Created in COMSOL Multiphysics 6.2

# The KdV Equation and Solitons

## *Introduction*

The Korteweg-de Vries (KdV) equation, formulated in 1895 by Korteweg and de Vries, models water waves. It contrasts sharply to the Burgers equation because it introduces no dissipation and the waves travel seemingly forever. In 1965, Zabusky and Kruskal named such waves *solitons.*

The KdV equation with boundary conditions and initial value is formulated as

$$\begin{aligned} u_t + u_{xxx} &= 6uu_x \qquad \text{in } \Omega = [-8, 8] \\ u(-8, t) &= u(8, t), \qquad \text{periodic} \\ u(x, 0) &= -6\,\text{sech}^2(x) \end{aligned}$$

The equation models the steepening and dispersion of wavefronts but does not support a train of simple harmonic waves. Such trains comprise the wave crests normally associated with the ocean: simply a momentary constructive interference of contributing waves moving at different speeds. However, the equation does support solitons, single “humps” that travel without changing shape or speed for unexpectedly long distances.

Indeed, Perry and Schimke (Ref. 2) concluded from shipboard oceanographic measurements that bands of choppy water in the Andaman Sea, which lies east of the Bay of Bengal and west of Burma (Union of Myanmar) and Thailand, are associated with large-amplitude oceanic internal waves. Satellite images have since clarified that these waves originate on shallow banks on a layer between warm and cool water. Further, Osborne and Burch (Ref. 1) analyzed oceanographic data in an effort to assess the forces of underwater current fluctuations associated with such waves on offshore drilling rigs. They concluded that the visually observed roughness bands are caused by internal solitons that follow the KdV equation (Ref. 3).

A more recent development is the application of the KdV equation to another type of waves — light waves. Today solitons have their primary practical application in optical fibers. Specifically, a fiber’s linear dispersion properties level out a wave while the nonlinear properties give a focusing effect. The result is a very stable, long-lived pulse (Ref. 3). It is amazing that researchers have discovered a formula for such waves:

$$u = \frac{v}{\left[2\cosh^2\left(\frac{1}{2}\sqrt{v}\right)(x - vt - f)\right]}$$

This equation says that the pulse speed is what determines the pulse amplitude and the pulse width. The following simulation illustrates this effect. An initial pulse, which does not conform to the formula, immediately breaks down into two pulses of different amplitudes and speeds. The two new pulses follow the formula and thus can travel forever. While the formula does not reveal how solitons interact, the simulation shows that they can collide and reappear, seemingly unchanged, just as linear waves do, another counterintuitive observation that is difficult to observe without predictions by computing.

## *Model Definition*

In the model, the term $uu_x$ describes the focusing of a wave and $u_{xxx}$ refers to its dispersion. The balancing of these two terms permits waves to travel with their shape unchanged.

Because COMSOL Multiphysics does not evaluate third derivatives directly, you rewrite the original equation above as a system of two variables to solve it:

$$u_{1t} + u_{2x} = 6u_1u_{1x}$$
$$u_{1xx} = u_2$$

Using the General Form PDE interface, you need to define two dependent variables, `u1` and `u2`, and identify the $d_a$, $\Gamma$, and $F$ coefficients in the following equation:

$$d_a\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot \Gamma = F$$

- Only the first equation has a time derivative, and it is with respect to $u_1$, so only $d_a(1, 1)$ is $1$; the other three components are zero.
- The divergence is a space derivative with respect to $x$. This means that the $\Gamma$ component from the first equation is $u_2$, which you type as `u2`. The $\Gamma$ component from the second equation is $u_{1x}$, which you express using COMSOL Multiphysics syntax as `u1x`.
- The $F$ term components are the right-hand side of the equations: $F_1$ is $6u_1u_{1x}$ (type `6*u1*u1x`), and $F_2$ is $u_2$ (type `u2`).

The initial condition for $u_1$ uses a hyperbolic cosine function to provide an interesting waveform to start with. For $u_2$, you must provide the second space derivative of this function to provide consistent initial conditions.

The boundary conditions are periodic boundary conditions: the solution at one end is always identical to the one at the other end of the domain.

## *Results*

The following plot shows how solitons collide and reappear with their shape intact.

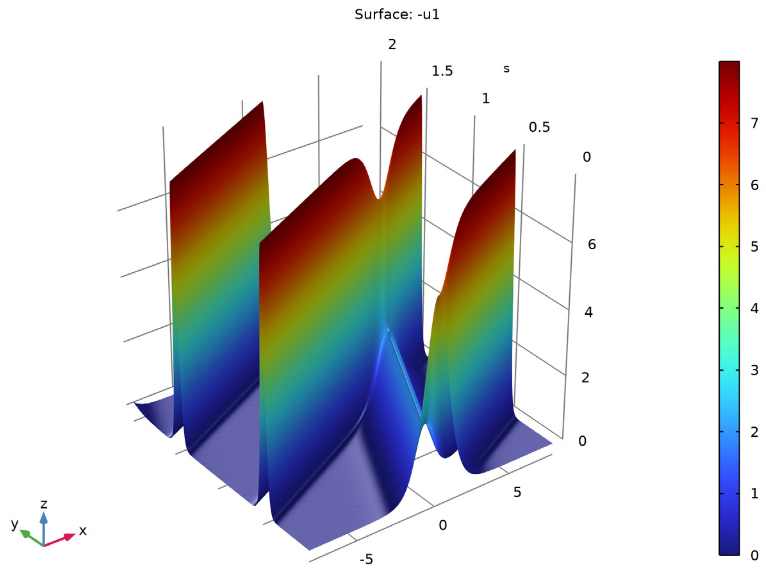



*Figure 1: Solution visualizing a soliton collision.*

## *References*

1. A.R. Osborne and T.L. Burch, “Internal Solitons in the Andaman Sea”, *Science*, vol. 208, no. 4443, pp. 451–460, 1980.

