Problem description
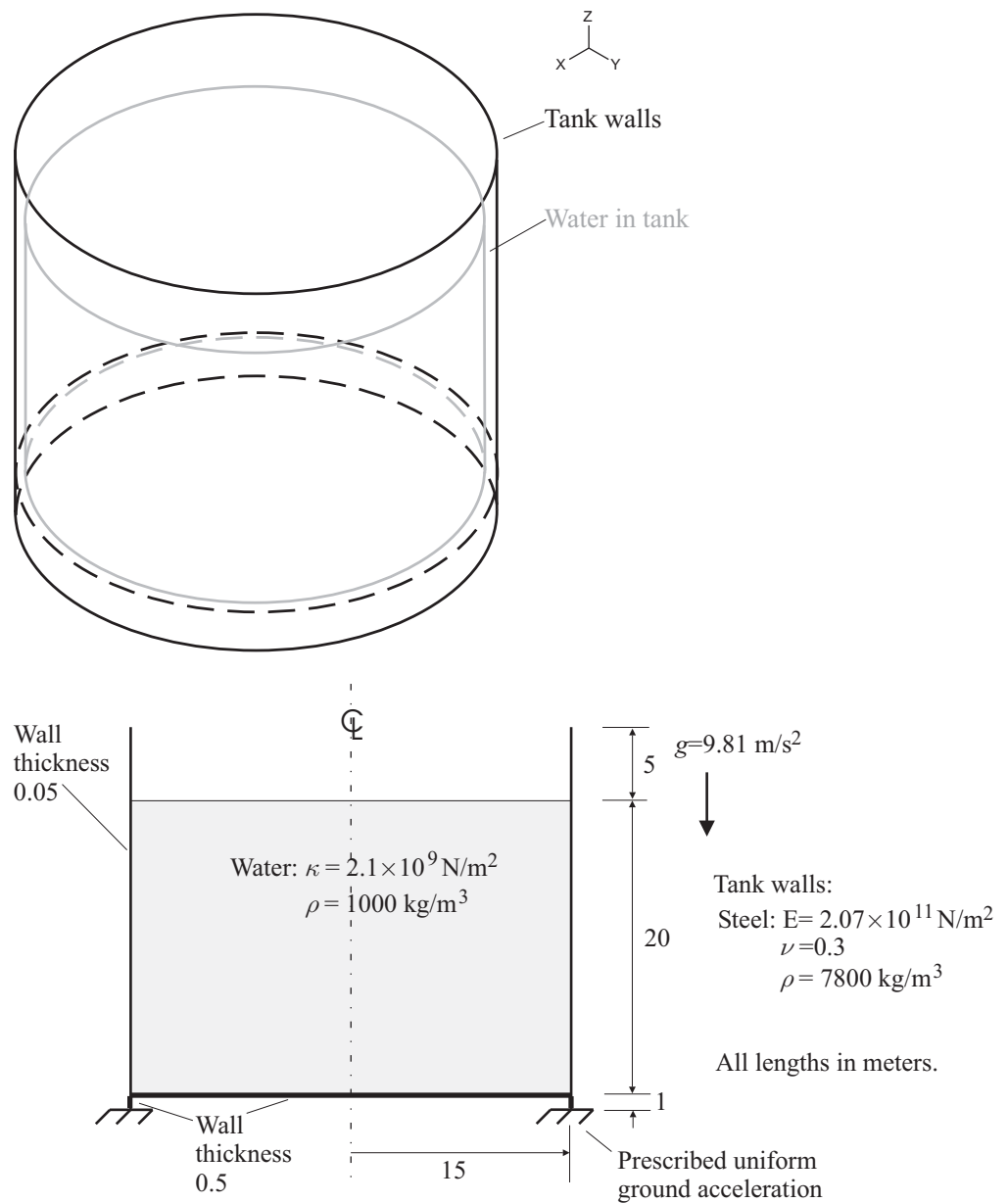
A cylindrical water tank is subjected to gravity loading and ground accelerations, as shown in the figures below:

Shell elements are used to model the tank walls, and potential-based fluid elements are used to model the water in the tank.

Three analyses are performed:

Static analysis: The gravity load is applied and the fluid pressures and tank stresses are calculated.

Frequency analysis: The gravity load is applied, then the first four hundred natural frequencies of the fluid-structure system are calculated. The corresponding modal participation factors for ground motions are obtained.

Dynamic time history analysis: The gravity load is applied, then a ground acceleration acting in the x direction is applied. The ground acceleration is sinusoidal with period equal to the period of the first sloshing mode (as determined in the frequency analysis).

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

- Specifying a potential interface of type free-surface
- Specifying the tolerances for phi model completion
- Performing frequency analysis with potential-based fluid elements
- Specifying mass-proportional loads of type ground acceleration for potential-based fluid elements

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 this model contains more than 900 nodes.

Much of the input for this problem is stored in files `prob63_1.in`, `prob63_tf.txt`. You need to copy files `prob63_1.in`, `prob63_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.

Model definition for the static analysis

### Geometry and mesh definitions

The geometry and mesh definitions are stored in file `prob63_1.in`.

The tank wall is modeled using geometry points 1 to 5, lines 1 to 4, surfaces 1 to 4, material 1 and element group 1 (type shell).

The water is modeled using geometry points 101 to 104, surface 101, volume 101, material 101 and element group 101 (type 3D fluid).

The geometry surfaces of the tank wall and the geometry volume of the water are obtained by revolution.

