COMSOL MULTIPHYSICS®

ALE Fluid-Structure Interaction

SOLVED WITH COMSOL MULTIPHYSICS 3.5a

© COPYRIGHT 2008. All right reserved. No part of this documentation may be photocopied or reproduced in any form without prior written consent from COMSOL AB. COMSOL, COMSOL Multiphysics, COMSOL Reaction Engineering Lab, and FEMLAB are registered trademarks of COMSOL AB. Other product or brand names are trademarks or registered trademarks of their respective holders.

COMSOL

ALE Fluid-Structure Interaction

Introduction

The following example demonstrates techniques for modeling fluid-structure interactions in COMSOL Multiphysics. It illustrates how fluid flow can deform surrounding structures and how to solve for the flow in a continuously deforming geometry using the arbitrary Lagrangian-Eulerian (ALE) technique.

The model geometry consists of a horizontal flow channel in the middle of which is an obstacle, a narrow vertical structure (Figure 1). The fluid flows from left to right, except the obstacle forces it into a narrower path in the upper part of the channel, and it imposes a force on the structure's walls resulting from the viscous drag and fluid pressure. The structure, being made of a deformable material, bends under the applied load. Consequently, the fluid flow also follows a new path, so solving the flow in the original geometry would generate incorrect results.

The ALE method handles the dynamics of the deforming geometry and the moving boundaries with a moving grid. COMSOL Multiphysics computes new mesh coordinates on the channel area based on the movement of the structure's boundaries and on so called mesh smoothing. The Navier-Stokes equations that solve the flow are formulated for these moving coordinates.

The structural mechanics portion of the model does not require the ALE method, and COMSOL Multiphysics solves it in a fixed coordinate system as usual. However, the strains the model computes in this way are the only source for computing the deformed coordinates with ALE.

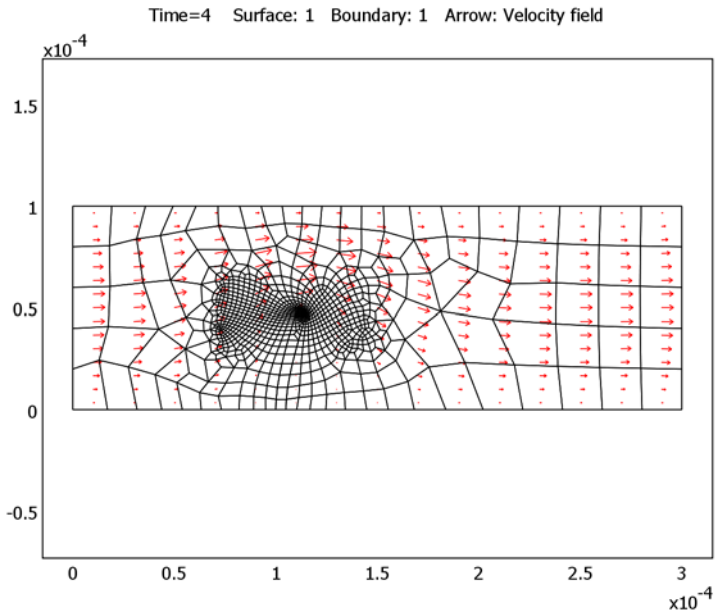

Figure 1: Fluid flows into this horizontal flow channel from the left, and it enters with a parabolic velocity profile. A narrow vertical structure in the channel (the straight vertical structure) forces the flow into a narrower path. Due to fluid pressure and viscous drag, the originally vertical structure bends (the vertical structure shifted to the right). This simulation models the fluid flow in a deformed, moving mesh that follows the movement of the bending structure.

Model Definition

In this example the flow channel is 100 μm high and 300 μm long. The vertical structure—5 μm wide, 50 μm high, and with a semicircular top—sits 100 μm away from the channel's left boundary. Assume that the structure is long in the direction perpendicular to the image.

The fluid is a water-like substance with a density $\rho = 1000$ kg/m^3 and dynamic viscosity $\eta = 0.001$ Pa·s. To demonstrate the desired techniques, assume the structure consists of a flexible material with a density $\rho = 7850$ kg/m^3 and Young's modulus $E = 200$ kPa.