2. R.B. Perry and G.R. Schimke, “Large-Amplitude Internal Waves Observed Off the Northwest Coast of Sumatra”, *J. Geophys. Res.*, vol. 70, no. 10, pp. 2319–2324, 1965.

3. G. Strang, *Applied Mathematics*, Wellesley-Cambridge, 1986.

**Application Library path:** `COMSOL_Multiphysics/Equation_Based/kdv_equation`

## *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 In the **Number of dependent variables** text field, type 2.

5 Click **Study**.

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

7 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**.

Keeping track of units is not important in this model; by turning off unit support, you avoid the need to specify dimensions for equation coefficients and coordinates to get rid of unit warnings.

### 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 |
|---|
| -8 |
| 8 |

### GENERAL FORM PDE (G)

*Periodic Condition 1*

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **General Form PDE (g)** and choose **Periodic Condition**.

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

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

*General Form PDE 1*

1 In the **Model Builder** window, 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 `u2` on the first row.

4 In the Γ text-field array, type `u1x` on the second row.

5 Locate the **Source Term** section. In the $f$ text-field array, type `6*u1*u1x` on the first row.

6 In the $f$ text-field array, type `u2` on the second row.

7 Locate the **Damping or Mass Coefficient** section. In the $d_a$ text-field array, type `0` in the second column of 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 $u1$ text field, type `-6*sech(x)^2`.

4 In the $u2$ text field, type `-24*sech(x)^2*tanh(x)^2+12*sech(x)^2*(1-tanh(x)^2)`.

## 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 `0.1`.

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(0,0.0025,2)`.

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

5 In the **Relative tolerance** text field, type `3e-6`.

*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 **Time Stepping** section.

4 From the **Method** list, choose **Generalized alpha**.

The **Generalized alpha** time stepper is well suited for wave problems. For an accurate solution, use tighter tolerance settings.

5 Click to expand the **Absolute Tolerance** section. From the **Tolerance method** list, choose **Manual**.

6 In the **Absolute tolerance** text field, type `3e-7`.

7 Locate the **Time Stepping** section. In the **Amplification for high frequency** text field, type `0.98`.

8 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** node, then click **Fully Coupled 1**.

9 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

10 In the **Tolerance factor** text field, type `0.1`.

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

## RESULTS

*General Form PDE*

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

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

3 In the **Times (s)** list, select **0.025**.

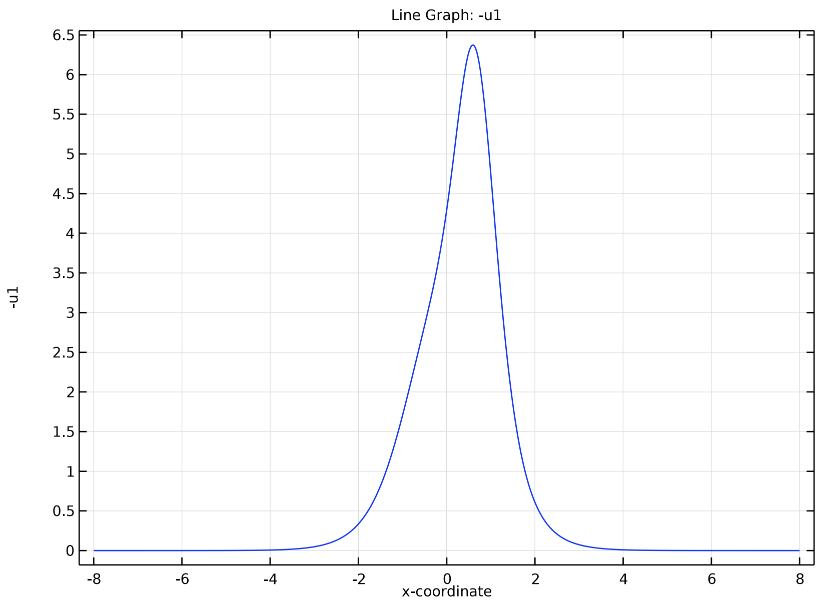
*Line Graph 1*

1 In the **Model Builder** window, expand the **General Form PDE** node, then click **Line Graph 1**.

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

**3** In the **Expression** text field, type `-u1`.

**4** In the **General Form PDE** toolbar, click **Plot**.




*The solution to the KdV equation at 0.025 s.*

To visualize the solution, extrude results along the time axis.

### *Parametric Extrusion 1D 1*

In the **Results** toolbar, click **More Datasets** and choose **Parametric Extrusion 1D**.

### *2D Plot Group 2*

In the **Results** toolbar, click **2D Plot Group**.

### *Surface 1*

**1** Right-click **2D Plot Group 2** and choose **Surface**.

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

**3** In the **Expression** text field, type `-u1`.

### *Height Expression 1*

**1** Right-click **Surface 1** and choose **Height Expression**.

**2** Click the **Zoom Extents** button in the **Graphics** toolbar.

Compare with the plot shown in Figure 1.