The geometry before revolution is shown in the figure below.
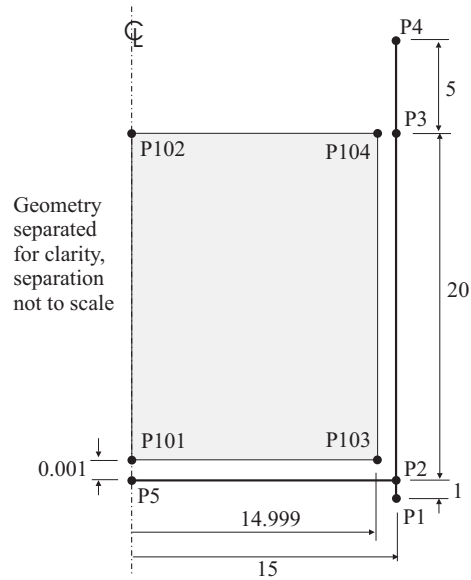
Notice that the geometry used to mesh the tank and water is separated. As a result, the meshing of the shell and fluid meshes is compatible, but separated slightly, with separate nodes for the shell and fluid meshes. Later on, we will specify a tolerance used to connect these meshes during the phi model completion phase (during creation of the .dat file).

Choose File→Open Batch, choose file `prob63_1.in` and click OK. When you click the Mesh Plot icon ▦ and the Color Element Groups icon ▦., the graphics window should look something like this:
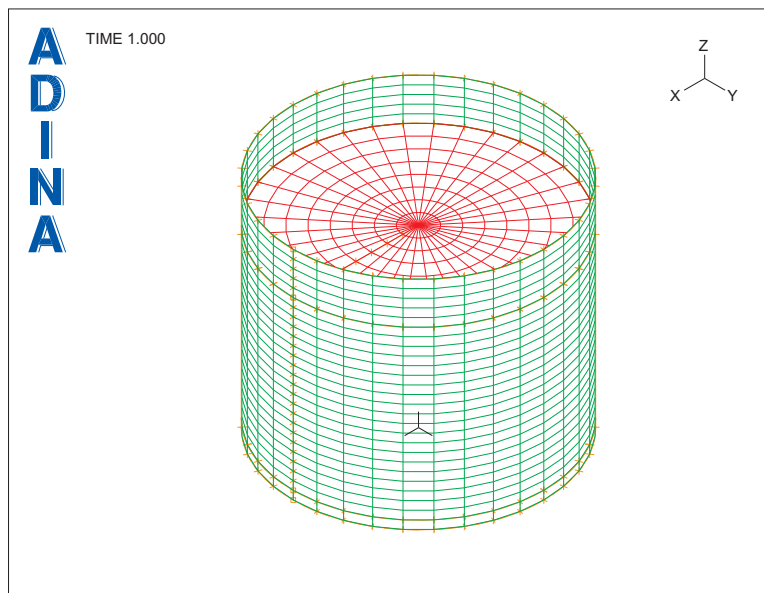

Now click the Wire Frame icon ▱ and the Node Symbols icon ⊤. Use the Pick icon ▸ and the mouse to rotate the plot so that you are viewing the underside of the tank. You will notice that the shell mesh (in green) is compatible with the fluid mesh (in red). Now click the Node Labels icon ⊤ and the Zoom icon ⌕, then enlarge the mesh so that the individual elements and nodes are visible. You will notice that separate nodes are used for the shell and fluid meshes. (You might find it useful to rotate the mesh using the mouse after the mesh is enlarged, in order to visualize the mesh more easily.) Click the Clear icon CLEAR, then the Mesh Plot icon ▦ to return to the original view.

We need to specify tolerances corresponding to the mesh separation. Choose Model→ Phi Model Completion, set the "Tolerance for Nodal Coincidence" to Custom, set the "Custom Tolerances for Nodal Coincidence" to 0.01, 0.01, 0.01 and click OK. (The unit is the length unit of the model; 0.01 m is larger than the separation of 0.001 m used in the geometry definition, and 0.01 m is smaller than the element size.)

Free surface

It is necessary to specify that the top of the fluid is a free surface. Choose Model→ Boundary Conditions→Potential Interface and add Potential Interface Number 1. Set the Type to Free Surface, the "Apply to" field to Surfaces, enter 102 in the first row of the table and click OK. When you click the Redraw icon ![Redraw icon], the graphics window should look something like this:
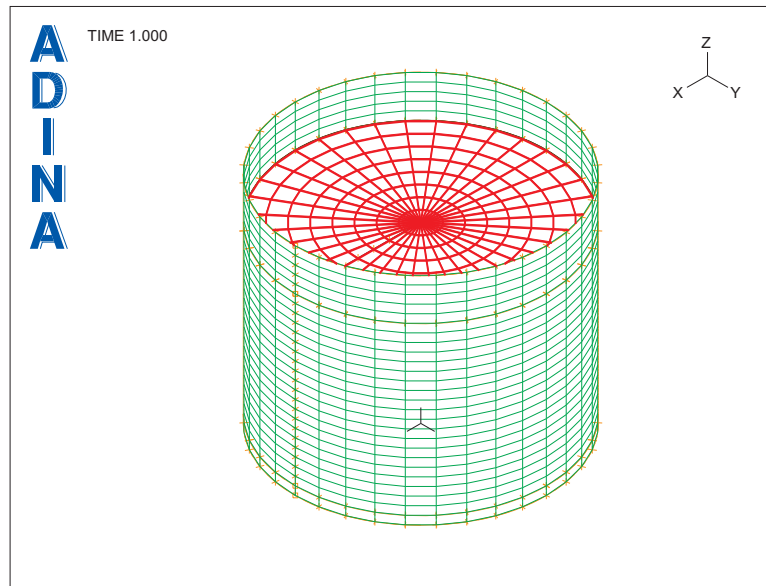
The thick lines on the top of the fluid mesh represent the potential interface.

Gravity load

Click the Apply Load icon ![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 Load dialog box, add load number 1, set the Magnitude to 9.81, make sure that the Direction is set to (0, 0, -1) and click OK. In the Apply Load dialog box, set the Time Function to 1 and click OK.

By default, the analysis is static with constant time function and time step size equal to 1.0.

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

First click the Save icon ▣ and save the database to file `prob63`. Click the Data File/Solution icon ▤, set the file name to `prob63a`, make sure that the Run Solution button is checked and click Save.

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 `prob63a`.