The model consists of a fluid part, solved with the Navier-Stokes equations in the flow channel, and a structural mechanics part, which you solve in the obstacle. A Moving

Mesh (ALE) application mode makes sure the flow domain is deformed along with the obstacle.

Transient effects are taken into account in both the fluid and the structure. The structural deformations are modeled using large deformations in the Plane Strain application mode. The displacements and displacement velocities are denoted u, v, u_t, and v_t, respectively. Fluid flow is described by the Navier-Stokes equations, where the velocity components and pressure are denoted u_2, v_2, and p_2, respectively.

FLUID FLOW

The fluid flow in the channel is described by the Navier-Stokes equations for the velocity field, $\mathbf{u} = (u, v)$, and the pressure, p, in the spatial (deformed) moving coordinate system:

$$\rho\frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot [-p\mathbf{I} + \eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] + \rho((\mathbf{u} - \mathbf{u}_m) \cdot \nabla)\mathbf{u} = \mathbf{F}$$
$$-\nabla \cdot \mathbf{u} = 0 \qquad (1)$$

In these equations, $\mathbf{I}$ denotes the unit diagonal matrix and $\mathbf{F}$ is the volume force affecting the fluid. Assume that no gravitation or other volume forces affect the fluid, so that $\mathbf{F} = 0$. The coordinate system velocity is $\mathbf{u}_m = (u_m, v_m)$.

At the channel entrance on the left, the flow has fully developed laminar characteristics with a parabolic velocity profile but its amplitude changes with time. At first flow increases rapidly, reaching its peak value at 0.215 s; thereafter it gradually decreases to a steady-state value of 5 cm/s. The centerline velocity in the x direction, u_{in} (see Figure 4 on page 7), with the steady-state amplitude U comes from the equation

$$u_{\text{in}} = \frac{U \cdot t^2}{\sqrt{(0.04 - t^2)^2 + (0.1t)^2}} \qquad (2)$$

where t must be expressed in seconds.

At the outflow (right-hand boundary), the condition is $p = 0$. On the solid (non deforming) walls, no-slip conditions are imposed, $u = 0$, $v = 0$, while on the deforming interface the velocities equal the deformation rate, $u_0 = u_t$ and $v_0 = v_t$.

STRUCTURAL MECHANICS

The structural deformations are solved for using an elastic formulation and a nonlinear geometry formulation to allow large deformations.

For boundary conditions, the obstacle is fixed to the bottom of the fluid channel. All other object boundaries experience a load from the fluid, given by

$$\mathbf{F}_T = -\mathbf{n} \cdot (-p\mathbf{I} + \eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)) \quad (3)$$

where $\mathbf{n}$ is the normal vector to the boundary. This load represents a sum of pressure and viscous forces.

MOVING MESH

The motion of the deformed mesh is modeled using Winslow smoothing. For more information on this formulation, please refer to "The Moving Mesh Application Mode" on page 455 in the *COMSOL Multiphysics Modeling Guide*. This is the default smoothing when using the predefined Fluid-Structure Interaction coupled application modes. The boundary conditions control the displacement of the moving mesh with respect to the initial geometry. At the boundaries of the obstacle, this displacement is the same as the structural deformation. At the exterior boundaries of the flow domain, the deformation is set to zero in all directions.

Results and Discussion

Figure 2 shows the geometry deformation and flow at $t = 4$ s when the system is close to its steady state. Due to the channel's small dimensions, the Reynolds number of the flow is small (R << 100), and the flow stays laminar in most of the area. The swirls are

restricted to a small area behind the structure. The amount of deformation as well as the size and location of the swirls depend on the magnitude of the inflow velocity.

