Sloshing Tank

Introduction

This 2D model demonstrates the ability of COMSOL Multiphysics to simulate dynamic free surface flow with the help of a moving mesh. The study models fluid motion with the incompressible Navier-Stokes equations. The fluid is initially at rest in a rectangular tank. The motion is driven by the gravity vector swinging back and forth, pointing up to 4 degrees away from the downward y direction at its extremes.

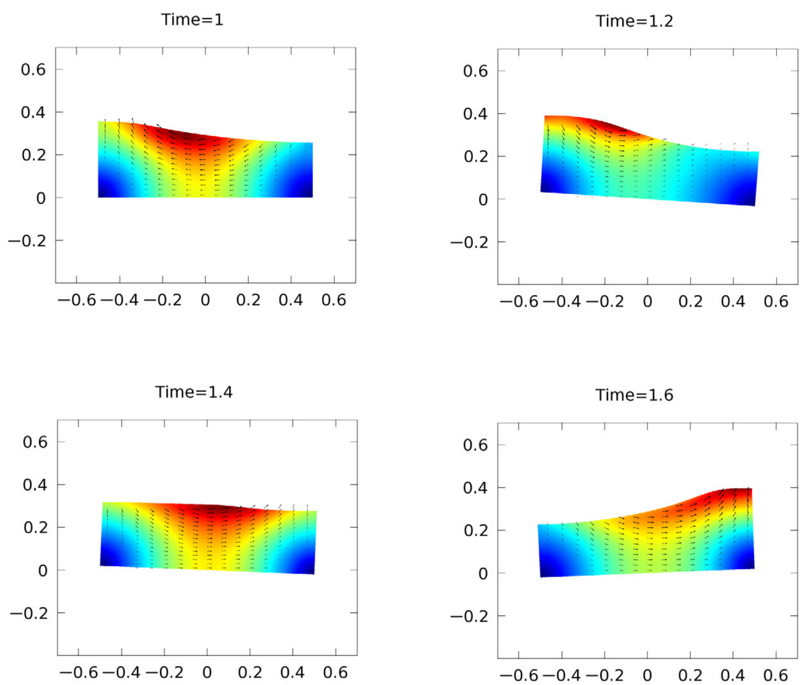

Figure 1: Snapshots of the velocity field at t = 1 s, t = 1.2 s, t = 1.4 s, and t = 1.6 s. The inclination of the gravity vector is indicated by the leaning of the tank.

Because the surface of the fluid is free to move, this model is a nonstandard computational task. The ALE (arbitrary Lagrangian-Eulerian) technique is, however, well suited for addressing such problems. Not only is it easy to set up using the Moving Mesh interface in COMSOL Multiphysics, but it also has the advantage that it represents the free surface boundary with a domain boundary on the moving mesh. This allows for the accurate evaluation of surface properties such as curvature, making surface tension analysis possible. Note, however, that this example model neglects surface tension effects.

Model Definition

DOMAIN EQUATIONS

This model describes the fluid dynamics with the incompressible Navier-Stokes equations:

$$\rho\frac{\partial \mathbf{u}}{\partial t} + \rho\mathbf{u}\cdot\nabla\mathbf{u} - \nabla\cdot(-p\mathbf{I} + \eta(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)) = \mathbf{F}$$

$$\nabla\cdot\mathbf{u} = 0$$

where ρ is the density, $\mathbf{u} = (u, v)$ is the fluid velocity, p is the pressure, $\mathbf{I}$ is the unit diagonal matrix, η is the viscosity, and $\mathbf{F}$ is the volume force. In this example model, the material properties are for glycerol: $\eta = 1.49$ Pa·s, and $\rho = 1.27\cdot10^3$ kg/m^3. The gravity vector enters the force term as

$$F_x = \rho g\sin(\phi_{\max}\sin(2\pi f t))$$

$$F_y = -\rho g\cos(\phi_{\max}\sin(2\pi f t))$$

where $g = 9.81$ m/s^2, $\phi_{\max} = 4°$, and $f = 1$ Hz.

With the help of the Moving Mesh interface, you can solve these equations on a freely moving deformed mesh, which constitutes the fluid domain. The deformation of this mesh relative to the initial shape of the domain is computed using Winslow smoothing. For more information, please refer to the chapter "Deformed Meshes" in the *COMSOL Multiphysics User's Guide*. COMSOL Multiphysics takes care of the transformation of the Navier-Stokes equations to the formulation on the moving mesh.

