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.

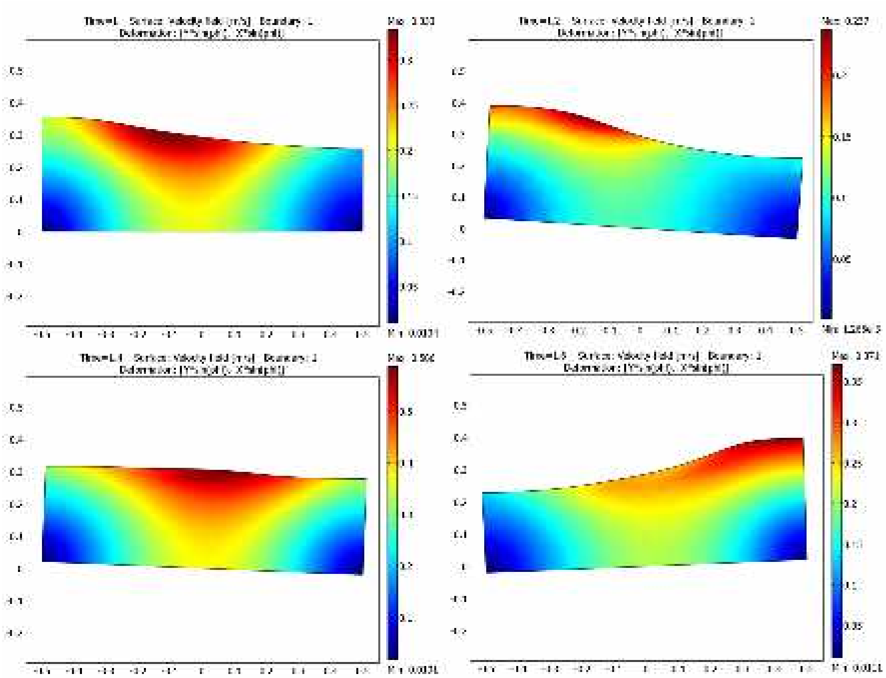

Figure 6-15: Snapshots of the velocity field at t = 1 s, t = 1.2 s, t = 1.4 s, and t = 1.6 s. The inclination of the gravity vector is indicated by the leaning of the tank.

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 (ALE) application mode 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, $\phi_{\max} = 4\pi/180$, and f = 1 Hz.

With the help of the Moving Mesh (ALE) application mode, 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 Moving Mesh Application Mode" on page 401 in the *COMSOL Multiphysics Modeling 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 $\mathbf{n} = (n_x, n_y)^T$ is the boundary normal. To enforce this boundary condition, select the Symmetry boundary type in the Incompressible Navier-Stokes application mode. 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 non-ideal weak constraints. Ideal weak constraints (the other type of weak constraints) do not remove this effect of the constraint force. For more information about weak constraints, see "Using Weak Constraints" on page 300 in the *COMSOL Multiphysics Modeling Guide*. The Incompressible Navier-Stokes application mode does not make use of weak constraints by default, so you need to activate the non-ideal weak constraints.

The following weak expression, which you add to the model, enforces the slip boundary condition without a constraint force acting on the moving mesh equations:

(6-9) $\hat{\lambda}(\mathbf{u} \cdot \mathbf{n}) - \lambda(\hat{\mathbf{u}} \cdot \mathbf{n})$

for some Lagrange multiplier variable λ. Here λ and $\mathbf{u}$ 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

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

where p_0 is the surrounding (constant) pressure and η the viscosity in 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

