# The Level Set Method

Fluid flow with moving interfaces or boundaries occur in a number of different applications, such as fluid-structure interaction, multiphase flows, and flexible membranes moving in a liquid. One way to track moving interfaces is to use a *level set method*. A certain contour line of the globally defined function, the level set function, then represents the interface between phases. With the Level Set modeling interface you can move the fluid-fluid interface within any velocity field.

### THE LEVEL SET METHOD

The level set method is a technique to represent moving interfaces or boundaries using a fixed mesh. It is useful for problems where the computational domain can be divided into two domains separated by an interface. Each of the two domains can consist of several parts. Figure 7-3 shows an example where one of the domains consists of two separated parts. The interface is represented by a certain level set or isocontour of a globally defined function, the level set function $\phi$. In COMSOL Multiphysics, $\phi$ is a smooth step function that equals zero in a domain and one in the other. Across the interface, there is a smooth transition from zero to one. The interface is defined by the 0.5 isocontour, or the level set, of $\phi$. Figure 7-4 shows the level set representation of the interface in Figure 7-3.

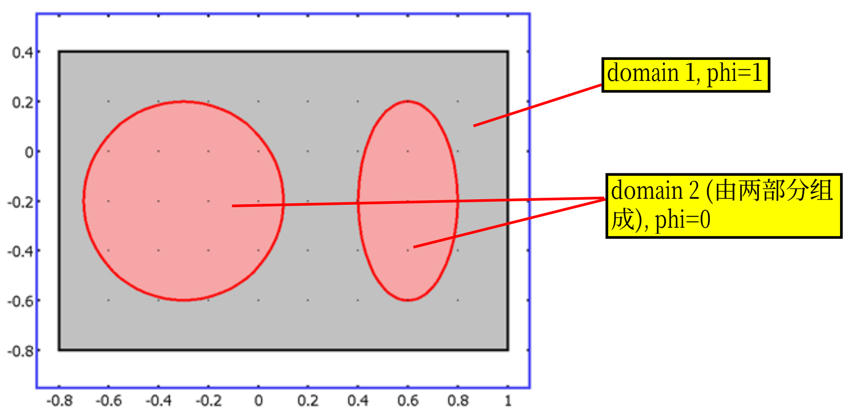


*Figure 7-3: Example of two domains divided by an interface. In this case, one of the domain consists of two parts. Figure 7-4 shows the corresponding level set representation.*

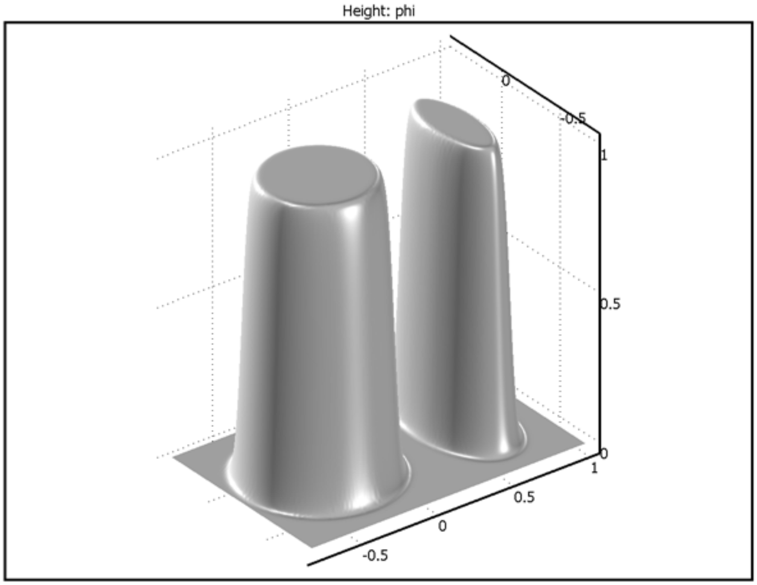


*Figure 7-4: Surface plot of the level set function corresponding to Figure 7-3.*

The modeling interface solves the following equation in order to move the interface with the velocity field **u**:

$$\frac{\partial\phi}{\partial t} + \mathbf{u}\cdot\nabla\phi = \gamma\nabla\cdot\left(\varepsilon\nabla\phi - \phi(1-\phi)\frac{\nabla\phi}{|\nabla\phi|}\right) \tag{7-30}$$

The terms on the left-hand side give the correct motion of the interface, while those on the right-hand side are necessary for numerical stability. The parameter, $\varepsilon$, determines the thickness of the region where $\phi$ goes smoothly from zero to one and is typically of the same order as the size of the elements of the mesh. By default, $\varepsilon$ is constant within each domain and equals the largest value of the mesh size, $h$, within the domain. The parameter $\gamma$ determines the amount of reinitialization or stabilization of the level set function. It needs to be tuned for each specific problem. If $\gamma$ is too small, the thickness of the interface might not remain constant, and oscillations in $\phi$ may appear because of numerical instabilities. On the other hand, if $\gamma$ is too large, the interface moves incorrectly. A suitable value for $\gamma$ is the maximum magnitude of the velocity field **u**.