Post-processing for the static analysis

Fluid-structure interface elements

We would like to check the fluid-structure interfaces that were automatically generated as part of the phi model completion process in .dat file generation.

In the Model Tree, right-click on EG101 and choose Display. The graphics window should look something like this:

Use the mouse to rotate the mesh. The entire mesh is covered by thick lines; these lines indicate the fluid-structure interface elements.
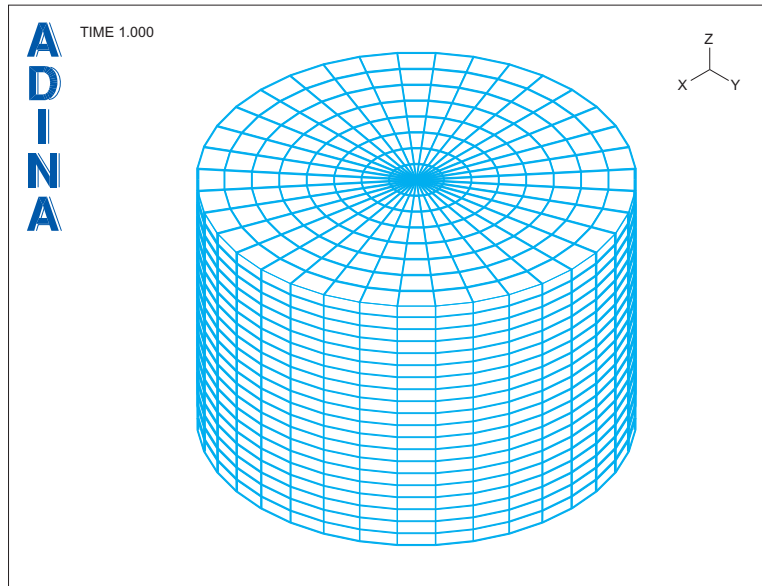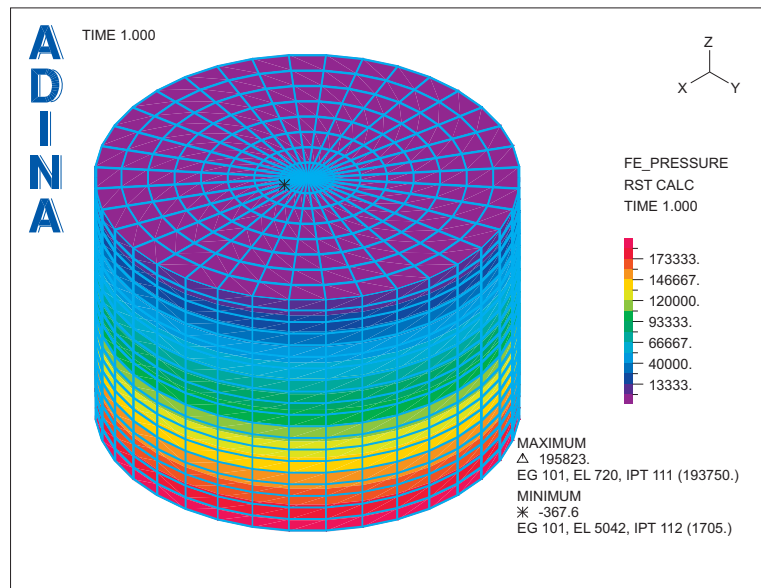
Fluid pressures

We would like to check the static fluid pressures. Click the Create Band Plot icon [icon], set the Band Plot Variable to (Stress: FE_PRESSURE) and click OK. The graphics window should look something like this:

The fluid pressure is nearly zero at the free surface, varies linearly along the side of the tank and is maximum at the bottom. The value of 195823 (Pa) at the bottom of the tank closely matches the expected value $p = \rho gh = 196200$ Pa . (In this expression, ρ is, of course, the fluid density and h is the depth of the water in the tank.)

Tank displacements and stresses

We would like to view the tank displacements and stresses. Click the Clear icon [icon], then, in the Model Tree, right-click on EG1 and choose Display. Click the Scale Displacements icon [icon], the Quick Vector Plot icon [icon] and use the mouse to rotate the mesh, until the graphics window looks something like the figure on the next page.
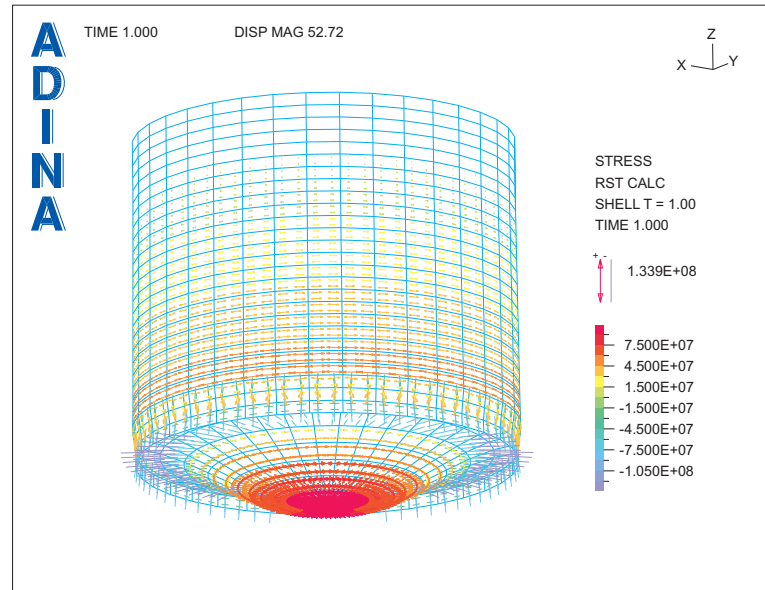
Above the bottom of the tank, the tank walls are primarily carrying hoop stresses. An estimate for the hoop stress is $\sigma_{hoop} = (\rho g h)\dfrac{r}{t}$, where r is the tank radius and t is the tank wall thickness. At the base of the tank, this formula gives the value of 5.89E7 Pa. Unfortunately it is difficult to see from the above plot if the ADINA Structures results give this value, since the largest stresses in the tank are at the center of the tank floor.

