Model created in COMSOL Multiphysics 6.4

Dam Breaking on a Column, Level Set

Introduction

Wave impact problems are important in engineering of structures, for example in locations where the event of tsunami is probable. Predicting the forces of waves acting on objects can be crucial for offshore structures and structures placed near water. If the structure is subjected to high waves, flooding must also be accounted for. One of the simplest and widely used ways used to model this kind of problems is the shallow water equations system.

In this transient study, the Two-Phase Flow, Level Set interface is used instead to model the impact of a water wave on a column. A body of water with a height of 0.3 m is initially contained behind a gate. At the onset of the simulation, the gate is suddenly released and the body of water forms a wave moving toward the structure. After impacting, the water continues forward movement until it is reflected at the wall of the tank and impinges second time on the column. The pressure force on the column is computed and can be compared with the experimental results available in Ref. 1 as well as with the results obtained using the Shallow Water Equations, Time Explicit interface in Dam Breaking on a Column, Shallow Water Equations.

Model Definition

The geometry and initial configuration of the experiment are depicted in Figure 1. A 1.60 m long, 0.61 m wide, and 0.60 m high tank is used. A 0.40 m long, 0.61 m wide, and 0.30 m high bulk of water is initially contained behind a gate which is instantly released at the simulation onset. A tall solid column with 0.12 m wide square base is placed inside the tank 0.50 m downstream of the gate and 0.25 m from one of the sidewalls. The experimental facility did not allow for a complete drainage of the tank and a thin layer of water of approximately 0.01 m has to also be accounted for.

The pressure force per unit length on a boundary can be obtained using an integration operator,

$$\mathbf{F}_{\mathbf{p}} = \int_{\Gamma} p\mathbf{n}d\Gamma$$

where Γ represents the front and back boundaries of the column.

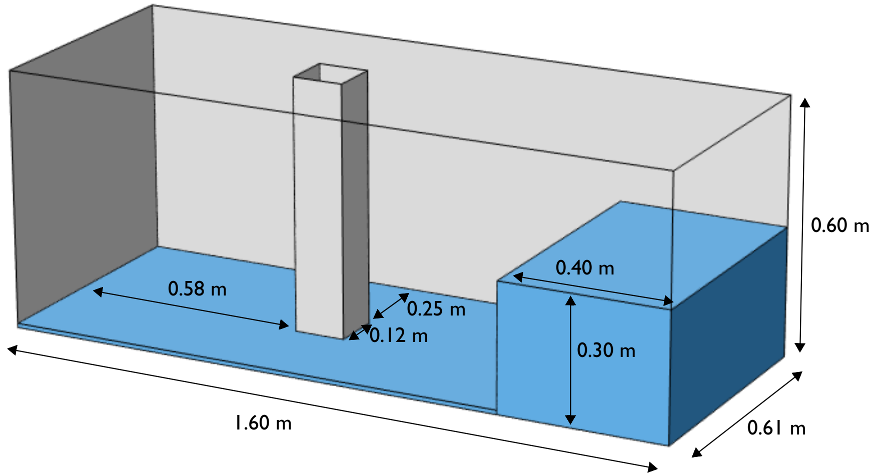


Figure 1: Geometry and initial water configuration.

Results and Discussion

Figure 2 shows the position of the free surface at various times. After the release of the gate, the body of water collapses due to gravity and forms a wave moving toward the column. Upon impacting the structure, the wave is divided and its center part rides up the column's upstream face. The sides of the wave front rejoin in the wake downstream the structure and later become reflected from the downstream wall of the tank. The wave is weakened after the reflection and impinges again on the downstream face of the column. The wave then continues toward the upstream wall, while slowly decaying.

The net y-component of the pressure force acting on the column is plotted in Figure 3. The computed force captures the impact of the evolving wave on the front and rear parts of the structure with maxima at t = 0.35 s and t = 1.5 s, respectively. Compared with the measured forces reported in Ref. 1, the agreement is good. The results are better than the ones obtained with the shallow water equations in Dam Breaking on a Column, Shallow Water Equations.