### CONSERVATIVE AND NON-CONSERVATIVE FORM

If the velocity is divergence free, that is, if

$$\nabla \cdot \mathbf{u} = 0 \tag{7-31}$$

the volume (area for 2D problems) bounded by the interface should be conserved if there is no inflow or outflow through the boundaries. To obtain exact numerical conservation, you can switch to the conservative form

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (\mathbf{u}\phi) = \gamma \nabla \cdot \left( \varepsilon \nabla \phi - \phi(1-\phi)\frac{\nabla \phi}{|\nabla \phi|} \right) \tag{7-32}$$

in the **Level Set** feature's **Settings** window.

Using the conservative level set form you obtain exact numerical conservation of the integral of $\phi$. Note, however, that the non-conservative form is better suited for numerical calculations and usually converges more easily. The non-conservative form, which is the default form, only conserves the integral of the level set function approximately, but this is sufficient for most applications.

### INITIALIZING THE LEVEL SET FUNCTION

Before you can solve Equation 7-30 or Equation 7-32, you must initialize the level set function such that it varies smoothly from zero to one across the interface. Do so by letting $\phi_0$ be zero on one side of the interface and one on the other. Then solve

$$\frac{\partial \phi}{\partial t} = \gamma \nabla \cdot \left( \varepsilon \nabla \phi - \phi(1-\phi)\frac{\nabla \phi}{|\nabla \phi|} \right) \tag{7-33}$$

using $\phi_0$ as the initial condition from $t = 0$ to $t \approx 5\varepsilon/\gamma$. The resulting $\phi$ is smooth across the interface and a suitable initial condition to the level set equation. The Level Set interface automatically sets up Equation 7-33 if you select **Transient with Initialization** from the **Studies** list in the **Select Study Type** page when adding the physics interface.

### VARIABLES FOR GEOMETRIC PROPERTIES OF THE INTERFACE

Geometric properties of the interface are often needed. The unit normal to the interface is given by

$$\mathbf{n} = \left.\frac{\nabla \phi}{|\nabla \phi|}\right|_{\phi = 0,5} \tag{7-34}$$

grad_phi表示法向向量
｜grad_phi｜表示模
两者相除就是单位法向向量

The curvature is defined as

$$\kappa = -\nabla \cdot \mathbf{n}|_{\phi = 0,5} \tag{7-35}$$

These variables are available in the modeling interface as the *interface normal* and *mean curvature*.

---

**Note:** It is only possible to compute the curvature explicitly when using second-order or higher-order elements.

---

## *The Level Set Interface*

The Level Set interface ( ) provides the equations and boundary conditions for using the level set method to track moving interfaces in fluid-flow models, solving for the level set function. The main feature is the **Model Settings** feature, which adds the level set equation and provides an interface for defining the level set properties and the velocity field. You find the Level Set interface in the **Mathematics>Moving Mesh** folder on the **Add Physics** page in the **Model Wizard**.

For a more extensive introduction to the physics and equations implemented by this interface, see the Introduction.

When you add the Level Set interface, it creates the **Level Set** node with default **Model Settings** and **No Flow** (the default boundary condition) nodes added. There is also a default **Initial Values** feature. Right-click the **Level Set** node to add other boundary conditions, for example. The following sections provide information about all feature nodes available in the Level Set interface.

The **Level Set** window contains the following sections:

### INTERFACE IDENTIFIER

This is the identifier for the interface, which you use to reach the fields and variables in expressions, for example.

The identifier appears in the **Identifier** edit field, and you can change it to any unique string that is a valid identifier. The default identifier (for the first Level Set interface in the model) is `ls`.

### DOMAINS

In this section you select the domains where you want to define the level set function and the level set equation that describes it. The default setting is to include all domains in the model. For information about selecting domains, see “Selecting and Deselecting Geometric Entities” on page 41 of the *COMSOL Multiphysics User's Guide*.

#### STABILIZATION

To display this section, select **Show More Options** from the **View** menu in the **Model Builder**.

This section contains settings for the stabilization methods. There are two types of stabilization methods for the level set equation: streamline diffusion and crosswind diffusion. Both are active per default and should remain active for optimal performance. To disable one or both of the stabilization methods, clear either or both of the **Streamline diffusion** and **Crosswind diffusion** check boxes.

#### ADVANCED SETTINGS

To display this section, select **Show More Options** from the **View** menu in the **Model Builder**.

In this section you can specify some advanced settings that you normally do not need to change. For more information about these settings, see “Advanced Settings” on page 114 of the *COMSOL Multiphysics User's Guide*.

##### *Convective Terms*

In this section you can switch from the default non-conservative form to the conservative form of the convective term (see “Conservative and Non-Conservative Form” on page 205). To do so, select the **Conservative form** from the **Convective term** list.

##### *Equation Form*

Typically, the automatic equation form setting is appropriate. The equation formulation then complies with the analysis type. To specify the equation formulation manually select one of the available forms from the **Equation form** list.

##### *Show All Model Inputs*

Select the **Show all model inputs** check box to make all model inputs appear in the **Model Input** section of the settings windows for the main model equation features. Such normally hidden model inputs include field variables in predefined multiphysics interfaces and model inputs from user-defined property groups.