BOUNDARY CONDITIONS FOR THE FLUID

There are two types of boundaries in the model domain. Three solid walls, that are modeled with slip conditions, and one free boundary (the top boundary). The slip boundary condition for the Navier-Stokes equations is

$$\mathbf{u}\cdot\mathbf{n} = 0$$

where $\mathbf{n} = (n_x, n_y)^{\mathrm{T}}$ is the boundary normal. Because the normal vector depends on the degrees of freedom for the moving mesh, a constraint force would act not only on the fluid equations but also on the moving mesh equations. This effect would not be correct, and one remedy is to use weak constraints. For more information about weak constraints, see the section "Using Weak Constraints" in the *COMSOL Multiphysics User's Guide*.

Activate the weak constraints to enforce the slip boundary condition without a constraint force acting on the moving mesh equations:

$$\hat{\lambda}(\mathbf{u}\cdot\mathbf{n})-\lambda(\hat{\mathbf{u}}\cdot\mathbf{n}) \tag{1}$$

for some Lagrange multiplier variable λ. Here $\hat{\lambda}$ and $\hat{\mathbf{u}}$ denote test functions. See the step-by-step instructions later in this model documentation for details.

The fluid is free to move on the top boundary. The stress in the surrounding environment is neglected. Therefore the stress continuity condition on the free boundary reads

$$(-p\mathbf{I}+\eta(\nabla\mathbf{u}+(\nabla\mathbf{u})^{\mathrm{T}}))\cdot\mathbf{n} = -p_0\mathbf{n}$$

where p_0 is the surrounding (constant) pressure and η the viscosity of the fluid. Without loss of generality, $p_0 = 0$ for this model.

BOUNDARY CONDITIONS FOR THE MESH

In order to follow the motion of the fluid with the moving mesh, it is necessary to (at least) couple the mesh motion to the fluid motion normal to the surface. It turns out that for this type of free surface motion, it is important to not couple the mesh motion to the fluid motion in the tangential direction. If you would do so, the mesh soon becomes so deformed that the solution no longer converges. The boundary condition for the mesh equations on the free surface is therefore

$$(x_t\, y_t)^{\mathrm{T}}\cdot\mathbf{n} = \mathbf{u}\cdot\mathbf{n}$$

where $\mathbf{n}$ is the boundary normal and $(x_t, y_t)^{\mathrm{T}}$ the velocity of mesh (see the section "Mathematical Description of the Mesh Movement" in the *COMSOL Multiphysics User's Guide*). In the Moving Mesh interface, you specify this boundary condition by selecting the tangent and normal coordinate system in the deformed mesh and by specifying a mesh velocity in the normal direction, where you enter the right-hand side expression from above as `u*nx+v*ny`. For this boundary condition with the weak constrains activated, Moving Mesh interface adds the weak expression

$$\hat{\lambda}((\,(x_t\, y_t)^{T}-\mathbf{u})\cdot\mathbf{n})-\lambda(\,(\hat{x}\,\hat{y})^{T}\cdot\mathbf{n})$$

to ensure that there are no constraint forces acting on the fluid equations. Here again, λ denotes some Lagrange multiplier variable (not the same as before) and $\hat{\lambda}$, $\hat{x}$, and $\hat{y}$ denote test functions.

To be able to follow the fluid motion with the mesh motion, the moving mesh must not be constrained in the tangential direction on the side walls. In the Moving Mesh interface, you specify this boundary condition by using the global coordinate system and setting the mesh displacement to zero in the x direction. At the bottom of the tank the mesh is fixed, which you obtain in a similar way by setting the mesh displacements to zero in both the x and y directions.

Results

Figure 2 below and Figure 1 show the tank at a few different points in time. The colors represent the velocity field. Whereas you set up the model using a fixed tank and a swinging gravity vector, deformation plots enable you to give the tank an inclination at the postprocessing stage. The inclination angle of the tank is exactly the same as the angle of the gravity vector from its initial vertical position.

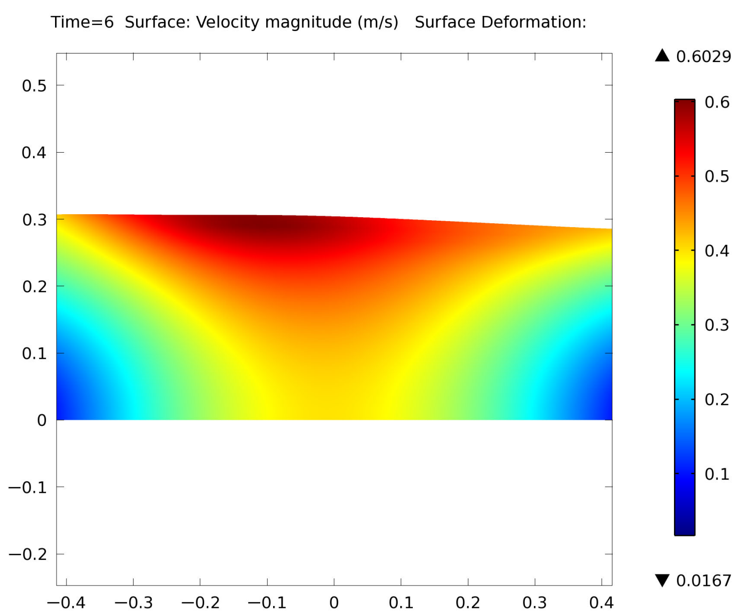

Figure 2: Velocity field inside the tank at t = 6 s.