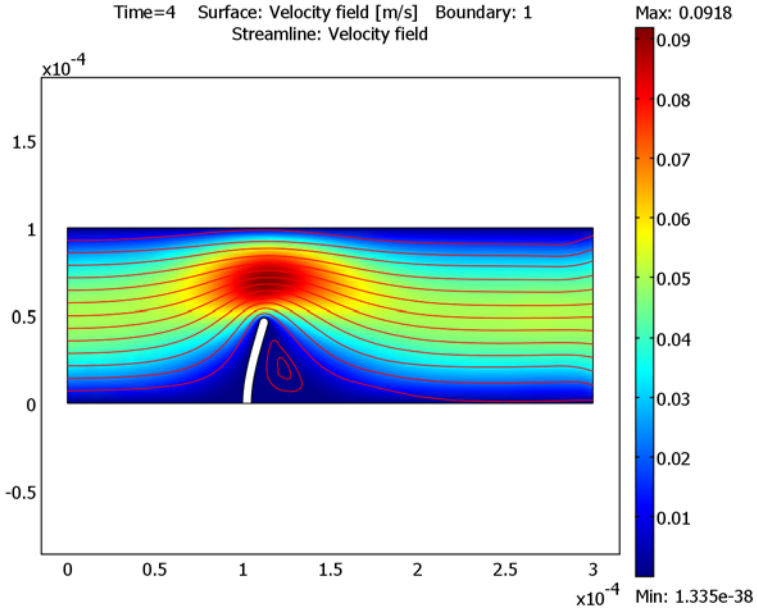

Figure 2: Flow velocity and geometry deformation at t = 4 s. The streamlines indicate the flow direction and the color indicates flow-velocity magnitude. Here the geometry deformation is scaled by unity.

Figure 3 shows the mesh velocity at $t = 0.15$ s. The boundaries of the narrow structure are the only moving boundaries of the flow channel. Therefore the mesh velocity also has its largest values near the structure. Depending on the current state of the deformation—whether it is increasing, decreasing or stationary—the mesh velocity can have a very different distribution. Figure 4 further illustrates this point; it compares the average inflow velocity to the horizontal mesh velocity and the horizontal mesh displacement just beside the top of the structure at the physical point $(1.05 \cdot 10^{-4}, 0.5 \cdot 10^{-4})$. Most of the time the deformation follows the inflow velocity quite closely. Whenever the inflow velocity starts to decrease, the deformation also decreases, which you can observe as the negative values on the horizontal mesh velocity. Toward the end

of the simulation, when inflow and structure deformation approach their steady-state values, the mesh velocity also decreases to zero.

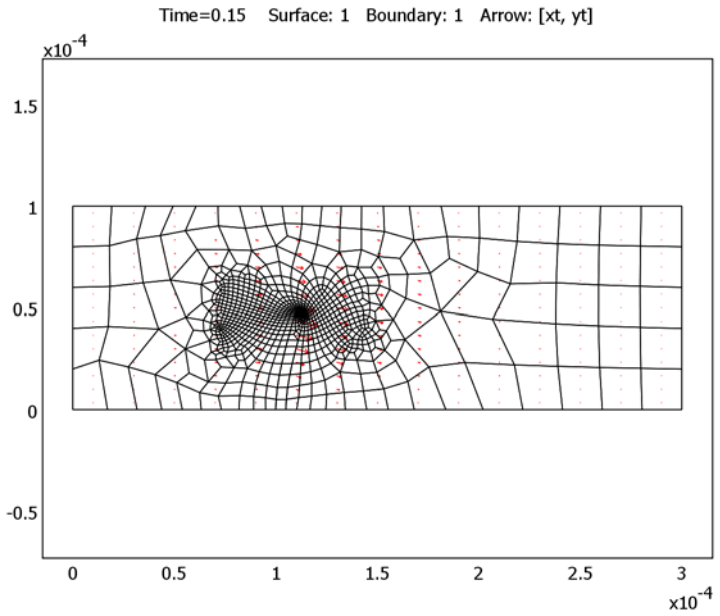

Figure 3: Mesh velocity (arrows) and mesh and geometry deformation at t = 0.15 s.