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

where $\mathbf{n}$ is the boundary normal and $(x_t, y_t)^T$ the velocity of mesh (see "Mathematical Description of the Mesh Movement" on page 392 in the *COMSOL Multiphysics Modeling Guide*). In the Moving Mesh (ALE) application mode, 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`. The Moving Mesh (ALE) application mode uses non-ideal weak constraints by default, and for this boundary condition it 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 λ, x, and y denote test functions. There is no need to modify this expression. Choose **Physics>Equation System>Boundary Settings** and select the free boundary (boundary 3) to see how to enter this expression in COMSOL Multiphysics. The expression implies that there is a flux (or force) on the free boundary for the moving mesh coordinate equations $\nabla x \cdot \mathbf{n} = \lambda n_x$ and $\nabla y \cdot \mathbf{n} = \lambda n_y$, respectively. Furthermore, 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 (ALE) application mode, 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 6-15 on page 247 shows the tank at a few different points in time. The colors represent the velocity field. Whereas the modeling is set up using a fixed tank and a swinging gravity vector, postprocessing using a deformation plot gives the tank a corresponding inclination. The inclination angle of the tank is exactly the same as the angle of the gravity vector from its initial vertical position.

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.

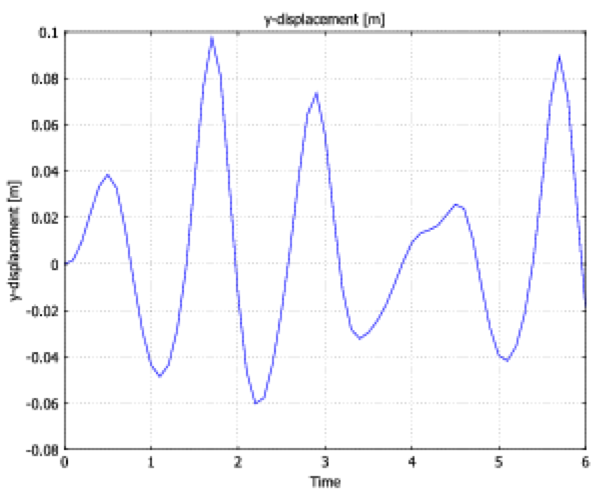

Figure 6-16: Wave height at X = 0.5 m for 0 £ t £ 20 s.

The movie file that accompanies this model shows the waves in the swinging tank, with a color scale indicating the vorticity.

Model Library Path: `COMSOL_Multiphysics/Fluid_Dynamics/sloshing_tank`

Modeling Using the Graphical User Interface

1 Start COMSOL Multiphysics.

2 In the **Model Navigator**, click the **Multiphysics** button.

3 Select **2D** from the **Space dimension** list.

4 Select **COMSOL Multiphysics>Deformed Mesh>Moving Mesh (ALE)>Transient analysis** and click **Add**.

5 Click the **Application Mode Properties** button.

6 Select **Winslow** from the **Smoothing method** list. Click **OK**.

7 Select **COMSOL Multiphysics>Fluid Dynamics>Incompressible Navier-Stokes>Transient analysis** and click **Add**.

8 Click **OK**.

GEOMETRY MODELING

1 Shift-click the **Rectangle/Square** button in the Draw toolbar.

2 Specify the rectangle settings according to the table below

PROPERTY	EXPRESSION
Width	1
Height	0.3
Position: Base	Corner
Position: x	-0.5
Position: y	0

3 Click the **Zoom Extents** button on the Main toolbar.

OPTIONS AND SETTINGS

1 Open the **Constants** dialog box from the **Options** menu and enter the following constants. The descriptions are optional. When done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
rho	1270[kg/m^3]	Glycerol density
nu	1.49[Pa*s]	Glycerol viscosity
phi_max	(4*pi/180)[rad]	Maximum angle of inclination
freq	1[Hz]	Frequency
g	9.81[m/s^2]	Acceleration due to gravity

2 From the **Options** menu, choose **Expressions>Scalar Expressions**.

3 Enter the following scalar variables with names, expressions, and descriptions (the descriptions are optional); when done, click **OK**.

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

PHYSICS SETTINGS

Properties

1 In the Incompressible Navier-Stokes application mode, choose **Properties** from the **Physics** menu.

2 In the **Application Mode Properties** dialog box, select **On** from the **Weak constraints** list and **Non-ideal** from the **Constraint type** list; then click **OK**.