Let's plot just the elements on the tank wall. Click the Change Zone icon ![icon], click the ... button to the right of the Zone Name field, add zone TANK_WALLS, click the Edit button, enter

ELEMENTS 1 TO 936 OF ELEMENT GROUP 1

in the first row of the table, then click OK to close the dialog box. In the Change Zone of Mesh Plot dialog box, make sure that the Zone Name is set to TANK_WALLS, then click OK.

The plot shows both tensile and compressive principal stresses. To show only the tensile principal stresses, click the Modify Vector Plot icon ![icon], click the Rendering... button, set the "Vector Plot of Principal Values" to Maximum and click OK twice to close both dialog boxes. The graphics window should look something like the figure on the next page.
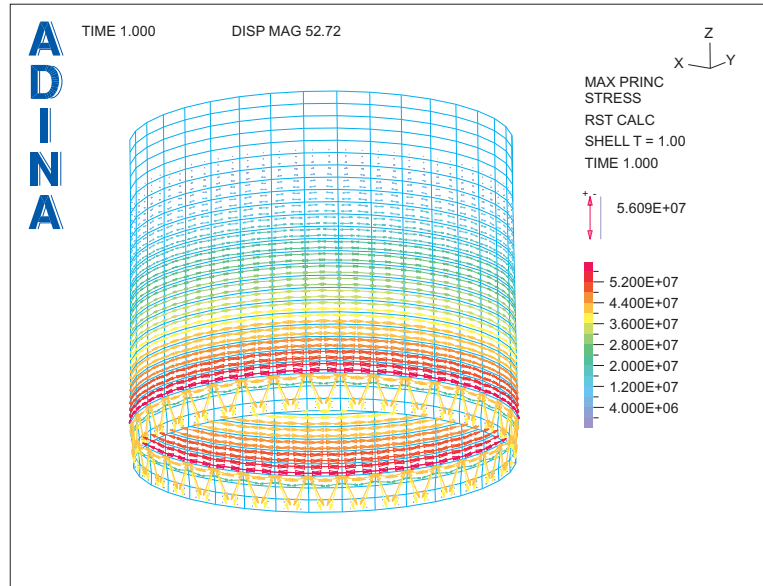
This plot shows that the maximum calculated hoop stresses are close to the estimated value (of course the estimate does not account for the stiffening effect of the tank floor).

Model definition for the frequency analysis

Set the Program Module drop-down list to ADINA Structures (you can discard all changes) and choose database file prob63.idb from the recent file list near the bottom of the File menu.

Restart analysis

Choose Control→Solution Process, set the Analysis Mode to Restart Run and click OK.

Frequency / modal participation factor analysis

Set the Analysis Type to Modal Participation Factors and click the Analysis Options icon to the right of this field. In the Modal Participation Factors dialog box, make sure that the Type of Excitation Load is Ground Motion, then click the Settings... button. In the Frequencies (Modes) dialog box, set the Solution Method to Lanczos Iteration, set the Number of Frequencies / Mode Shapes to 400, set the Number of Mode Shapes to be Printed to 400, check the "Allow Rigid Body Mode" button and click OK twice to close both dialog boxes.

In order to reduce the amount of information written to the porthole file, we do not want to save element results in this run. Choose Control→Porthole→Volume, uncheck the "Individual Element Results" field and click OK.

We also want to calculate the total mass of the model. Choose Control→Miscellaneous Options, check the "Calculate Mass Properties" field and click OK. (The total mass is used to calculate the percent modal masses in the listings below.)

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

Do not click the Save icon ▭, instead choose File→Save As and save the database to file `prob63_fr`.

Click the Data File/Solution icon ▤!, set the file name to `prob63_frb`, 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 static analysis. Enter restart file `prob63a` and click Copy.

This analysis might take around 5 minutes or so to run.

When the analysis finishes, 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 `prob63_frb`.

Post-processing for the frequency analysis

Error measures for the frequency solution

Choose List→Extreme Values→Zone, set the first "Variable to List" to (Frequency/Mode: EIGENSOLUTION_ERROR_MEASURE) and click Apply. On our computer, the maximum error measure is 3.92454E-04, corresponding to mode 147. You might get a different value, but the error measure should also be small (less than 1E-3).

Mode shapes and natural frequencies

When you click the Color Element Groups icon , the graphics window should look something like this:

The direction of the sloshing mode might be different on your computer platform, but the frequency should be the same.

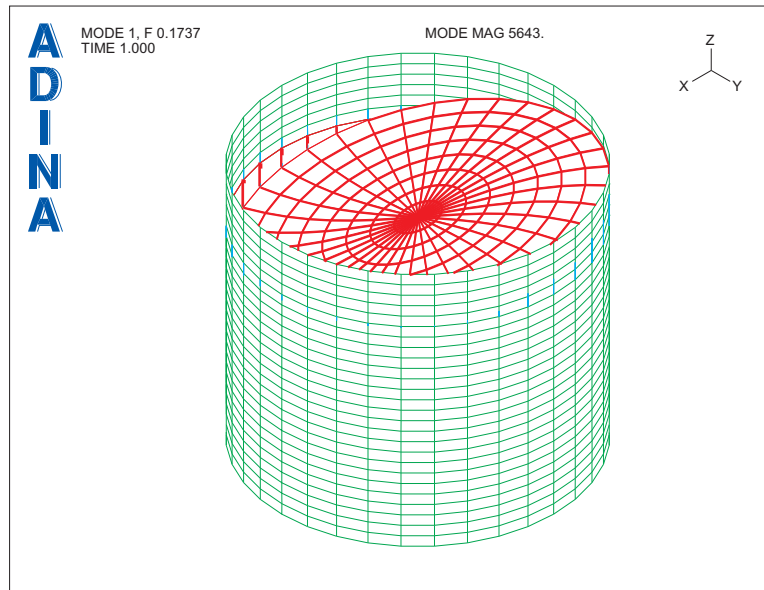Click the Next Solution icon ▶ to view the next mode shape. Mode 2 has the same frequency as mode 1, as must be expected due to the symmetry of the model. Thus the first sloshing mode is represented by both modes 1 and 2.