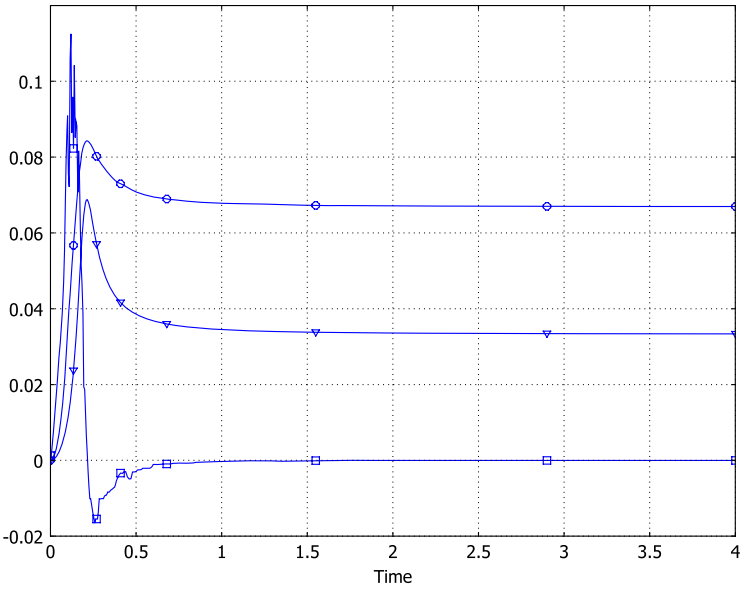

Figure 4: Inflow velocity, horizontal mesh velocity, and mesh deformation. The curve with triangles shows the average x direction velocity at the inflow boundary (m/s); the curve with circles shows $10^4 \times$mesh displacement in the x direction (dx_ale; m) at the geometry point ($1.05 \cdot 10^{-4}$, $0.5 \cdot 10^{-4}$); and the curve with squares shows $10^3 \times$mesh velocity in the x direction (xt; m/s), also at the point ($1.05 \cdot 10^{-4}$, $0.5 \cdot 10^{-4}$).

Figure 5 and Figure 6 compare the meshes at different times. The first image shows the initial mesh, which you generate prior to solving the model. This mesh is equally distributed around the top of the structure. The second image shows the mesh in its deformed form. Because the structure deforms more in the horizontal direction, the

mesh also changes more in this direction: On the left, the mesh elements are stretched; on the right, they are compressed in the x direction.

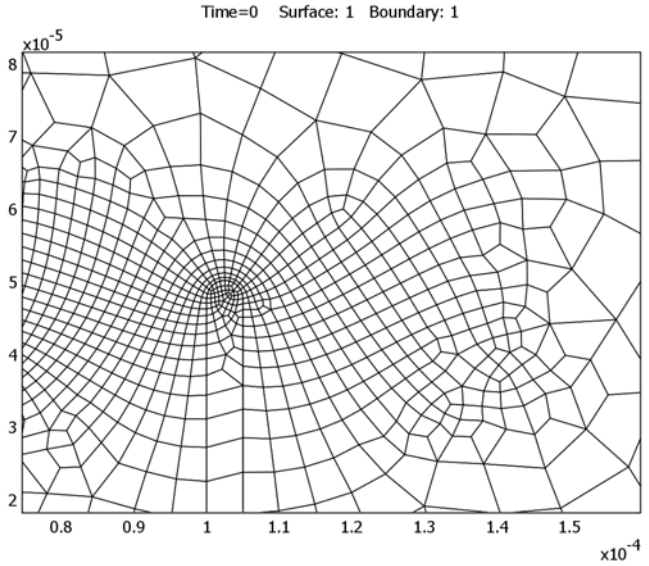

Figure 5: Initial (undeformed) geometry and mesh near the top of the structure.

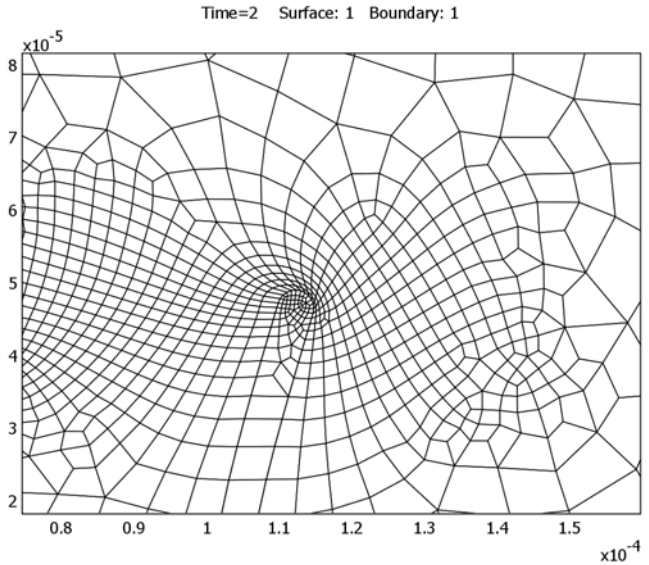

Figure 6: Deformed geometry and deformed mesh near the top of the structure at $t = 2$ s.