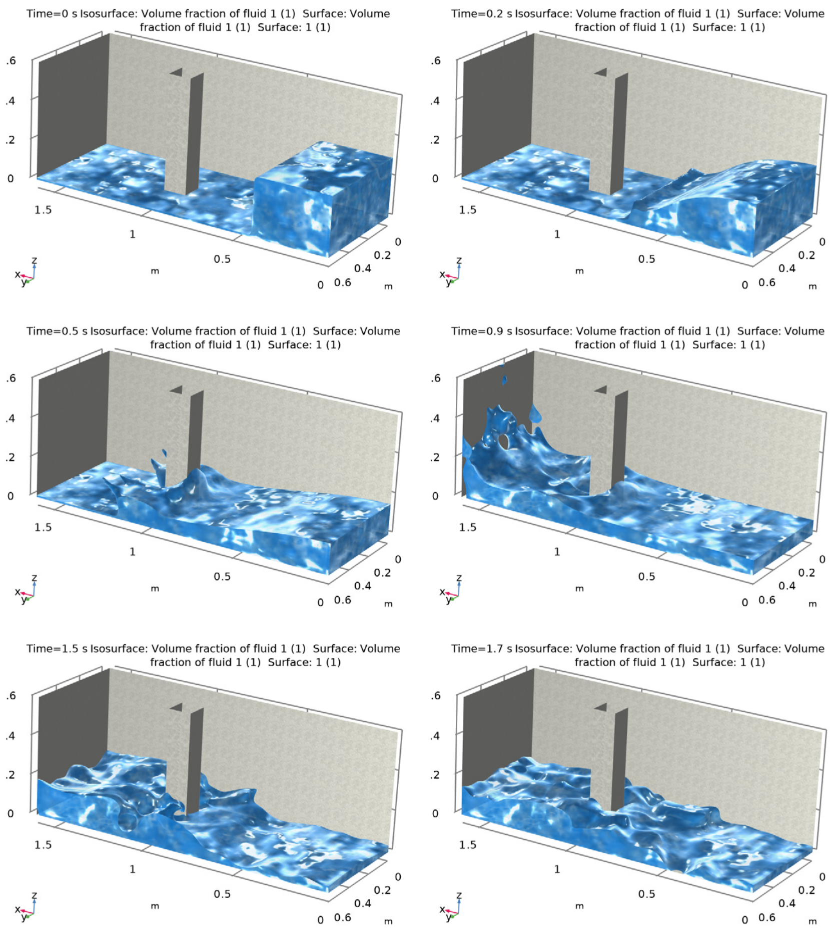


Figure 2: Water free surface at t = 0 s, 0.2 s, 0.5 s, 0.9 s, 1.5 s, and 1.7 s.

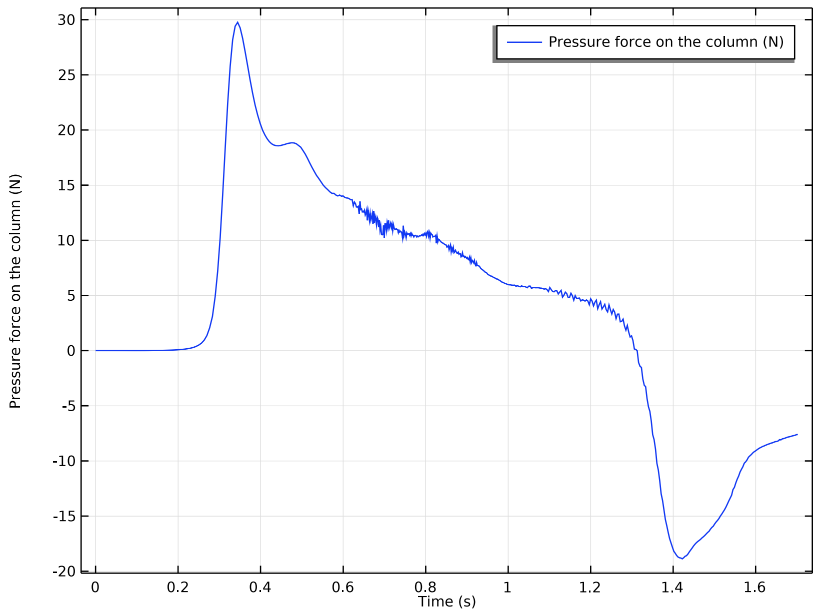


Figure 3: Force in the y direction acting on the structure.

Notes About the COMSOL Implementation

This model uses the Block Navier–Stokes preconditioner instead of the default preconditioner for GMRES when solving the Navier–Stokes equations. This preconditioner can uncouple the pressure and velocity equations when solving, and may provide faster solutions for large time dependent models with incompressible flow and high Reynolds numbers (low viscosities).

Reference

1. P.E. Raad and R. Bidoae, "The three-dimensional Eulerian-Lagrangian marker and micro cell method for the simulation of free surface flows," *J. Comput. Phys.*, vol. 203, pp. 668–699, 2005.

