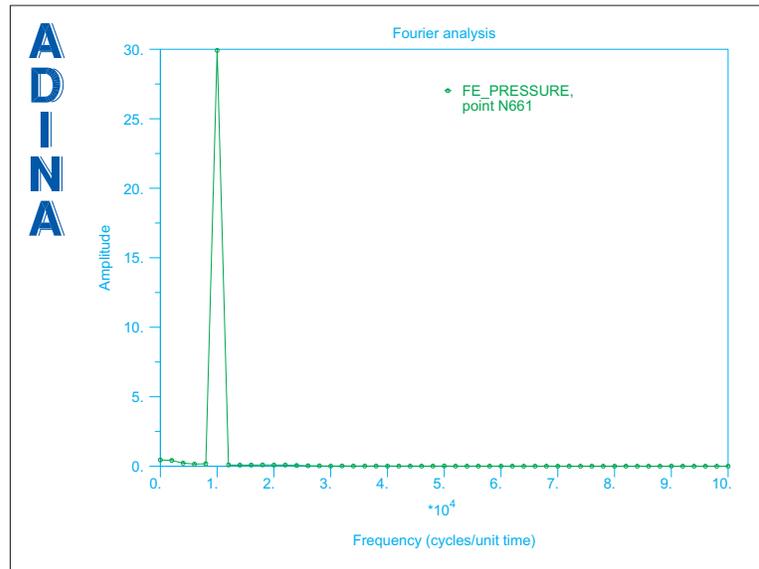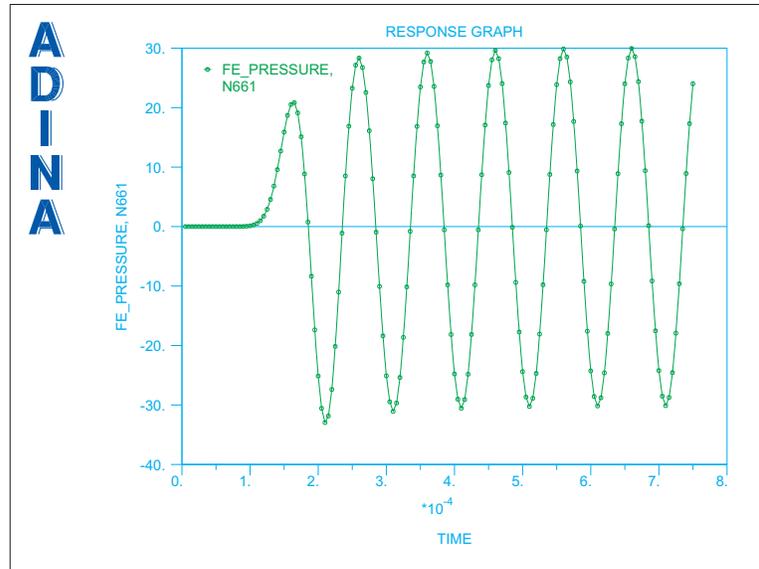
We want to examine the numerical values in the graph. Choose Graph→List. The amplitude at frequency 1.0000E+04 should be 2.98335E+01, which is close to the analytical pressure of 30.84 at $r = 0.1$ m.

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

This page intentionally left blank.