Modeling in COMSOL Multiphysics

This example implements the model using three application modes, which the predefined Fluid-Structure Interaction multiphysics entry sets up automatically:

- The Incompressible Navier-Stokes application mode computes the fluid dynamics. It is active only in the area of the flow channel.
- The Plane Strain application mode solves the model's structural mechanics portion. You activate it only in the area of the narrow structure. To get a more accurate computation of the large strains, large deformation analysis is the default setting. The deformation velocity solved with this application mode acts as the boundary condition for the Incompressible Navier-Stokes and the Moving Mesh (ALE) application modes.
- The Moving Mesh (ALE) application mode solves for the deformed mesh.

To solve the FSI problem efficiently, the model makes use of the transient segregated solver to solve for the fluid velocity and pressure in one group and the structural displacements and the mesh displacements in another group.

Model Library path: `MEMS_Module/Microfluidics_Models/fluid_structure_interaction`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. In the **Model Navigator**, select **2D** from the **Space dimension** list.
2. From the **Application Modes** list, select **MEMS Module>Fluid-Structure Interaction>Plane Strain with Fluid Interaction>Transient analysis**.
3. Click **OK** to close the **Model Navigator**.

GEOMETRY MODELING

1. Shift-click the **Rectangle/Square** button on the Draw toolbar.
2. In the **Width** edit field type `3e-4`, and in the **Height** edit field type `1e-4`. Click **OK**.
3. Click the **Zoom Extents** button on the Main toolbar.
4. Shift-click the **Rectangle/Square** button on the Draw toolbar.

5 In the **Width** edit field type `5e-6`, and in the **Height** edit field type `4.75e-5`. Enter the following positions: **X:** `1e-4` **Y:** `0`. Click **OK**.

6 Draw a circle by shift-clicking the **Ellipse/Circle (Centered)** button.

7 Enter `2.5e-6` in the **Radius** edit field and the positions **X:** `1.025e-4` **Y:** `4.75e-5`. Click **OK**.

8 From the **Draw** menu, open the **Create Composite Object** dialog box. Select **C1** and **R2**, clear the **Keep Interior Boundaries** check box, and then click **OK**.

PHYSICS SETTINGS

1 From the **Options** menu, choose **Constants**.

2 In the **Constants** dialog box, define the following constants with names, expressions, and (optionally) descriptions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
eta	1e-3[Pa*s]	Fluid viscosity
rho	1000[kg/m^3]	Fluid density
U	3.33[cm/s]	Inlet mean velocity at steady state
E	200[kPa]	Young's modulus

3 From the **Options** menu, choose **Scalar Expressions**.

4 Define the following expression (the description is optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
u_in	U*t^2/ sqrt(t^4-0.07[s^2]*t^2+0.0016[s^4])	Inlet mean velocity

Subdomain Settings

1 In the **Multiphysics** menu, make sure **Incompressible Navier-Stokes (mmglf)** is selected.

2 From **Physics** menu, select **Subdomain Settings**.

3 Select Subdomain 1. From the **Group** list, select **Fluid domain**, then apply the following settings:

PROPERTY	VALUE
ρ	rho
η	eta

4 Select Subdomain 2, then select **Solid domain** from the **Group** list. This deactivates the incompressible Navier-Stokes equations in Subdomain 2 (the solid).

5 Click **OK** to close the dialog box.

6 From the **Multiphysics** menu, select **Plane Strain (smpn)**.

7 From the **Physics** menu, select **Subdomain Settings**.

8 Select Subdomain 1, then select **Fluid domain** from the **Group** list to deactivate the structural mechanics equations in the fluid.

9 Select Subdomain 2, then select **Solid domain** from the **Group** list. In the **E** edit field for the Young's modulus, type `E`.

10 Click **OK** to close the dialog box.

11 From the **Multiphysics** menu, select **Moving Mesh (ALE)**.

12 From the **Physics** menu, open the **Subdomain Settings** dialog box.

13 Apply the following settings by selecting from the **Group** list.

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2
Group	Fluid domain	Solid domain

For the fluid domain, this means that the displacements are free. In the solid domain, the deformation of the solid determines the mesh displacement.

Boundary Conditions

1 From the **Multiphysics** menu, select **Incompressible Navier-Stokes (mmglf)**.

2 From the **Physics** menu, open the **Boundary Settings** dialog box.

3 Select Boundaries 4, 6, 9, and 10 (the sides and top of the obstacle), then select **Structural velocity** from the **Group** list.