Application Library path: `CFD_Module/Multiphase_Flow/dam_break_column_ls`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Fluid Flow** > **Multiphase Flow** > **Two-Phase Flow, Level Set** > **Laminar Flow**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics** > **Time Dependent with Phase Initialization**.

6 Click **Done**.

GEOMETRY 1

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `1.6`.

4 In the **Depth** text field, type `0.61`.

5 In the **Height** text field, type `0.6`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	**Thickness (m)**
`Layer 1`	`0.01`

Block 2 (blk2)

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.4`.

4 In the **Depth** text field, type `0.61`.

5 In the **Height** text field, type `0.3`.

Block 3 (blk3)

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.12`.

4 In the **Depth** text field, type `0.12`.

5 In the **Height** text field, type `0.6`.

6 Locate the **Position** section. In the **x** text field, type `0.9`.

7 In the **y** text field, type `0.25`.

8 Locate the **Assigned Attributes** section. Select the **Construction geometry** checkbox.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **blk1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the objects **blk2** and **blk3** only.

6 Select the **Keep objects to subtract** checkbox.

Extract 1 (extract1)

1 In the **Geometry** toolbar, click **Extract**.

2 In the **Settings** window for **Extract**, locate the **Entities or Objects to Extract** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 On the object **dif1**, select Domain 2 only.

5 From the **Input object handling** list, choose **Create remainder object**.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **blk2** and **extract1(1)** only.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** checkbox.

5 Click **Build All Objects**.

DEFINITIONS

Integration 1 (intop1)

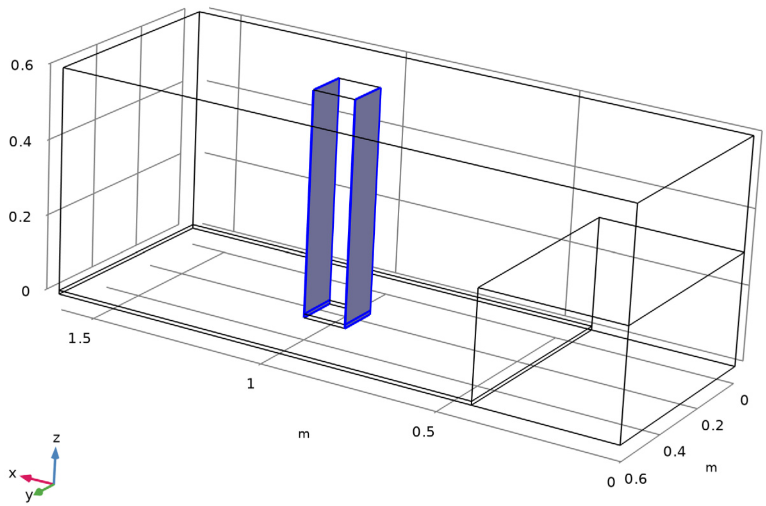
1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

5 Select Boundaries 15, 17, 21, and 22 only.



Global Variable Probe 1 (var1)

1 In the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type `Fp` in the **Variable name** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `intop1(p*spf.nxmesh)`.

4 Select the **Description** checkbox. In the associated text field, type `Pressure force on the column`.

MULTIPHYSICS

Two-Phase Flow, Level Set 1 (tpf1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Multiphysics** click **Two-Phase Flow, Level Set 1 (tpf1)**.

2 In the **Settings** window for **Two-Phase Flow, Level Set**, locate the **Material Properties** section.

3 Click **Add Multiphase Material**.

MATERIALS

Phase 1 (mpmat1.phase1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** > **Multiphase Material 1 (mpmat1)** click **Phase 1 (mpmat1.phase1)**.

2 In the **Settings** window for **Phase**, locate the **Link Settings** section.

3 Click **Add Material from Library** . This button is found when expanding the options next to the **Material** list.

ADD MATERIAL TO PHASE 1 (MPMAT1.PHASE1)

1 Go to the **Add Material to Phase 1 (mpmat1.phase1)** window.

2 In the tree, select **Built-in** > **Water, liquid**.

3 Click **Add Material**.

MATERIALS

Phase 2 (mpmat1.phase2)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** > **Multiphase Material 1 (mpmat1)** click **Phase 2 (mpmat1.phase2)**.

