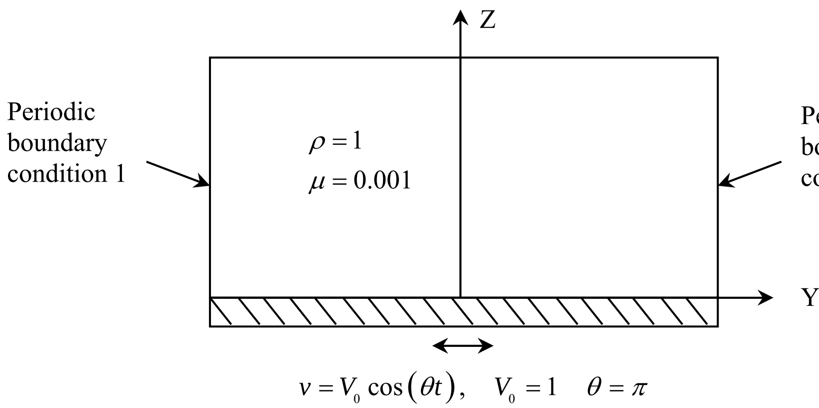


The second Stokes problem is 2D fluid flow above a plate that moves horizontally in a harmonic manner, schematically shown in the figure above.

Due to the viscous effect, the fluid above the plate is horizontally driven by the plate, and the velocity decreases along the vertical Z direction. The analytical solution of this problem can be written in the following form:

$$v = V_0 e^{-\eta} \cos(\theta t - \eta)$$
$$w = 0$$

where v is the horizontal velocity, w is the vertical velocity, V_0 is the maximum velocity magnitude, ρ and μ are the density and the dynamic viscosity of the fluid, θ is the angular frequency of the plate oscillation, and η is the normalized Z coordinate $z\sqrt{\rho\theta/(2\mu)}$. For this problem, we set $V_0 = 0.01$ and $\theta = \pi$, corresponding to the period $T = 2$.

In this problem, we will demonstrate the following topics:

- Periodic boundary condition
- Initial condition specified with a 2D tabular spatial function

We assume that you have worked through problems 1 to 61, 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 can be solved with the 900 nodes version of the ADINA System.

Much of the input for this problem is stored in files `prob62_1.in`, `prob62_1.plo`. You need to copy these files 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 CFD.

Defining model control data

Problem heading: Choose Control→Heading, enter the heading "Problem 62: 2D incompressible flow driven by a horizontally oscillating plate, the Second Stokes Problem" and click OK.

Flow assumptions: Choose Model→Flow Assumptions, set the Flow Dimension field to 2D (in YZ Plane), uncheck the "Include Heat Transfer" button, 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. This composite scheme has second order accuracy in time.

Element formulation: Choose Control→Solution Process, set the "Flow-Condition-Based Interpolation Elements" to FCBI-C (do not close the dialog box yet).

Outer iteration settings: In the Solution Process dialog box, click the "Outer Iteration…" button, then click the "Advanced Settings…" button. In the Outer Iteration Advanced Settings dialog box, set "Equation Residual Use" to All, and the "Tolerance" to 1E-06; then set "Variable Residual Use" to All and the "Tolerance" to 1E-06. Click OK three times to close all three dialog boxes.

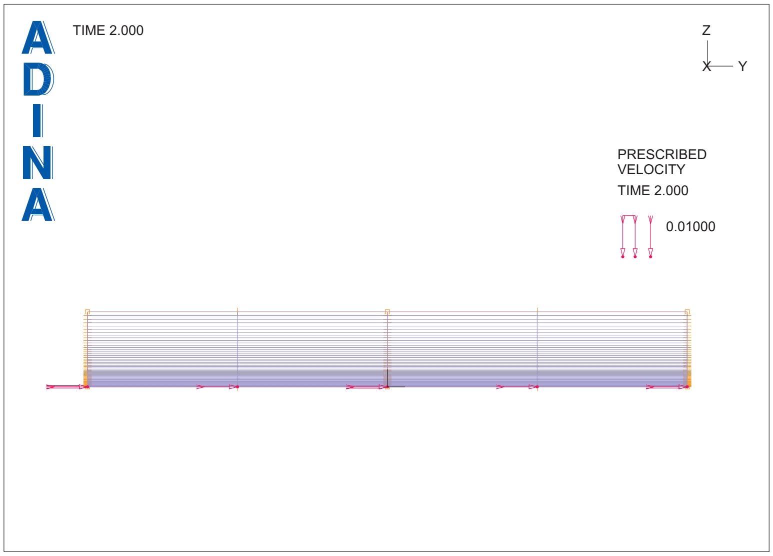
Time step setting: The period *T* is the characteristic time scale for this problem. Generally we need 20 to 100 time steps in one time period to accurately capture the transient physics, see the ADINA CFD Theory and Modeling Guide, section 13.1.3 for details. For this problem, we use 20 time steps with a step size $T/20 = 0.1$.