#### DISCRETIZATION

To display this section, select **Show More Options** from the **View** menu in the **Model Builder**.

In this section you can specify the order of the shape functions. The default is to use second-order Lagrange elements. For more information about the discretization settings, see “Discretization” on page 115 of the *COMSOL Multiphysics User’s Guide*.

#### DEPENDENT VARIABLES

Here you can change the name of the dependent variable for the volume fraction of the fluid. To change the name, type a new name in the **Volume fraction of fluid 2** edit field.

## *Model Settings*

The **Model Settings** feature adds Equation 7-30 and provides the possibility to define the associated level set parameters and the velocity field.

The **Model Settings** window contains the following sections:

#### DOMAINS

In this section you select the domains where you want to use the properties for the Level Set interface defined in this **Model Settings** feature node. For information about selecting domains, see “Selecting and Deselecting Geometric Entities” on page 41 of the *COMSOL Multiphysics User’s Guide*.

#### LEVEL SET PARAMETERS

In this section you define the following level set parameters:

- Enter a value or expression for the parameter $\gamma$ (see Equation 7-30) in the **Reinitialization parameter** edit field. The default value is 1 (SI unit: m/s).
- Enter a value or expression for the parameter $\varepsilon_{ls}$ (see Equation 7-30) in the **Parameter controlling interface thickness** edit field. The default expression is `hmax/2`, which means that the value is half of the maximum mesh element size in the region where the interface passes. The SI unit for $\varepsilon_{ls}$ is meters (m).

#### CONVECTION

In this section you enter the velocity field **u** that defines the convection that moves the interfaces. Enter values or expressions for the components ($u$, $v$, and $w$ in 3D, for example) of the velocity field in the **Velocity field** edit fields.

## *Initial Values*

The **Initial Values** feature adds an initial value for the level set function $\phi$ that can serve as an initial condition for a transient simulation. If you need to specify more than one set of initial values, you can add additional **Initial Values** features.

The **Initial Values** window contains the following sections:

#### DOMAINS

In this section you select the domains where you want to define initial values for the two-phase flow variables. For information about selecting domains, see "Selecting and Deselecting Geometric Entities" on page 41 of the *COMSOL Multiphysics User's Guide*.

#### INITIAL VALUES

Here you enter a value or expression for the initial value of the volume fraction of fluid2 The value must be in the range from 0 to 1. The default value is 0.

## *Boundary Conditions*

The Level Set interface contains the following boundary conditions:

- Initial Interface
- Inlet
- No Flow (the default boundary condition)
- Outlet
- Symmetry

## *Initial Interface*

The **Initial Interface** feature defines the boundary as the initial position of the interface.

The **Initial Interface** window contains the following section:

#### BOUNDARIES

In the **Boundaries** section you select the boundaries that represent the initial position of the interface. For information about selecting boundaries, see "Selecting and Deselecting Geometric Entities" on page 41 of the *COMSOL Multiphysics User's Guide*.

## *Inlet*

The **Inlet** feature adds a boundary condition for inlets (inflow boundaries) in a Level Set interface. At inlets you must specify a value of the level set function $\phi$. Typically you set $\phi$ to either 0 or 1.

The **Inlet** window contains the following sections:

#### BOUNDARIES

In the **Boundaries** section you select the boundaries that represent inlets. For information about selecting boundaries, see “Selecting and Deselecting Geometric Entities” on page 41 of the *COMSOL Multiphysics User’s Guide*.

#### SETTINGS

In this section you specify a value of the level set function $\phi$ in the **Level set function value** edit field. The value must be in the range from 0 to 1. The default value is 0.

## *No Flow*

The **No Flow** feature adds a boundary condition that represents boundaries where there is no flow across the boundary. This is the default boundary condition.

The **No Flow** window contains the following section:

#### BOUNDARIES

In the **Boundaries** section you select the boundaries that are no-flow boundaries. For information about selecting boundaries, see “Selecting and Deselecting Geometric Entities” on page 41 of the *COMSOL Multiphysics User’s Guide*.

## *Outlet*

The **Outlet** feature adds a boundary condition for outlets (outflow boundaries) in a Level Set interface. This feature imposes no boundary condition on the level set function.

The **Outlet** window contains the following section:

#### BOUNDARIES

In the **Boundaries** section you select the boundaries that represent outlets. For information about selecting boundaries, see “Selecting and Deselecting Geometric Entities” on page 41 of the *COMSOL Multiphysics User’s Guide*.

### *Symmetry*

The **Symmetry** feature adds a boundary condition for boundaries that represent a symmetry line or symmetry plane.

The **Symmetry** window contains the following section:

#### BOUNDARIES

In the **Boundaries** section you select the boundaries that are symmetry boundaries. For information about selecting boundaries, see "Selecting and Deselecting Geometric Entities" on page 41 of the *COMSOL Multiphysics User's Guide*.

---

**Note:** You do not need to specify a boundary condition for axial symmetry. For the symmetry axis at $r = 0$, the program automatically provides a suitable boundary condition and adds an **Axial Symmetry** feature to the model that is valid on the axial symmetry boundaries only.

---