Created in COMSOL Multiphysics 5.6

The Shallow Water Equations

Introduction

The shallow water equations are frequently used for modeling both oceanographic and atmospheric fluid flow. Models of such systems lead to the prediction of areas eventually affected by pollution, coastal erosion, and polar ice-cap melting.

Comprehensive modeling of such phenomena using physical descriptions such as the Navier-Stokes equations can often be problematic, due to the scale of the modeling domains as well as through resolving free surfaces. The shallow water equations, of which there are a number of representations, provide an easier description of such phenomena.

This 1D model investigates the settling of a wave over a variable bed as a function of time. The initial wave and the shape of the bed are represented by mathematical relations so that it is easy to change parameters such as the wave amplitude or the bed's shape.

This example uses the *Saint-Venant's* shallow water equations, which are the following:

$$\frac{\partial z}{\partial t} + \nabla \cdot (zv) = 0$$

and

$$\frac{\partial v}{\partial t} + (v \cdot \nabla)v = -g\nabla z_s + z^{-1}\nabla \cdot (z\nu\nabla v)$$

where z is the thickness of the water layer (m), v is the velocity (m/s), g is the gravity constant (m/s^2), and ν is the kinematic viscosity (m^2/s). The definition of the thickness of the water layer, z, is $z_\mathrm{s} - z_\mathrm{f}$, where z_s and z_f are the measures in Figure 1 below. For further details, see Ref. 1.

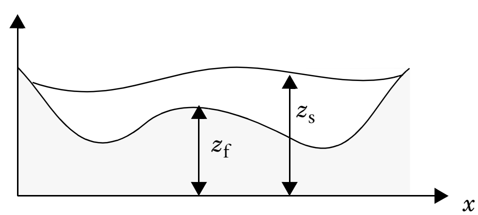

Figure 1: Representative vertical section through the fluid domain showing the bed of a lake and the water surface.

Artificial Stabilization

With time, the flow develops discontinuities known as hydraulic jumps. Use artificial stabilization to replace the jumps by steep fronts that can be resolved on the grid. Small amplitude waves on still water of depth z move with velocity $\sqrt{gz}$. The maximal propagation velocity is $v_{\mathrm{phase}} = |v| + \sqrt{gz}$ for water waves.

Stabilize the solution by adding artificial viscosity chosen to make the cell Reynolds number of order unity. To do so, add the term $\mathrm{tune}\, v_{\mathrm{phase}} h$ to the kinematic viscosity ν. Here, tune is an $O(1)$ tuning parameter and h is the local element size. You add the contribution to the divergence term of the conservation law so that it does not affect the shock speeds. The modification is first order in element size.

Model Definition

This application studies a simple example of shallow water in a channel with bottom topography shown in Figure 1. Notice the difference in scale between the x and y directions.

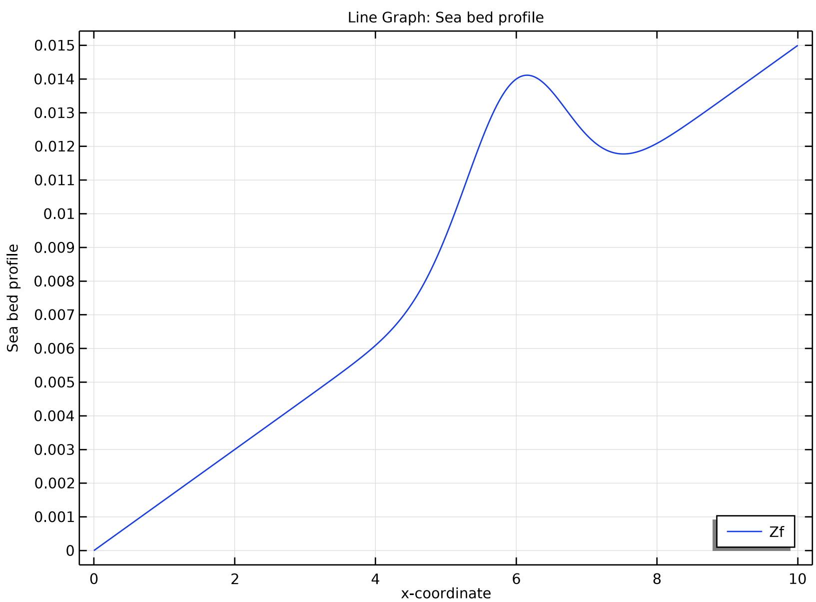

Figure 2: Sea bed profile, z_{f}, used in the model.

Constraints ($v = 0$) are implemented at both ends, while the physics are described by the equations above. The initial condition is a wave profile, which the following expression defines:

$$z_0 = 2 \cdot 10^{-2} - z_\mathrm{f} + 5 \cdot 10^{-3} e^{-(x-3)^2}$$

where z_f is the analytical expression for the sea bed profile (see Figure 2). The elevation of the water surface is $z + z_\mathrm{f}$, while Figure 3 shows $z_0 + z_\mathrm{f}$.