Notice that the free surface slips along the tank wall. This is why separate nodes are used for the shell and fluid meshes. (The elements in the potential-based mesh appear to be overdistorted, see comments at the end of this primer problem for an explanation.)

Repeatedly click the Next Solution icon ▶ to view more of the mode shapes. The first 269 mode shapes all involve different motions of the free surface, with very little motion of the tank walls.

Modal participation factors and modal masses

Choose List→Info→MPF. The first table in the listing gives the modal participation factors for each ground motion direction, the second table gives the modal masses, the third table gives the modal masses expressed as percentages of the total mass, the fourth table gives the accumulated modal masses and the fifth table gives the accumulated modal masses expressed as percentages of the total mass. Scroll to the bottom of the listing. You should notice that the accumulated modal masses for mode 400 are approximately 81.8% in the x and y directions, and approximately 87.5% in the z direction.

Now scroll upwards to the table with heading

		PERCENT	PERCENT	PERCENT
MODE	FREQUENCY	MASS(X)	MASS(Y)	MASS(Z)

We observe that modes 1 and 2 have frequency 1.73742E-01 (Hz) and that the sum of the x percent modal masses for the two modes is about 26.1%. (Because the modes are repeated, the percent modal mass for each of the modes, considered separately, will be different on different computer platforms, but the sum of the percent masses will be the same.) The sum of the y percent modal masses for the first two modes is equal to the sum of the x percent modal masses, as must be expected by symmetry.

Observe that the percent modal masses for most of the remaining modes are zero. This is because these modes are symmetric. Ground motions will not trigger these modes.

The next modes with significant modal mass are modes 316 and 317, with frequency 5.90436 (Hz).

To display mode 316, choose Definitions→Response, set the Response Name to MESHPLOT00001, set the Mode Shape Number to 316 and click OK. When you click the Redraw icon , the graphics window should look something like the figure on the next page.

This mode involves both sloshing and structural deformation. Since this is a repeated mode, the direction of the mode might be different on your computer platform, but the frequency should be the same.
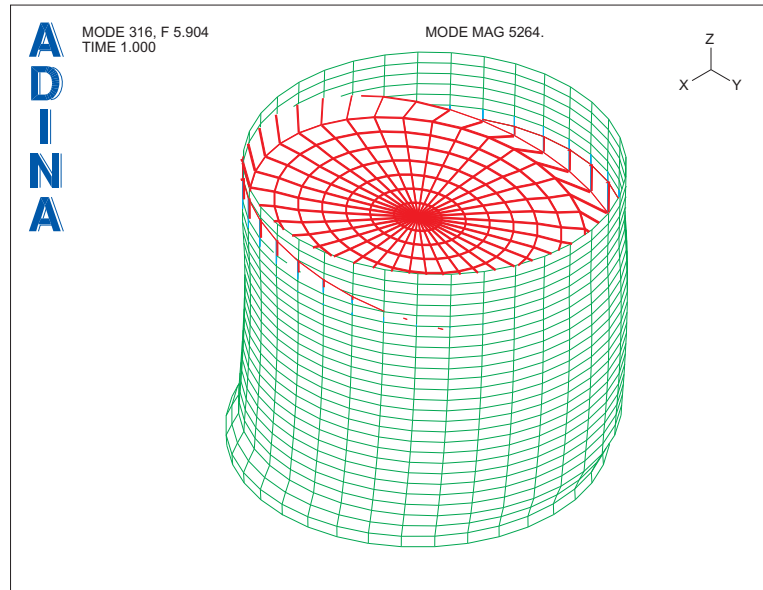
Model definition for the time history analysis

Set the Program Module drop-down list to ADINA Structures (you can discard all changes) and choose database file `prob63.idb` (not `prob63_fr.idb`) from the recent file list near the bottom of the File menu.

Restart analysis, dynamic analysis

Choose Control→Solution Process, set the Analysis Mode to Restart Run and click OK.

Set the Analysis Type to Dynamics-Implicit.

Ground acceleration

In the time history analysis, we would like to trigger the first sloshing mode. To do this, we will input a sinusoidal ground acceleration in the x direction with period equal to the period of the first sloshing mode. From the frequency analysis, the first sloshing mode has frequency 1.73742E-01 Hz, so the period is 5.75566 sec.

The ground acceleration time history that we will apply is shown in the figure on the next page.
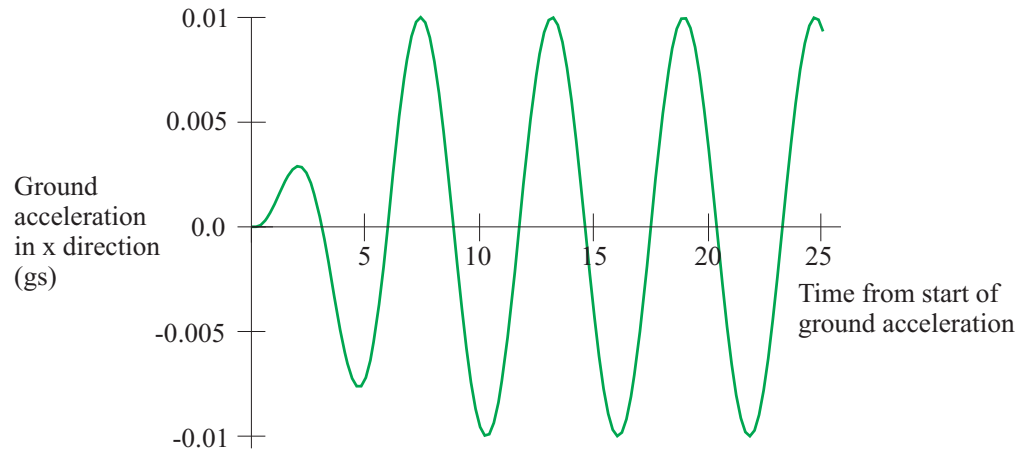
This time function is generated from the equation

$$\ddot{u}_{gx} = (0.01g)\frac{t}{t_p}\sin\frac{2\pi t}{t_p}, t \le t_p$$
$$= (0.01g)\sin\frac{2\pi t}{t_p}, t > t_p$$

and $t_p = 5.75566$ sec . See the notes at the end of this primer problem for more information on why the ground acceleration time history is not simply applied as a sinusoidal function.