2 In the **Settings** window for **Phase**, locate the **Link Settings** section.

3 Click **Add Material from Library** . This button is found when expanding the options next to the **Material** list.

ADD MATERIAL TO PHASE 2 (MPMAT1.PHASE2)

1 Go to the **Add Material to Phase 2 (mpmat1.phase2)** window.

2 In the tree, select **Built-in** > **Air**.

3 Click **Add Material**.

MATERIALS

1 Click the **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.

3 Click **OK**.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.

3 Select the **Include gravity** checkbox.

4 Click to expand the **Advanced Settings** section. Find the **Linear solver** subsection. Select the **Use Block Navier–Stokes preconditioner** checkbox.

Pressure Point Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

2 Select Point 1 only.

3 In the **Settings** window for **Pressure Point Constraint**, locate the **Pressure Constraint** section.

4 Clear the **Compensate for hydrostatic pressure** checkbox.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 Clear the **Compensate for hydrostatic pressure** checkbox.

LEVEL SET (LS)

Level Set Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Level Set (ls)** click **Level Set Model 1**.

2 In the **Settings** window for **Level Set Model**, locate the **Level Set Model** section.

3 In the γ text field, type `10`.

Initial Values, Fluid 2

1 In the **Model Builder** window, click **Initial Values, Fluid 2**.

2 Select Domain 2 only.

MULTIPHYSICS

Wetted Wall 1 (ww1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Multiphysics** click **Wetted Wall 1 (ww1)**.

2 In the **Settings** window for **Wetted Wall**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Finer**.

4 Click **Build All**.

STUDY 1

Step 2: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 2: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,0.05,1.7)`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 2** node, then click **Velocity Field (comp1.u)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 From the **Method** list, choose **Manual**.

6 In the **Scale** text field, type `10`.

7 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** click **Time-Dependent Solver 1**.

8 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

9 Find the **Algebraic variable settings** subsection. In the **Fraction of initial step for Backward Euler** text field, type `1`.

10 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

1 In the **Model Builder** window, click **Results**.

2 In the **Settings** window for **Results**, locate the **Update of Results** section.

3 Select the **Only plot when requested** checkbox.

Volume Fraction of Fluid 1 (ls)

1 In the **Model Builder** window, under **Results** click **Volume Fraction of Fluid 1 (ls)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 Clear the **Plot dataset edges** checkbox.

Slice 1

1 In the **Model Builder** window, expand the **Volume Fraction of Fluid 1 (ls)** node.

2 Right-click **Slice 1** and choose **Delete**. Click **Yes** to confirm.

Material Appearance 1

1 Right-click **Isosurface 1** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Water**.

Transparency 1

1 Right-click **Isosurface 1** and choose **Transparency**.

2 In the **Settings** window for **Transparency**, locate the **Transparency** section.

3 Find the **Transparency** subsection. Set the **Transparency** value to **0.15**.

Surface 1

1 In the **Model Builder** window, right-click **Volume Fraction of Fluid 1 (ls)** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Level Set** > **ls.Vf1 - Volume fraction of fluid 1 - 1**.

3 Click to expand the **Range** section. Select the **Manual data range** checkbox.

4 In the **Minimum** text field, type `0.5`.

5 In the **Maximum** text field, type `1`.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 In the list, choose **6**, **12**, and **13**.

5 Click **Remove from Selection**.

Material Appearance 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Water**.

Transparency 1

1 Right-click **Surface 1** and choose **Transparency**.

2 In the **Settings** window for **Transparency**, locate the **Transparency** section.

3 Find the **Transparency** subsection. Set the **Transparency** value to **0.15**.

Surface 2

1 In the **Model Builder** window, right-click **Volume Fraction of Fluid 1 (ls)** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `1`.

Selection 1

1 Right-click **Surface 2** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `2, 3, 5, 10, 11, 15-24` in the **Selection** text field.

5 Click **OK**.

Material Appearance 1

1 In the **Model Builder** window, right-click **Surface 2** and choose **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Steel (scratched)**.

5 In the **Volume Fraction of Fluid 1 (ls)** toolbar, click **Plot**.

STUDY 1

In the **Study** toolbar, click **Compute**.

RESULTS

Volume Fraction of Fluid 1 (ls)

1 In the **Model Builder** window, under **Results** click **Volume Fraction of Fluid 1 (ls)**.

2 In the **Volume Fraction of Fluid 1 (ls)** toolbar, click **Plot**.

3 In the **Model Builder** window, click **Volume Fraction of Fluid 1 (ls)**.

4 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

5 From the **Time (s)** list, choose **0.9**.

6 In the **Volume Fraction of Fluid 1 (ls)** toolbar, click **Plot**.