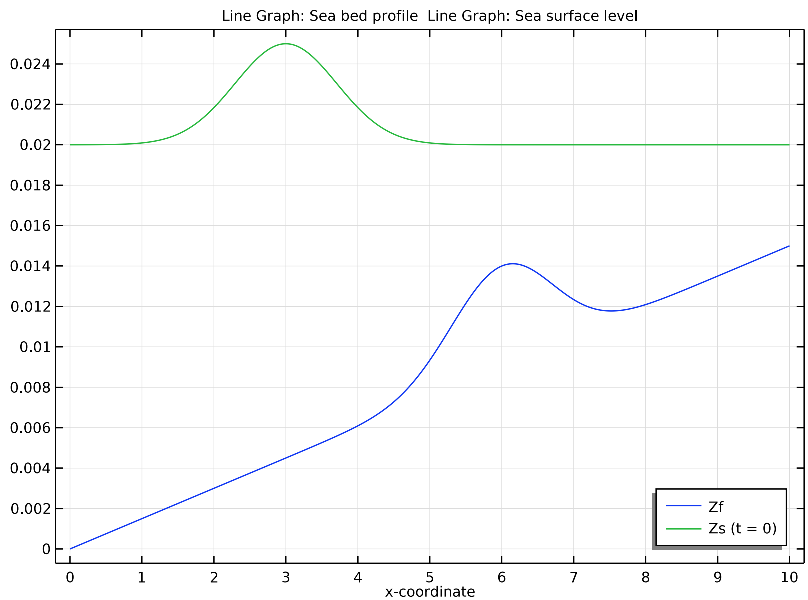

Figure 3: The initial water surface profile, $z_0 + z_\mathrm{f}$, and the sea bed profile, z_f.

Results and Discussion

The simulation runs for 60 seconds. Figure 4 shows the water surface and slope of the sea bed at six output times toward the beginning of the simulation.

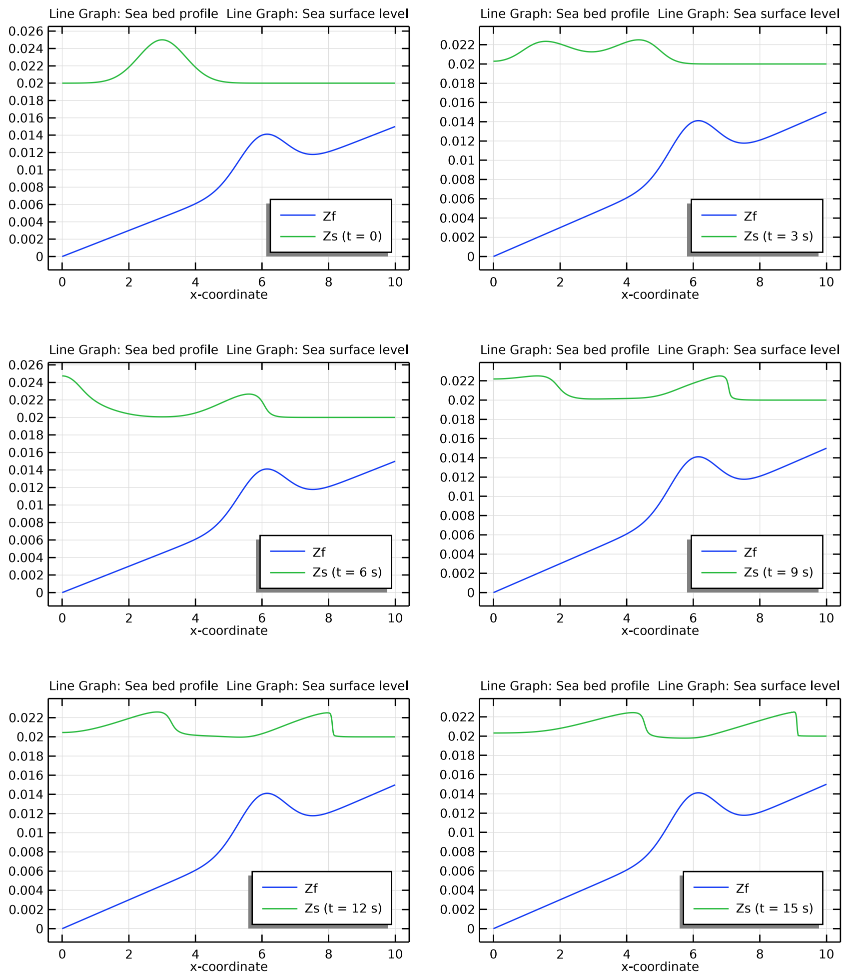

Figure 4: The water level and the slope of the sea bed at six output times. Time spans from t = 0 to t = 15 at steps of 3 seconds.

The simulation clearly shows the influence of the topography of the sea bed on the elevation of the water surface. Another interesting visualization of the results is an animation, which is easy to create using COMSOL Multiphysics.

Notes About the COMSOL Implementation

The modeling procedure is straightforward using the General Form PDE interface with two dependent variables: `Z` and `V`. It is easy to define expressions, such as the one that describes the initial wave profile, z_0, as a variable in the model.