To illustrate the dynamics in the tank, you can plot the wave height versus time at one of the vertical walls, as in the following plot.

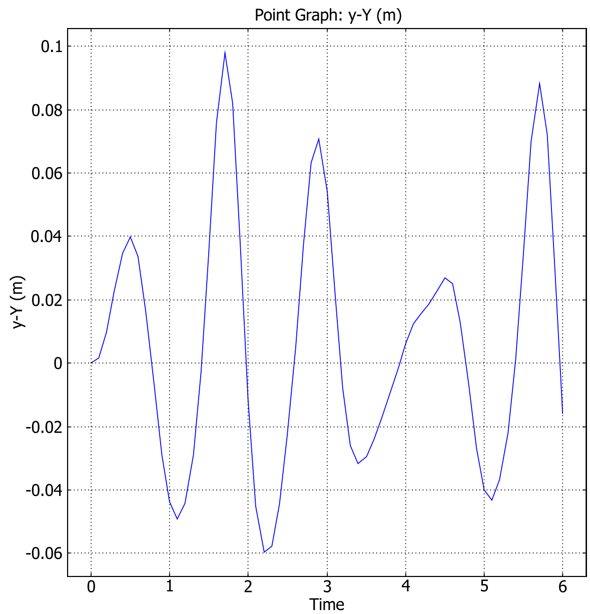

Figure 3: Wave height at $X = 0.5$ *m for* $t \in [0, 6\ s]$.

Model Library path: `COMSOL_Multiphysics/Fluid_Dynamics/sloshing_tank`

Modeling Instructions.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D** button.

3 Click **Next**.

4 In the **Add Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.

5 Click **Add Selected**.

6 In the **Add Physics** tree, select **Mathematics>Deformed Mesh>Moving Mesh (ale)**.

7 Click **Add Selected**.

8 Click **Next**.

9 In the **Studies** tree, select **Preset Studies for Selected Physics>Time Dependent**.

10 Click **Finish**.

GEOMETRY 1

Rectangle 1

1 In the **Model Builder** window, right-click **Model 1>Geometry 1** and choose **Rectangle**.

2 Go to the **Settings** window for Rectangle.

3 Locate the **Size** section. In the **Height** edit field, type `0.3`.

4 Locate the **Position** section. In the **x** edit field, type `-0.5`.

5 Click the **Build All** button.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 Go to the **Settings** window for Parameters.

3 Locate the **Parameters** section. In the **Parameters** table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
phi_max	4[deg]	Maximum angle of inclination
freq	1[Hz]	Frequency

DEFINITIONS

Variables 1

1 In the **Model Builder** window, right-click **Model 1>Definitions** and choose **Variables**.

2 Go to the **Settings** window for Variables.

3 Locate the **Variables** section. In the **Variables** table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
phi	phi_max*sin(2*pi*freq*t)	Angle of inclination
grav_x	g_const*sin(phi)	Gravity vector x component
grav_y	-g_const*cos(phi)	Gravity vector y component

Here, `g_const` is a predefined constant for the acceleration of gravity.

MATERIALS

Material 1

1 In the **Model Builder** window, right-click **Model 1>Materials** and choose **Material**.

2 Go to the **Settings** window for Material.

3 Locate the **Material Contents** section. In the **Material Contents** table, enter the following settings:

PROPERTY	NAME	VALUE
Density	rho	1270
Dynamic viscosity	mu	1.49

LAMINAR FLOW

1 In the **Model Builder** window, click **Model 1>Laminar Flow**.

2 Go to the **Settings** window for Laminar Flow.

3 Locate the **Physical Model** section. From the **Compressibility** list, select **Incompressible flow**.

Volume Force 1

1 Right-click **Model 1>Laminar Flow** and choose **Volume Force**.

2 Select Domain 1 only.

3 Go to the **Settings** window for Volume Force.

4 Locate the **Volume Force** section. Specify the **F** vector as

grav_x*spf.rho	x
grav_y*spf.rho	y

Wall 2

1 In the **Model Builder** window, right-click **Laminar Flow** and choose **Wall**.

2 Select Boundaries 1, 2, and 4 only.

3 Go to the **Settings** window for Wall.

4 Locate the **Boundary Condition** section. From the **Boundary condition** list, select **Slip**.

5 From the **Model Builder** window's **View Menu**, choose **Show More Options**.

6 Locate the **Constraint Settings** section. Select the **Use weak constraints** check box. As discussed in the Model Definition section, turning on weak constraints avoids adding a superficial constraint force.