Choose Control→Time Step, set the Number of Steps to 20 and the Magnitude to 0.1 in the first row of the table, then click OK.

Defining geometry and materials

We have prepared a batch file (`prob62_1.in`) that contains the model geometry, material, mesh, and loadings.

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob62_1.in`, and click Open. The graphics window should look something like this:



The loading in this problem is the velocity on the bottom boundary. The harmonic oscillation of this loading is achieved through time function 2.

Time function 2: Choose Control→Time Function, set the Time Function Number to 2, and click "Graph" to see the time function. Click OK to close the dialog box.

Defining the periodic boundary conditions

In the problem, the left (line 4) and right (line 6) boundaries are set as periodic boundary conditions so that the infinite physical domain can be represented by a finite computational domain.

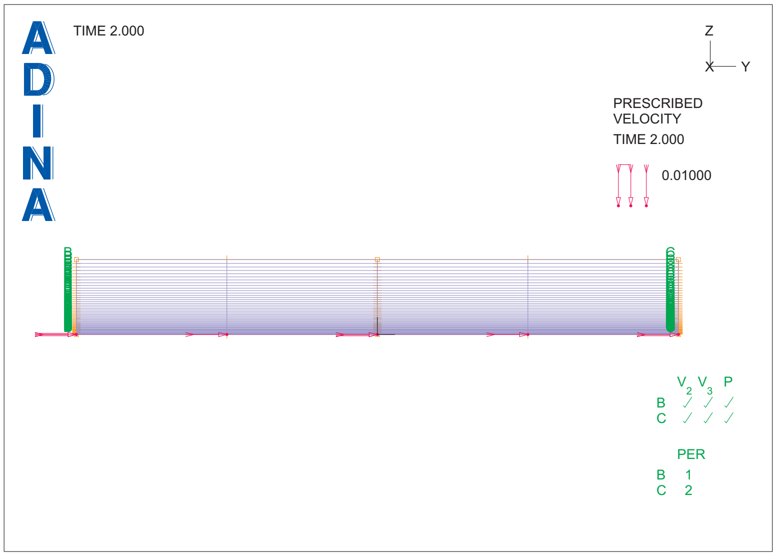
Periodic boundary conditions: Click the Special Boundary Condition icon SBC and add boundary condition 1. Set the "Type" to "Periodic". Double-click in the first row and column of the table, use the mouse to pick the left vertical line and press the Esc key to return to the "Define Special Boundary Condition" dialog box. Line number 4 should be entered in the table.

Now click the ... button to the right of the “Geometric Transformation” field. In the Define Transformation dialog box, add Transformation number 1. This transformation will be used by ADINA CFD to find the partner boundary (line 6) of the periodic boundary condition 1 (line 4). Set the “Coordinate Axes” X, Y, Z to (0, 2, 0). This is the positive coordinate increment from line 4 to line 6. Click OK to close the “Define Transformation” dialog box. In the “Define Special Boundary Condition” dialog box, set the “Geometric Transformation” to 1 and click Save (do not close the dialog box yet).

In the Define Special Boundary Condition dialog box, add boundary condition 2. Verify that the “Type” is “Periodic”. Double-click in the first row and column of the table, use the mouse to pick the right line and press the Esc key. Line number 6 should be entered in the table. Click Save (do not close the dialog box yet).

Click the “Boundary Condition Pair…” button. In the Define Pairs of Boundary Condition dialog box, set B.C. #1 to 1 and B.C. #2 to 2 in the first row of the table, then click OK to close this dialog box. Click OK to close the Define Special Boundary Condition dialog box.

When you click the Boundary Plot icon , the graphic window should look something like this:



Defining initial conditions

The initial condition profile of the Stokes second problem can be obtained directly from its analytical solution at $t = 0$:

$$v = V_0 e^{-\eta} \cos(-\eta)$$

Since this initial velocity is not constant, we need to use a spatial function to specify this initial condition. Note: if a uniform zero velocity is used as the initial condition, we can still get the oscillating velocity solution, but it will take a longer computational time to get the periodic oscillating solution.

Initial Condition: Click the Surface/Face Labels icon to show the surface labels (S1 and S2 in this problem). Choose Model→Initial Conditions→Define and add initial condition INIT.

Enter the following information into the first two rows of the table and click Save.

Variable	Value
Y-VELOCITY	0.01
Z-VELOCITY	0.0

Now click the Apply... button. In the Apply Initial Conditions dialog box, set the “Apply to” field to Face/Surface and make sure that the “Initial Condition” is “INIT”. Enter the following information into the first two rows of the table and click Save.

Face/Surface	Body #	Spatial Function
1		1
2		1

In this way, spatial function 1 is used along with initial condition INIT to determine the initial condition for both surfaces: the magnitude of the initial velocities is given by INIT × (spatial function 1). Click OK twice to close both dialog boxes.