The time step size is chosen to be 0.2 sec, so that there are about 30 time steps per loading cycle.

In the dynamic analysis, we restart from time 1.0. In addition, we would like to have one dynamic time step with zero ground acceleration. So the ground acceleration time history is shifted by 1.2 sec, so that the ground acceleration starts at time 1.2.

To apply this ground acceleration, we will input a mass-proportional load with magnitude 0.0981 (m/s^2), acting in direction (-1,0,0), with the time function shown in the figure on the next page.
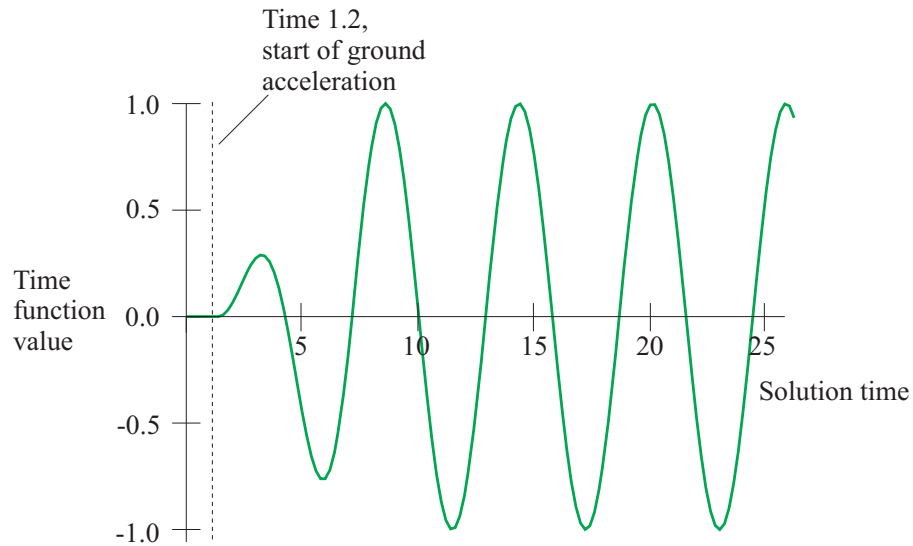
Choose Control→Time Step, and, in the first row of the table, set the # of steps to 125 and the Magnitude to 0.2, then click OK.

Choose Control→Time Function, add Time Function Number 2, click Import... and open file `prob63_tf.txt`, then click OK to close the Define Time Function dialog box.

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 Load dialog box, add load number 2, set the Magnitude to 0.0981 and set the Direction to (-1,0,0). Also set "Interpret Loading as (for potential-based fluid element only)" to "Ground Acceleration" and click OK. In the Apply Load dialog box, set the Load Number to 2, the Time Function to 2 and click OK.

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

Do not click the Save icon ⬛, instead choose File→Save As and save the database to file `prob63_td`.

Click the Data File/Solution icon ⬛, set the file name to `prob63_tdb`, 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 static analysis. Enter restart file `prob63a` and click Copy.

This analysis might take around 2 minutes or so to run.

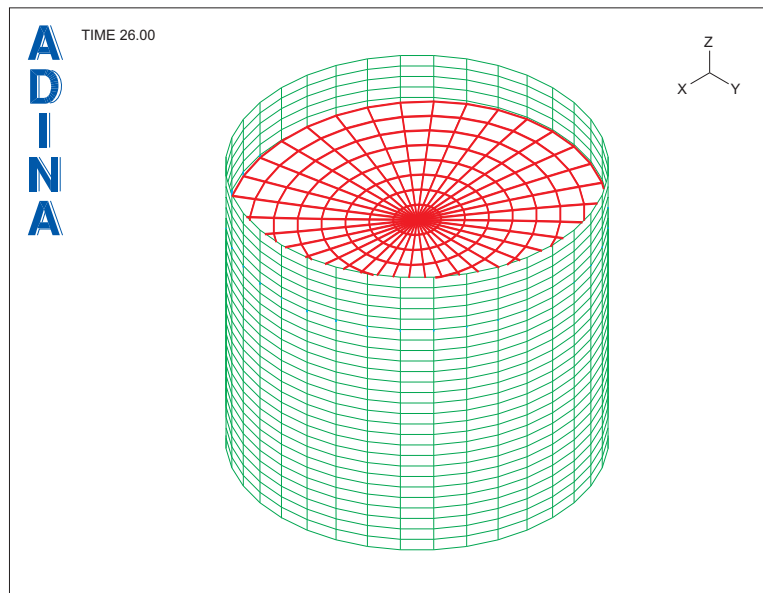When the analysis finishes, close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon ![icon] and open porthole file `prob63_tdb`.

Post-processing for the time history analysis

When you click the Color Element Groups icon ![icon], the graphics window should look something like this:

Click the Movie Load Step icon ![icon] to animate the solution. You will observe that the free surface moves with the shape given by mode shapes 1 and 2, and that the amplitude of the motion increase in each cycle.

Dynamic fluid pressure

We would like to visualize the fluid pressure due to the ground acceleration. Thus we need to subtract the static pressure (pressure from the static analysis) from the total pressure.

The static pressure itself consists of two parts, the fluid reference pressure, defined as $p_{ref} = \rho g z$, and an offset needed to set the reference pressure to zero on the free surface (approximately at $z = 21$). The variable FLUID_REFERENCE_PRESSURE gives the value of p_{ref}, but we need to calculate the offset ourselves. The offset is approximately

$(\rho g)(21) = 206010$.

