Deformation of free surface under pressure– 2D model with surface tension

Muriel CARIN
LIMATB, Université de Bretagne-Sud / UEB Centre de recherche C. Huygens, Rue de Saint Maudé, BP 92116 – 56321 LORIENT Cedex - FRANCE - muriel.carin@univ-ubs.fr

Solved with COMSOL Multiphysics 3.5a

Introduction

This model is concerned with the simulation of incompressible Newtonian fluid flow problems with surface tension. The fluid is initially at rest in a square tank. A Gaussian pressure is applied on the free surface which deformed the initially flat surface. This model is developed for a 2D transient analysis. The movement and deformation of the computational domain are accounted for by employing the Arbitrary Lagrangian-Eulerian (ALE) description of the fluid kinematics.

Model definition

Figure 1 shows the initial shape of the tank. Here a 2D model is used but can be treated in 2D axial-symmetry by adapting the model entitled “Square drop oscillation under surface tension - 2D axi-symmetric model” proposed by M Carin in a companion paper.

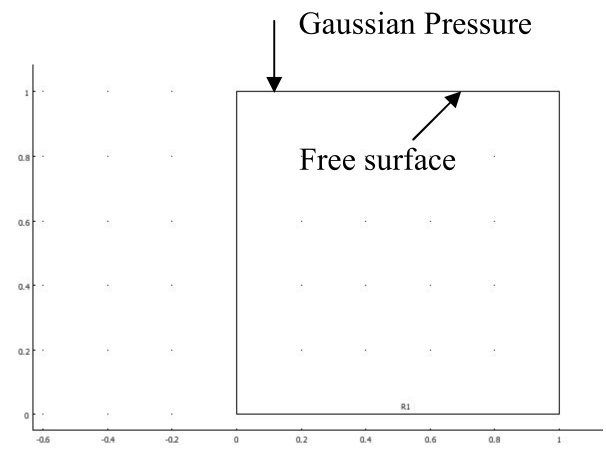

Figure 1: Initial geometry of the tank.

MASS AND MOMENTUM TRANSPORT

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

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

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

where ρ is the density, $\mathbf{u} = (u, v)$ is the fluid velocity, p is the pressure, **I** is the unit diagonal matrix, η is the dynamic viscosity, and **F** is the volume force. In this model, the material properties are: η = 0.2 Pa·s and ρ = 1 kg/m^3. The gravity is neglected here.

With the help of the Moving Mesh (ALE) application mode, these equations are solved 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.

BOUNDARY CONDITIONS

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) on which surface tension acts. In the presence of surface tension phenomena, the boundary condition satisfies:

$$\left[-p\mathbf{I}+\eta\ (\nabla\mathbf{u}+(\nabla\mathbf{u})^{\mathbf{T}}\right].\mathbf{n}=-P_a\mathbf{n}+\Gamma\gamma\mathbf{n}$$

where $\vec{n}$ is the outward unit normal, P_a is the surrounding pressure which is assumed to have a gaussian distribution along the x-axis, Γ is the curvature and γ is the surface tension which is supposed constant.

The main difficulty here is the calculation of the term of surface tension. By applying the weak form to this boundary condition, one obtains:

$$\int_{\Gamma}\tilde{u}\sigma\cdot\mathbf{n}\,dS=\int_{\Gamma}-P_a\tilde{u}\cdot\mathbf{n}\,dS+\int_{\Gamma}\Gamma\gamma\tilde{u}\cdot\mathbf{n}\,dS$$

where σ is the stress tensor and $\tilde{u}$ denotes the test function.

By using the surface divergence theorem [1], one obtains:

$$\int_{\Gamma}\Gamma\gamma\tilde{u}\cdot\mathbf{n}\,dS=\int_{\Gamma}\gamma\nabla_s\tilde{u}\,dS-\int_{\partial\Gamma}\gamma\tilde{u}\cdot\mathbf{n_s}\,dS$$

where ∇_s represents the surface gradient operator. In this model, the contour integral is equal to zero. However, this integral must be included when considering wetting angle.

The model *Sloshing Tank* included in the model library provides a good starting point for specifying the correct boundary condition at the free surface. In particular, getting the correct reaction forces at the free boundary requires that non-ideal weak constraints are used. For more information about weak constraints, see “Using Weak Constraints” on page 358 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. Moreover, including the surface tension terms requires adding some extra contribution directly in the weak form at the boundary level as explained in details below.

Results and discussion

Figure 2 shows the shape of the domain at t=1s. Unless gravity effect is added through a vertical volume force of intensity -rho*g, there is no equilibrium state in this configuration.

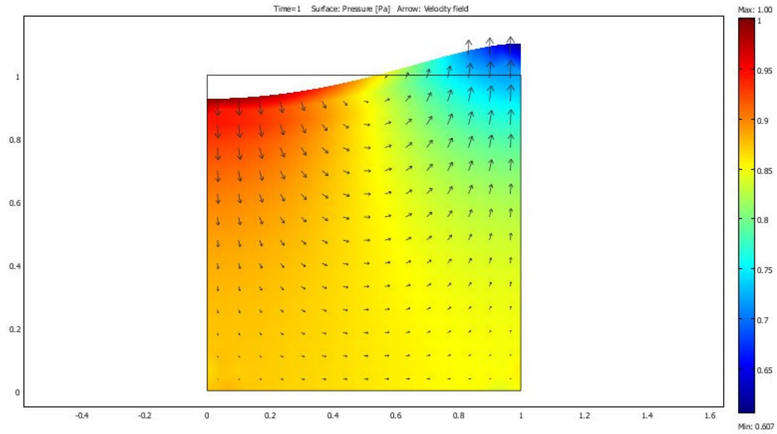

Figure 2: Pressure field and velocity vector field at t = 1 s.

