Two-Fluid Interface Dynamics

Report instructions

Version 1.0

1. INTRODUCTION

This Comsol-exercise concerns the implementation of a level-set method for the simulation of two-fluid interface system using the Single-Phase Laminar flow interface in Comsol. You are expected to compare your simulation results with experimental data based on your own measurements on a real two-fluid system. In addition, you should compare your result with the level-set implementation that can be found in Comsols Multiphase flow interface (a part of the CFD-module).

In addition to insights in two-fluid dynamics and the level-set method the exercise will give you experience in some topics that are important for the project in the second part of the course:

- Stabilization techniques (Streamline diffusion and Cross-wind diffusion)
- Experimental and simulation data comparison
- Sensitivity, Mesh quality and Mesh convergence study

Note: In Comsol the Single-Phase Laminar flow interface per default apply two numerical stabilization techniques (Streamline diffusion and Crosswind diffusion) which are crucial when solving Navier-Stokes equations. In addition to Navier-Stokes we will implement a PDE for the level-set function and this equation also needs to be stabilized by the same techniques. To achieve this we use a module in Comsol called Heat Transfer in Fluids, which per default have such stabilization implemented.

2. MEASURMENTS AND EVALUATION

Validate your simulation by comparing it with your data from the measurement on the real experiment. It is up to you to find a proper way to achieve such a comparison.

In addition to the data comparison you are expected to evaluate the implementation using a Sensitivity study, a Mesh quality study and a Mesh convergence study.

2.1 SENSITIVITY STUDY USING PARAMETRIC SWEEP

It is usually wise to investigate how sensitive the simulation implementation is on different parameter settings. You can do this manually by tuning the parameters or you can perform an automatic parameter study. In the following example we study how the density variation along a line through the two fluids is affected by changes in a parameter:

1. Under **Study** add a **Parametric Sweep**, choose the parameter you want to investigate and write in the Parameter list the values you want to test.
2. Under the node **Results / Dataset**, create a **Cut Line 2D**, select the **Parametric study** as Data set and define the line.
3. Under the node **Results**, create a **1D Plot Group** and choose the Cut Line 2D as your **Data set.** Add a **Line Graph** and plot your fluid density.
4. Run the simulation. Remember that the mph-file can become large when running a large parameter study.

Krister Wiklund
Dept. of Physics
Umeå University

Computational Fluid Dynamics
Simulation Lab

211014

2.2 MESH QUALITY STUDY

Under the node **Results**, create a **2D Plot Group** and choose a **Mesh** plot. Under **Color** choose **Element color** to Quality.

2.3 MESH CONVERGENCE STUDY USING PARAMETRIC SWEEP

To show that the chosen mesh is good enough a mesh convergence study has to be done. This means that a specific variable is monitored for solutions on different meshes. Each mesh corresponds to a certain *Degrees of Freedom (DoF)* which indicates the computational load, in Comsol it can be found as the predefined variable *numberofdofs*. A fine mesh corresponds to large DoF and thus high computational load.

To show mesh convergence you plot your monitor variable against DoF and hopefully you will see a plateau region where higher DoF does not change the monitor variable value. The smallest DoF in the plateau is the most optimal mesh in terms of accuracy and computational load.

1. Under **Global Definitions / Parameters** define a parameter Meshtest and assign it the value one.
2. Under the **Mesh / Sequence** type choose User-controlled mesh. Under **Mesh / Size 1** choose **Custom** and check all boxes. Under Maximum element size write Meshtest*0.008.
3. Under **Study / Parametric Sweep**, choose the parameter Meshtest and write in the Parameter list 1 2 3.
4. Under the node **Results**, create a **1D Plot Group**, add a **Line Graph** and plot your monitoring variable against the DoF (*numberofdofs*).

3. REPORT INSTRUCTION

In this lab we focus on the results and the analysis of the results. The report that you individually hand in, according to the instructions below, is therefore a very brief report.

3.1 DEFINITION OF BRIEF REPORT IN THIS COURSE

Do not use bullet lists, discuss instead using a running text. Use max 4 figures and about a half page discussion for each figure. In the report the following parts should be included:

Alternative 1

- Front page (with the same data as in a full report, no Abstract)
- Result section (Clear figures and tables in which interesting parts are pointed at and very shortly commented)
- Discussion section (Each figure and table in the Result section are referred to and analyzed)

Alternative 2

- Front page (with the same data as in a full report, no Abstract)
- Result and Discussion are combined into one section (Clear figures and tables in which interesting parts are pointed at and discussed)
- Conclusion

Krister Wiklund   Computational Fluid Dynamics   211014
Dept. of Physics   Simulation Lab
Umeå University

3.2 INSTRUCTIONS FOR REPORT SUBMISSION

To make the organization of report submission easy for the examiner and to decrease the grading time the following instructions should be obeyed:

1. The report must be in a PDF-format
2. The name of the PDF must have the following form:

   Levelset_FIRSTNAME_SECONDNAME_DATE.pdf

   FIRSTNAME= Your first name
   SECONDNAME= Your second name
   DATE = The date you hand in the report and it should be on the format 211125

3. The report is submitted through Canvas / Assignments
4. **Deadline for submission is 25 November kl. 23:56**

Hint: Remember that the experiment is 3D and your simulation is 2D.

A simple model of the glass friction contribution:

1) Assume that z-direction is into the fluid and z=0 is at the glass surface and z=H/2 is in the middle of the channel, then the viscous stress magnitude in x-direction (from one glass side) is determined by mu_global*du/dz at z=0. The gradient in z-direction is however not known in the 2D-simulation.

2) A first rough approximation of the gradient is u/(H/2), where u is the 2D-simulation velocity in x-direction. We thus assume a linear velocity profile in z-direction and that the 2D-simulation corresponds to the center plane of the experimental device.

3) The stress can be introduced into the simulation by adding a volume force expressed in the following way:

 -2*mu_global*u/(H/2)*ElementArea/ElementVolume

where ElementArea=h^2, ElementVolume=h^3 and h is Comsols pre-defined element length.