Choose Definitions→Variable→Resultant, add variable DYNAMIC_PRESSURE, define it as

```
<FE_PRESSURE> - <FLUID_REFERENCE_PRESSURE> - 206010
```

and click OK.

We would also like to plot the fluid elements without the interface elements. Click the Define Zone icon , add zone FLUID, click the Edit button, enter

ELEMENT 1 TO 100000 OF ELEMENT GROUP 101

in row 1 of the table, then click OK. (Since there are less than 100000 elements in group 101, the above command selects all of the elements in group 101, without selecting any of the interface elements.) Then, in the Model Tree, right click on zone FLUID and choose Display.

Now click the Create Band Plot icon , choose variable (User Defined: DYNAMIC_PRESSURE) and click OK. The graphics window should look something like this:
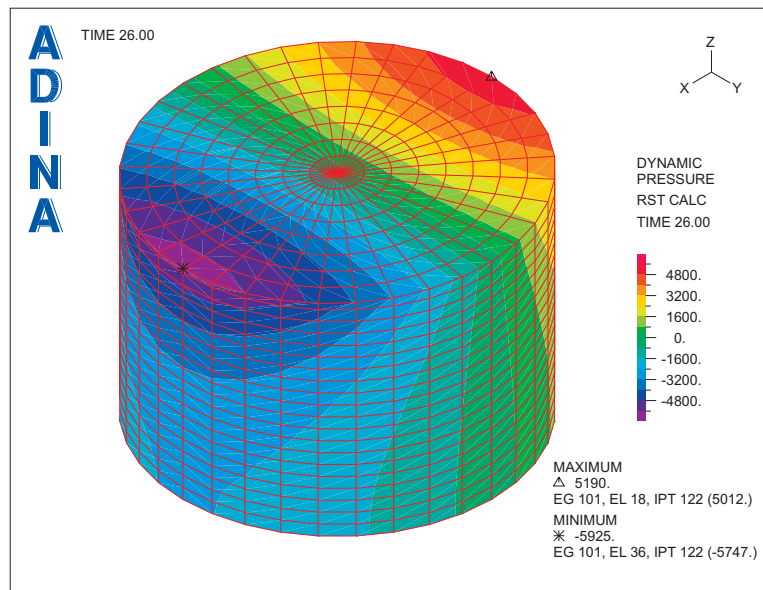
You will notice that the plotted dynamic pressure is non-zero on the free surface, see notes at the end of this problem for an explanation.

Click the Movie Load Step icon ▣ to create an animation of the dynamic pressure.

Now click the First Solution icon ◀ . Notice that the dynamic pressure is constant with value -367.6 (Pa) at time 1.2. Recall that there is no ground acceleration applied at time 1.2. The nonzero dynamic pressure is due to the approximate offset of 206010. This offset was computed using the undeformed fluid height of $z = 21$. However, due to the fluid compressibility and the tank compliance, the actual fluid height in the static solution is slightly different than 21.

Click the Clear Band Plot icon ▨ to remove the band plot.

Fluid velocities

Click the Create Vector Plot icon ↗, set the Vector Quantity to VELOCITY and click OK. The velocity vectors should all have values very close to 0 (on our computer, all values are of the order 1E-13). This shows that the first dynamic solution, with zero ground acceleration, is the same as the static solution.

Now click the Last Solution icon ▶ . The velocity vectors are all very long, because the same scale factor is used as was used in the previous plot. Click the Clear Vector Plot icon ▨, click the Create Vector Plot icon ↗, set the Vector Quantity to VELOCITY and click OK.

Unfortunately some of the vectors have the same color as the mesh plot. Click the Color Element Groups icon ▤ to uncolor the mesh plot. The graphics window should look something like the figure on the next page.

Click the Movie Load Step icon ▣ to create an animation of the fluid velocities.
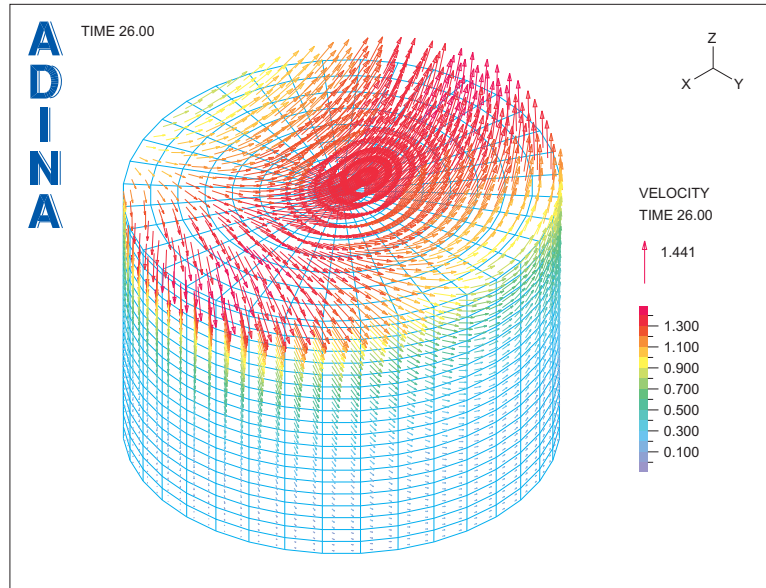
Dynamic tank deformations

Click the Clear icon ![CLEAR], then in the Model Tree, right click on zone EG1 and choose

Display. Click the Modify Mesh Plot icon ![icon] and click the Model Depiction button. Set the "Option for Plotting Original Mesh" to "Use Configuration at Reference Time", and set the "Reference Time for Original Mesh" to 1.0, then click OK twice to close both dialog boxes.

When you click the Scale Displacements icon ![10%], the graphics window should look something like the figure on the next page.

Recall that when ground motions are applied using mass-proportional loads, the computed displacements are relative to the ground motion.