Open Boundary 1

1 In the **Model Builder** window, right-click **Laminar Flow** and choose **Open Boundary**.

2 Select Boundary 3 only.

MOVING MESH

1 In the **Model Builder** window, click **Model 1>Moving Mesh**.

2 Go to the **Settings** window for Moving Mesh.

3 Locate the **Free Deformation Settings** section. From the **Mesh smoothing type** list, select **Winslow**.

Free Deformation 1

1 Right-click **Model 1>Moving Mesh** and choose **Free Deformation**.

2 Select Domain 1 only.

Prescribed Mesh Displacement 2

1 In the **Model Builder** window, right-click **Moving Mesh** and choose **Prescribed Mesh Displacement**.

2 Select Boundaries 1 and 4 only.

3 Go to the **Settings** window for Prescribed Mesh Displacement.

4 Locate the **Prescribed Mesh Displacement** section. Clear the **Prescribed y displacement** check box.

Prescribed Mesh Velocity 1

1 In the **Model Builder** window, right-click **Moving Mesh** and choose **Prescribed Mesh Velocity**.

2 Select Boundary 3 only.

3 Go to the **Settings** window for Prescribed Mesh Velocity.

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, select **Boundary System 1 (sys1)**.

5 Locate the **Prescribed Mesh Velocity** section. Clear the **Prescribed t1 velocity** check box.

6 In the v_y edit field, type `u*nx+v*ny`.

7 Locate the **Constraint Settings** section. Select the **Use weak constraints** check box.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, click **Study 1>Step 1: Time Dependent**.
2 Go to the **Settings** window for Time Dependent.
3 Locate the **Study Settings** section. In the **Times** edit field, type `range(0,0.1,6)`.
4 Select the **Relative tolerance** check box.
5 In the associated edit field, type `0.001`.
6 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.

Solver 1

Exclude the pressure and the moving mesh variables from error estimation. The equations for those variables do not include time derivatives and become algebraic.

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1** node, then click **Time-Dependent Solver 1**.
2 Go to the **Settings** window for Time-Dependent Solver.
3 Click to expand the **Advanced** section.
4 From the **Error estimation** list, select **Exclude algebraic**.
5 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

2D Plot Group 1

The default plot shows the velocity magnitude in the material frame. You can visualize the tank inclination using the deformation plot feature.

1 Go to the **Settings** window for 2D Plot Group.
2 Locate the **Plot Settings** section. Clear the **Plot data set edges** check box.
3 In the **Model Builder** window, expand the **2D Plot Group 1** node.
4 Right-click **Surface 1** and choose **Deformation**.
5 Go to the **Settings** window for Deformation.
6 Locate the **Expression** section. In the **x component** edit field, type `Y*sin(phi)`.
7 In the **y component** edit field, type `-X*sin(phi)`.
8 Locate the **Scale** section. Select the **Scale factor** check box.
9 In the associated edit field, type `1`.
10 Click the **Plot** button and compare the result with the plot in Figure 2.

To produce the series of snapshots of the velocity field shown in Figure 1, proceed with the following steps:

1 In the **Model Builder** window, right-click **2D Plot Group 1** and choose **Arrow Surface**.
2 Go to the **Settings** window for Arrow Surface.
3 Locate the **Coloring and Style** section. From the **Color** list, select **Black**.
4 In the **Model Builder** window, click **Surface 1**.
5 Go to the **Settings** window for Surface.
6 Locate the **Coloring and Style** section. Clear the **Color legend** check box.
7 In the **Model Builder** window, click **2D Plot Group 1**.
8 Go to the **Settings** window for 2D Plot Group.
9 Locate the **Data** section. From the **Time** list, select **1**.
10 Locate the **Plot Settings** section. Select the **Title** check box.
11 In the associated edit field, type `Time=1`.
12 Click the **Plot** button to reproduce the upper-left plot in Figure 1.
13 To reproduce the remaining three plots, repeat Steps 9 through 12 for the time values 1.2, 1.4, and 1.6 s.

Report

To see the waves in action, do the following:

1 In the **Model Builder** window, right-click **Results>Report** and choose **Player**.
2 Right-click **Player 1** and choose **Generate Frame**.
3 Right-click **Player 1** and choose **Play**.
4 Right-click **Results** and choose **1D Plot Group**.

1D Plot Group 2

To get a more comprehensive overview of the sloshing, you can plot the y-displacement from equilibrium in a point as in Figure 3.

1 In the **Model Builder** window, right-click **Results>1D Plot Group 2** and choose **Point Graph**.
2 Select Vertex 4 only.
3 Go to the **Settings** window for Point Graph.
4 Locate the **y-Axis Data** section. In the **Expression** edit field, type `y-Y`.
5 Click the **Plot** button.