Examining the spatial function: Choose Geometry→Spatial Functions→Surface. In the Define Surface Function dialog box, the “Number of Grid Columns” is 3, which corresponds to the 3 nodes in the y direction for each surface; the “Number of Grid Rows” is 201, which corresponds to the 201 nodes in the Z direction for each surface. The surface-function definition is contained in the file `prob62_1.in`.

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

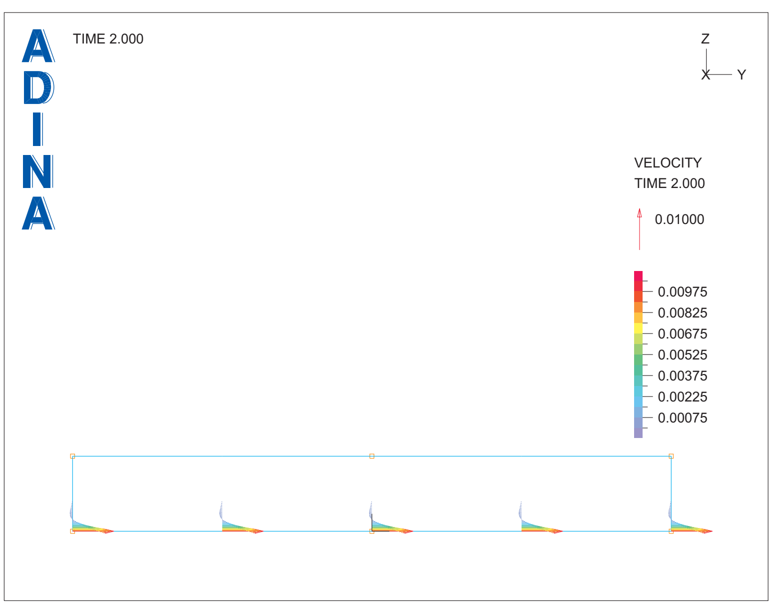
Click the Save icon and save the database to file `prob62`. Click the Data File/Solution icon, set the file name to `prob62`, make sure that the Run Solution button is checked and click Save.

When ADINA CFD 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, set the "File type" field to "ADINA-IN Database Files (*.idb)", open database file `prob62`, click the Open icon and open porthole file `prob62.por`.

Please notice that we first opened the ADINA-IN database, then loaded the porthole file. We did this so that we can define a model line on a geometry line; we can plot velocity profiles along the model line later on.

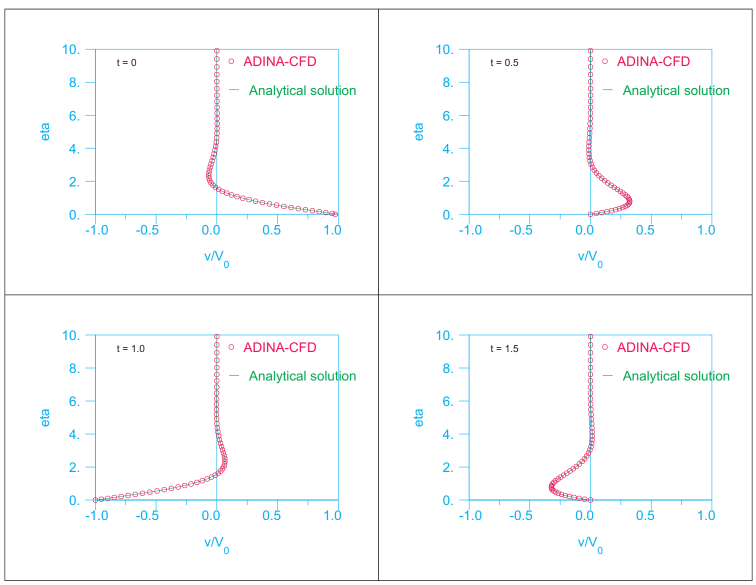
Examining the solution

Click the Show Geometry icon (to hide the geometry), the Model Outline icon and the Quick Vector Plot icon. The graphics window should look something like this:



Use the Previous Solution icon and Next Solution icon to display the other solutions at different times.

We have prepared the commands for creating the velocity profile along geometry line 2 in file `prob62_1.plo`. Choose File→Open Batch, navigate to the working directory or folder, select the file `prob62_1.plo` and click Open. The AUI processes the commands in the batch file. The graphics window should look something like this:



In these four plots, the velocity profiles at four different solution times are plotted along with the corresponding analytical solutions. In these plots, the x-axis is the normalized velocity v/V_0, and the y-axis is the normalized z coordinate η; the red circle symbol represents the computational solutions and the green line represents the analytical solutions.

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

Note

1) The surface function $f(u,v)$ can be defined as three types: linear, quadratic and tabular. We use tabular for this primer problem because the initial velocity profile for this problem is not either linear or quadratic. The range of the surface function is $0 \le u \le 1$ and $0 \le v \le 1$. The details on the surface functions, such as the orientations of the surface function axes, are available in the AUI Command Reference Manual Volume III: CFD & FSI, SURFACE-FUNCTION command.

This page intentionally left blank.