Click the Movie Load Step icon ![icon] to create an animation of the dynamic tank deformations.

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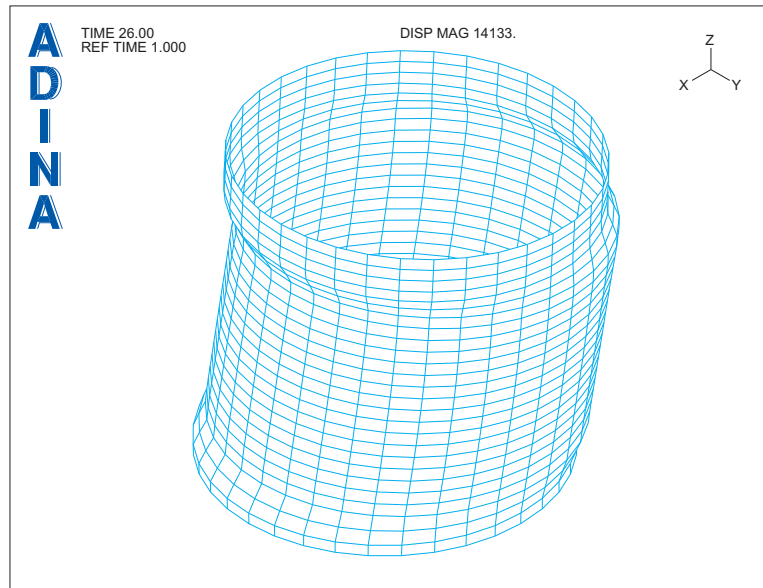
Notes

1) For general information about the potential-based fluid elements, see Section 2.11 of the ADINA Structures Theory and Modeling Guide. In particular, note that the linear potential-based formulation is used in this analysis, and the displacements of the fluid-structure boundary and the free surface are assumed to be small.

2) It is necessary to use separate but compatible meshes for the shell and fluid, in order that the free surface of the fluid slip along the tank walls. For a schematic example, see Example 1 in Section 2.11.15 of the ADINA Structures Theory and Modeling Guide.

3) Surface 4, which is used for the floor of the tank, is a degenerate surface, therefore during meshing of the shell elements, it is necessary to specify DEGENERATE=YES in the GSURFACE command.

4) Because both the structure and fluid are linear, this is a linear analysis. In the special case of a linear analysis, it is allowed to perform the frequency analysis immediately, without an initial static load. We do not demonstrate this here. The sequence of analyses given in this primer problem is applicable for both linear and nonlinear analysis.

5) In frequency analysis with potential-based fluid elements, it is a good idea to always check the eigensolution error measures, as demonstrated in this primer problem. If, for some reason, the error measures are large, ADINA Structures does not stop; instead ADINA Structures outputs the (wrong) eigenvalues and eigenvectors.

6) In the fluid mesh, the vertical displacements of the nodes that are not on the free surface are much smaller than the vertical displacements of the nodes on the free surface. This makes it appear as if the fluid elements on the free surface are undergoing large deformations while the remaining fluid elements are not moving. When the displacements are large enough, the elements appear to become overdistorted.

This is normal. The motions of the fluid inside potential-based fluid elements is unrelated to the motions (or lack of motions) of the nodes of these elements. The motions of potential-based fluid element nodes is used only to provide displacement boundary conditions for the potential-based elements.

7) The number of modes for motions of the free surface is approximately equal to the number of nodes on the free surface. Since free surface motions have very low frequency, most of the low modes are free surface motions. This is why it is necessary to compute several hundred modes for this problem.

As the mesh is refined and the number of nodes on the free surface increases, the number of modes for motions of the free surface also increases, and therefore the number of modes to be computed also increases.

8) For information about modal masses, see the ADINA Structures Theory and Modeling Guide, Section 9.1.

9) It is interesting to try the frequency analysis with more modes than are used in this problem, in order to see that the modal masses converge to the actual mass. We obtain the following results:

Number of modes	Highest natural frequency (Hz)	Percent modal mass in x direction	Percent modal mass in y direction	Percent modal mass in z direction
400	20.370	81.8	81.8	87.5
500	34.443	82.3	82.3	87.8
1000	150.558	95.6	95.6	94.8
2000	371.405	98.0	98.0	96.5
5000	733.238	98.5	98.5	97.5

10) Recall that when ground motions are applied using mass-proportional loads, the displacements are relative to the ground motion. Also recall that the direction of the mass-proportional load is opposite to the direction of the ground motion. See the ADINA Structures Theory and Modeling Guide, Section 5.4.4 for more information.

11) In potential-based fluid analysis with ground motions, ADINA Structures assumes that the ground displacement, velocity and acceleration are all zero at the start of the time history

analysis. If we had used $\ddot{u}_{gx} = (0.01g)\sin\dfrac{2\pi t}{t_p}$ throughout, the average ground velocity would have been non-zero, see discussion in the ADINA Structures Theory and Modeling Guide, Section 2.11.14.

12) The plotted dynamic pressure is non-zero at the top of the fluid mesh for the following reason. Internally, the fluid element degrees of freedom are the potentials. The displacements of the boundary nodes are used only to provide a coupling between the rest of the structure and the fluid elements. The pressures within the fluid elements are computed based on the undeformed element geometry.

Now consider node 1244, which is the node on the free surface corresponding to the maximum plotted dynamic pressure (8601 Pa). At time 26, the z displacement of this node is 8.7677E-1 (m), so this node is above the undeformed free surface. The pressure at this location on the undeformed free surface is $(\rho g)(8.76711\text{E-1}) = 8601$, which is the plotted value.

There is a subtle difference between the pressures computed within the fluid elements and the pressures computed within the fluid-structure interface elements. The pressures computed within the fluid elements do not contain the $\rho g u_z$ effect due to motion of the boundary (since this motion is assumed small). But the pressures computed within the fluid-structure interface elements do contain the $\rho g u_z$ effect.