Reference

1. O. Pironneau, *Finite Element Methods for Fluids*, John Wiley & Sons, 1989.

Application Library path: `COMSOL_Multiphysics/Equation_Based/shallow_water_equations`

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 **1D**.

2 In the **Select Physics** tree, select **Mathematics>PDE Interfaces>General Form PDE (g)**.

3 Click **Add**.

4 Click **Add Dependent Variable**.

5 In the **Dependent variables** table, enter the following settings:

Z
V

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Time Dependent**.

8 Click **Done**.

ROOT

1 In the **Model Builder** window, click the root node.

2 In the root node's **Settings** window, locate the **Unit System** section.

3 From the **Unit system** list, choose **None**.

Because the dependent variables, Z and V, are of different dimensions, it is convenient to switch off unit support and keep track of the units by hand instead.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
nu1	1e-6	1E-6	Kinematic viscosity (m^2/s)
x0	6	6	Sea bed ridge position (m)
a	0.005	0.005	Sea bed ridge height (m)
k1	0.0015	0.0015	Sea bed slope parameter
tune	0.1	0.1	Tuning parameter

GEOMETRY 1

Interval 1 (i1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Interval**.

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

3 In the table, enter the following settings:

Coordinates
0
10

4 Click **Build All Objects**.

DEFINITIONS

Variables 1

1 In the **Home** toolbar, click a= **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	**Expression**	**Description**
Zf	a*exp(-(x-x0)^2)+k1*x	Sea bed profile
dZfdx	d(Zf,x)	Sea bed profile, x derivative
Zs	Z+Zf	Sea surface level
Z0	0.02-Zf+0.005*exp(-(x-3)^2)	Initial water depth profile
vphase	abs(V)+sqrt(g_const*Z)	Maximal wave propagation velocity
nu	nu1+vphase*h*tune	Effective kinematic viscosity

Here, `g_const` is a predefined constant for the acceleration of gravity; with unit support turned off, it just takes the numeric value in SI units.

GENERAL FORM PDE (G)

General Form PDE 1

1 In the **Model Builder** window, under **Component 1 (comp1)>General Form PDE (g)** click **General Form PDE 1**.

2 In the **Settings** window for **General Form PDE**, locate the **Conservative Flux** section.

3 In the Γ text-field array, type `V*Z` on the first row.

4 In the Γ text-field array, type `-nu*Vx` on the second row.

5 Locate the **Source Term** section. In the f text-field array, type `0` on the first row.

6 In the f text-field array, type `-g_const*(Zx+dZfdx)-V*Vx+nu*Vx*Zx/Z` on the second row.

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 In the Z text field, type `Z0`.

Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Constraint**.

2 Click in the **Graphics** window and then press Ctrl+A to select both boundaries.

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

4 In the R text-field array, type `-V` on the second row.

MESH 1

Edge 1

In the **Mesh** toolbar, click **Edge**.

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.05`.

5 Click **Build All**.

STUDY 1

Step 1: Time Dependent

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

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

3 In the **Output times** text field, type `range(0,60)`.

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `1e-5`.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Absolute Tolerance** section.

4 From the **Tolerance method** list, choose **Manual**.

5 In the **Absolute tolerance** text field, type `1e-7`.

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

RESULTS

1D Plot Group 1

1 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

2 From the **Position** list, choose **Lower right**.

Line Graph 1

1 In the **Model Builder** window, expand the **1D Plot Group 1** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 From the **Time selection** list, choose **From list**.

5 In the **Times (s)** list, select **0**.

6 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>Zf - Sea bed profile**.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
Zf

10 In the **1D Plot Group 1** toolbar, click **Plot**.

The most interesting part of the results is obtained by looking at the global variable *Z*s, corresponding to the surface topography, at different times compared to the topography of the bottom.

Line Graph 2

1 Right-click **Results>1D Plot Group 1>Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions> Variables>Zs - Sea surface level**.

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

Legends
Zs (t = 0)

4 In the **1D Plot Group 1** toolbar, click **Plot**.

5 Locate the **Data** section. In the **Times (s)** list, select **3**.

6 Locate the **Legends** section. In the table, enter the following settings:

Legends
Zs (t = 3 s)

7 In the **1D Plot Group 1** toolbar, click **Plot**.

8 Locate the **Data** section. In the **Times (s)** list, select **6**.

9 Locate the **Legends** section. In the table, enter the following settings:

Legends
Zs (t = 6 s)

10 In the **1D Plot Group 1** toolbar, click **Plot**.

11 Locate the **Data** section. In the **Times (s)** list, select **9**.

12 Locate the **Legends** section. In the table, enter the following settings:

Legends
Zs (t = 9 s)

13 In the **1D Plot Group 1** toolbar, click **Plot**.

14 Locate the **Data** section. In the **Times (s)** list, select **12**.

15 Locate the **Legends** section. In the table, enter the following settings:

Legends
Zs (t = 12 s)

16 In the **1D Plot Group 1** toolbar, click **Plot**.

17 Locate the **Data** section. In the **Times (s)** list, select **15**.

18 Locate the **Legends** section. In the table, enter the following settings:

Legends
Zs (t = 15 s)

19 In the **1D Plot Group 1** toolbar, click **Plot**.