Subdomain Settings

Open the **Subdomain Settings** dialog box and apply the settings in the table below.

SETTINGS	SUBDOMAIN 1
ρ	`rho`
η	`nu`
F_x	`grav_x*rho`
F_y	`grav_y*rho`

Boundary Conditions

1 Open the **Boundary Settings** dialog box from the **Physics** menu and enter boundary conditions according to the table below.

SETTINGS	BOUNDARIES 1, 2, 4	BOUNDARY 3
Boundary type	Wall	Open boundary
Boundary condition	Slip	Normal stress
f_0		`0`

2 Click **OK**.

3 Go to the **Multiphysics** menu and select **Moving Mesh (ALE)**.

4 In the **Boundary Settings** dialog box, apply the following boundary conditions for the mesh displacements (only tangential movements on the sides and a fixed mesh at the bottom):

SETTINGS	BOUNDARIES 1, 4	BOUNDARY 2
dx	`0`	`0`
dy		`0`

5 On Boundary 3, select **Tangent and normal coord. sys. in deformed mesh** in the **Coordinate system** list. Then click the **Mesh velocity** button and type `u*nx+v*ny` in the **vn** edit field to specify the normal mesh velocity as $\mathbf{u} \cdot \mathbf{n}$.

6 On the **Weak Constr.** tab of the **Boundary Settings** dialog box, clear the **Use weak constraints** check box on Boundaries 1, 2, and 4. The strong constraints that you specified in the previous step are sufficient on these boundaries. Leave the **Use weak constraints** check box selected on Boundary 3.

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

MESH GENERATION

Click the **Initialize Mesh** button on the Main toolbar to initialize the mesh.

COMPUTING THE SOLUTION

1 Open the **Solver Parameters** window from the **Solve** menu.

2 Select **Time dependent** from the **Solver** list.

3 Enter `0:0.1:6` in the **Times** edit field.

4 Type `0.001` in the **Relative tolerance** edit field. This provides a 0.1% relative tolerance, which is one order of magnitude less than the default value.

5 Click the **Time Stepping** tab.

6 Select **Exclude algebraic** in the **Error estimation strategy** list. This excludes the pressure and the moving mesh variables from the error estimation. The equations for those variables do not include time derivatives and become algebraic when solving the equation system using the method of lines.

7 Click **OK**.

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

POSTPROCESSING AND VISUALIZATION

The default plot shows the x-component of the moving mesh deformation, in the spatial frame.

1 To plot the velocity field of the glycerol instead, go to the **Surface** tab in the **Plot Parameters** dialog box and select **Incompressible Navier-Stokes (ns)>Velocity field** from the list of expressions.

2 On the **General** page, clear the **Geometry edges** check box. Click **Apply** to see the plot and use the **Solution at time** list on the **General** tab to browse through the output times.

It is possible to visualize the inclination of the tank by clever use of the deformation plot feature:

3 On the **Deform** page, select the **Deformed shape plot** check box and Set the **Scale factor** to `1`. Enter `Y*sin(phi)` in the **X component** edit field and `-X*sin(phi)` in the **Y component** edit field on the **Subdomain Data** tab.

4 Still on the **Deform** page, click the **Boundary Data** tab. Once again, enter `Y*sin(phi)` in the **X component** edit field and `-X*sin(phi)` in the **Y component** edit field.

5 On the **Boundary** tab, select the **Boundary plot** check box. Enter `1` in the **Expression** edit field. Select to use a **Uniform color** and pick a black color using the **Color** button.

6 To get a more liquid-looking plot, you may want to go to the **Surface** page and set the **Colormap** to **bone**.

7 Click **Apply** to see the plot.

8 To see the waves in action, go to the **Animate** tab and click **Start Animation**, then click **OK**.

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

1 Open the **Domain Plot Parameters** dialog box from the **Postprocessing** menu.

2 On the **Point** tab, select Point 4 from the **Point selection** list.

3 Enter `dy_ale` in the **Expression** edit field, then click **OK** to see the plot.