To check the mass conservation, the evolution of the volume is represented in Figure 3. As the fluid is incompressible, the volume must be kept constant. The volume conservation is satisfied here at about 0.001 %.

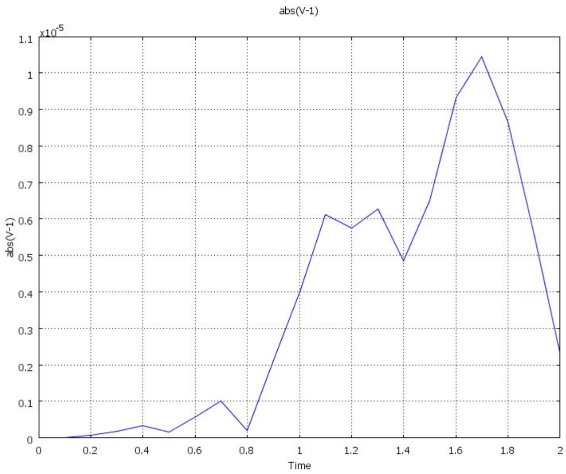

Figure 3: Conservation of volume.

References

1. M.A. Walkley, P.H. Gaskell, P.K. Jimack, M.A. Kelmanson and J.L. Summers, "Finite Element Simulation of Three-Dimensional Free-Surface Flow Problems", *Journal of Scientific Computing,* Vol. 24, No. 2, August 2005 (2005).

Modelling in COMSOL Multiphysics

MODEL NAVIGATOR

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	1
Position:	Base Corner
Position: X	0
Position: Y	0

3 Click **OK** to close the **Rectangle** dialog box.

4 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	1[kg/m^3]	Fluid density
eta	0.2[Pa*s]	Fluid viscosity
nu_surf	0.02[N/m]	Surface tension coefficient
Pmax	1[Pa]	Maximum pressure
rc	1[m]	Pressure distribution parameter

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

3 Type Pgaus in the **Name** field and enter Pmax*exp(-x^2/(2*rc^2)) in the **Expression** field, when done, click **OK**.

4 In the **Options** menu select **Integration Coupling Variables>Subdomain Variables** and choose Subdomain 1.

5 Type V in the **Name** field and enter 1 in the **Expression** field. Select **Frame(ale)** in the **Frame** list. Click OK.

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

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

Settings	**Subdomain 1**
ρ	rho
η	eta
Fx	0
Fy	0

2 In the **Stabilization** tab, clear Streamline diffusion (GLS) and Crosswind diffusion buttons.

Boundary Conditions

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

Settings	**Boundaries 1,2,4**	**Boundary 3**
Boundary type	wall	Open boundary
Boundary condition	slip	Normal stress
f0	0	Pgaus

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

3 In the **Boundary Settings** dialog box, apply the following boundary conditions for the mesh displacements :

Settings	**Boundaries 1,4**	**Boundary 2**
dx	0	0
dy		0

4 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 **u** · **n**.

5 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 specified in the previous step are sufficient on these boundaries. Leave the **Use weak constraints** check box selected on Boundary 3.

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

7 Open the **Equation System > Boundary Settings** from the **Physics** menu. Add to the pressure terms on the **Weak** tab for Boundary 3 :
-nu_surf*test(uTx) for u (5^{th} line)
-nu_surf*test(vTy) for v (6^{th} line)
These expressions represent the terms of the integral $\int_{\Gamma} \gamma \nabla_s \tilde{u}\, dS$ where $\nabla_s \tilde{u}$ is the surface gradient operator expressed in Cartesian coordinates. uTx and vTy represent the tangential derivative variables.

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

COMPUTING THE SOLUTION
1 Open the **Solver Parameters** dialog box from the **Solve** menu.

2 Enter 0:0.1:2 in the **Times** edit field on the **General** page.

3 Select the **Direct PARDISO** solver from the **linear system solver**.

4 Click the **Time Stepping** tab.

5 Select **Exclude algebraic** from the **Error estimation strategy** list. This excludes the pressure and the moving mesh variables from the error estimation.

6 Click **OK**.

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

2 To plot the pressure field, go to the **Surface** tab in the **Plot Parameters** dialog box and select **Incompressible Navier-Stokes (ns)>Pressure** from the list of expressions. To visualize the deforming mesh, select Wireframe in the Fill style list.

3 In the **General** tab, deselect the **Auto** check box of the **Element refinement** and type 1 in the element refinement edit field.

4 In the **Arrow** tab, select **Arrow Plot** and choose **Velocity field** in the **predefined quantities**. Change **color** in Black.

5 In the **Animate** tab, select **Use camera settings from main window**. Click the **Start Animation** button, then click **OK** to close the **Plot Parameters** dialog box.

6 To check the volume conservation, from the **postprocessing** menu, select Do**main** Plot parameters.

7 Click the Point tab and select any point.

8 In the **Expression** edit field, type abs(V-1) to visualize the relative error of volume and click **OK**.