4 For the remaining active boundaries, apply the following settings:

SETTINGS	BOUNDARY 1	BOUNDARY 8	ALL OTHERS
Boundary type	Inlet	Outlet	Wall
Boundary condition	Laminar inflow	Pressure, no viscous stress	No slip
U_0	`u_in`		
P_0		0	

5 Click **OK**.

6 From the **Multiphysics** menu, select **Plane Strain (smpn)**.

7 In the **Boundary Settings** dialog box, select Boundary 5 (the bottom of the obstacle). Then select **Fixed** from the **Constraint condition** list.

8 Select Boundaries 4, 6, 9, and 10 (the sides and top of the obstacle), then select **Fluid load** from the **Group** list. This sets up the edge loads using the Lagrange multipliers and specifies them to be defined as force/area.

9 Click **OK**.

10 From the **Multiphysics** menu, select **Moving Mesh (ALE)**.

11 In the **Boundary Settings** dialog box, select the following groups from the **Group** list to apply the appropriate mesh displacements:

SETTINGS	BOUNDARIES 4, 6, 9, 10	ALL OTHERS
Group	Structural displacement	Fixed

MESH GENERATION

1 From the **Mesh** menu, choose **Free Mesh Parameters**.

2 On the **Global** page, click the **Custom mesh size** button, then set the **Element growth rate** to `1.2`. This setting gives a slightly finer mesh near the tip of the pin.

3 Click the **Subdomain** tab, then select both subdomains. Set the **Method** to **Quad**.

4 Click **Remesh**. When the mesher has finished, click **OK**.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button on the Main toolbar.

2 From the **Solve** list, choose **Time dependent segregated**.

3 In the **Times** edit field type `range(0,0.005,0.4) range(0.41,0.01,0.8) range(0.85,0.05,4)`.

4 In the **Segregated groups** area, type `u2 v2 p2 lm5 lm6 lm7 Pinl_mmglf` in the **Components** edit field for Group 1. Type `u v x y` in the **Components** edit field for Group 2. Click the **Delete** button for Group 3 to remove that step.

5 Select the **Manual specification of segregated steps** check box.

6 In the **Manual specification of segregated steps** area, click the **Settings** button for Group 1.

7 In the **Damping/Termination Settings (Step 1)** dialog box, type `0.5` in the **Damping constant** edit field. Click **OK**.

8 In the **Manual specification of segregated steps** area, click the **Settings** button for Group 2.

9 In the **Damping/Termination Settings (Step 2)** dialog box, type `0.5` in the **Damping constant** edit field. Click **OK**.

The reduced damping is necessary to prevent that the moving mesh gives rise to inverted elements.

10 Click the **Time Stepping** tab.

11 Choose **BDF** from the **Method** list. For this multiphysics application, the BDF method works better than the generalized-α method for the time stepping.

12 Click **OK**.

13 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

To reproduce the plot in Figure 2 (this is also the plot that appears in the **Model Navigator** and when you open the prepared model file), follow these steps:

1 Click the **Plot Parameters** button on the Main toolbar.

2 On the **General** page, find the **Plot type** area. Select the **Surface**, **Boundary**, and **Streamline** check boxes; clear all the other check boxes.

3 Click the **Surface** tab. From the **Predefined quantities** list on the **Surface Data** page, select **Incompressible Navier-Stokes (mmglf)>Velocity field**.

4 Click the **Boundary** tab. Type 1 in the **Expression** edit field on the **Boundary Data** page.

5 Click the **Uniform color** option button, then click the **Color** button. Set the color to black, then click **OK** to close the **Boundary Color** dialog box.

6 Click the **Streamline** tab. From the **Predefined quantities** list on the **Streamline Data** page, select **Incompressible Navier-Stokes (mmglf)>Velocity field**.

7 Click the **Specify start point coordinates** option button. In the **X** and **Y** edit fields enter these expressions:

X	`0^(range(1,15)) 1.2e-4*1^(range(1,3))`
Y	`range(0,1e-4/14,1e-4) 0 0.175e-4 0.35e-4`

8 Click **OK** to close the **Plot Parameters** dialog box and generate the plot.