NTNU – Trondheim
Norwegian University of
Science and Technology

Oil Containment

Henning Bjoervik

Supervisors:
Professor Bjørnar Pettersen
Associate Professor Håvard Holm
Professor Hyun Chung (KAIST)

Scope of the work

The scope of this work is to analyze drainage failure of an oil boom, using the open-source library OpenFOAM.

This is a two-phase problem, and it will be analyzed using the interFoam solver in OpenFOAM. It is also interesting to see how the interface between the two phases react to different density, grid refinement etc. A study of this is beneficial for the drainage model as well.

Introduction

Oil spills causes severe damage to both environment and economy. To contain the oil and prevent it from spreading, oil containment booms are typically used. These booms are typically efficient in sheltered water, but less efficient in open water where forces from waves, wind and currents are applied.

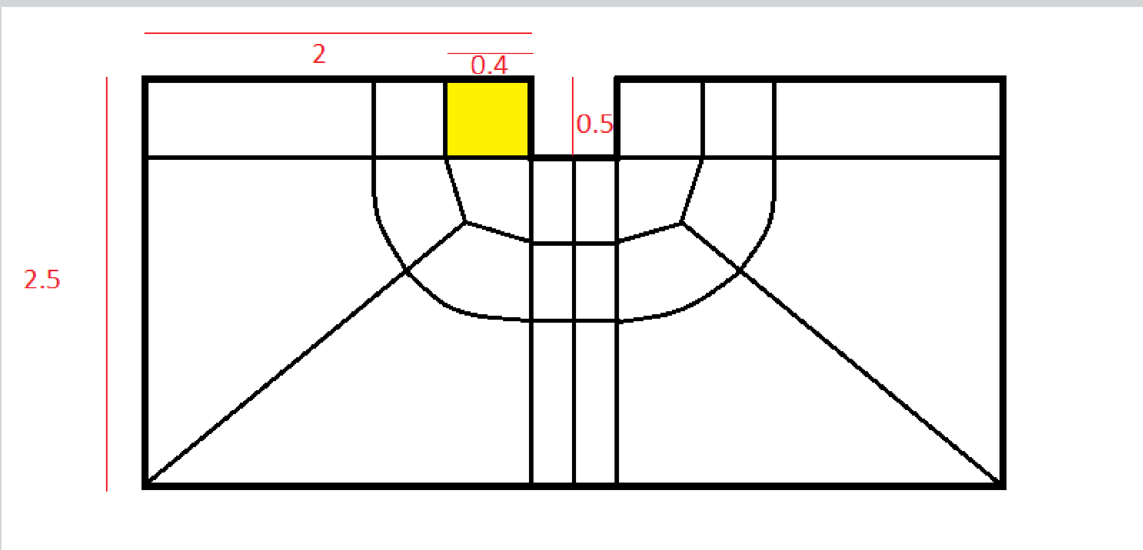
The thesis presents the oil boom concept and failure modes. As OpenFOAM is the software used in this thesis, a presentation of OpenFOAM and its structure is given.

A theoretical background for one-phase flow and two-phase flow is given, in addition to a description of how OpenFOAM solves the problems.

In the thesis a study of the following is carried out:

- Verification of interFoam solver
- One phase case
- Channel case with two-phase flow
- Two phase case

Methods and modelling



The oil boom is in this thesis modelled as a fixed object. Using a dynamic mesh to model its behaviour is a proposal for further work.

The mesh configuration used in the simulations is shown above. The area of interest, i.e. the marked area, is where we want the oil to circulate instead of being drained downwards. This mesh configuration lets us refine the mesh in front of the object without refining the mesh too much elsewhere in the domain.

The solver used in OpenFOAM is the interFoam solver. The solver uses the Volume of fluid method (VOF) for tracing the interface, while it uses PISO for coupling of pressure and velocity.

The VOF method solves the equations for each phase, where the phase location is solved using α, volume fraction. In each cell α decide how much of the cell is occupied by the respective phase. α takes on a value between 0 and 1 everywhere in the domain, where:

$$\alpha = \begin{cases} 1, & \text{for fluid 1} \\ \text{between 0 and 1,} & \text{for interface} \\ 0, & \text{for fluid 2} \end{cases} \tag{1}$$

The equation below is used to track the interface, i.e. determine *alpha* in each cell.

$$\frac{\delta\alpha}{\delta t} + \vec{\nabla} \cdot (\vec{V}\alpha) = 0 \tag{2}$$

This equation does not account for compression of the interface. To do this another term is added to equation 1, as shown below.

$$\frac{\delta\alpha}{\delta t} + \vec{\nabla} \cdot (\vec{V}\alpha) + \vec{\nabla} \cdot (\vec{V_r}\alpha(1-\alpha)) = 0 \tag{3}$$

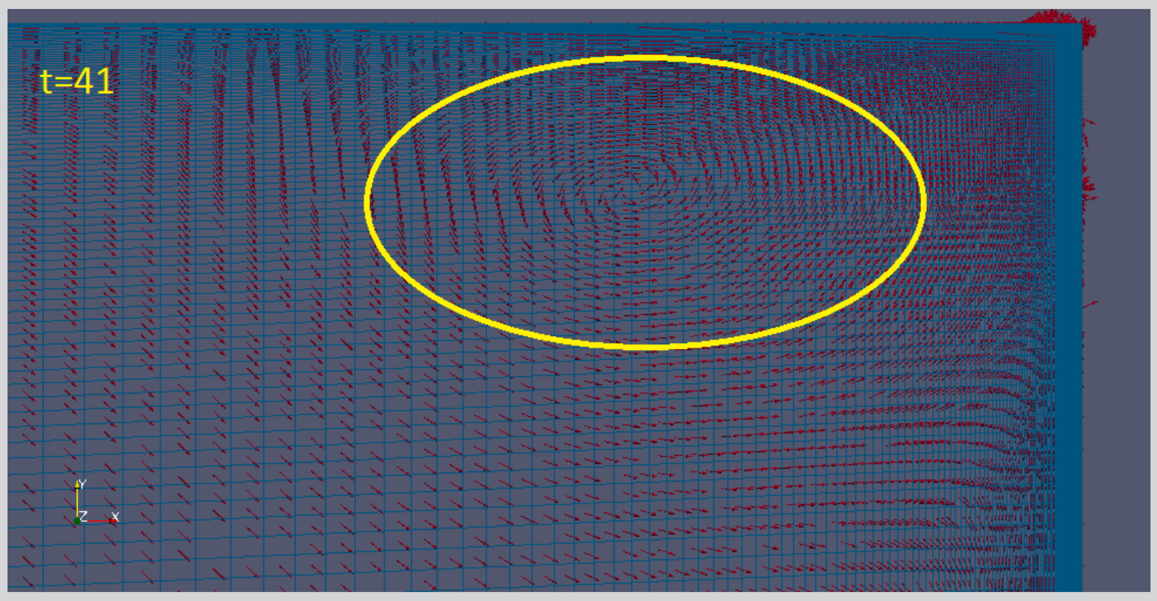
Where V_r is a velocity field suitable to compress at the interface. This additional term will be zero away from the interface as α approaches a value of 1 or 0 when the fluids are not mixed as in the interface.

Results

One-phase flow
The key point in these simulation is to preserve circulation of fluid upstream of the oil boom.

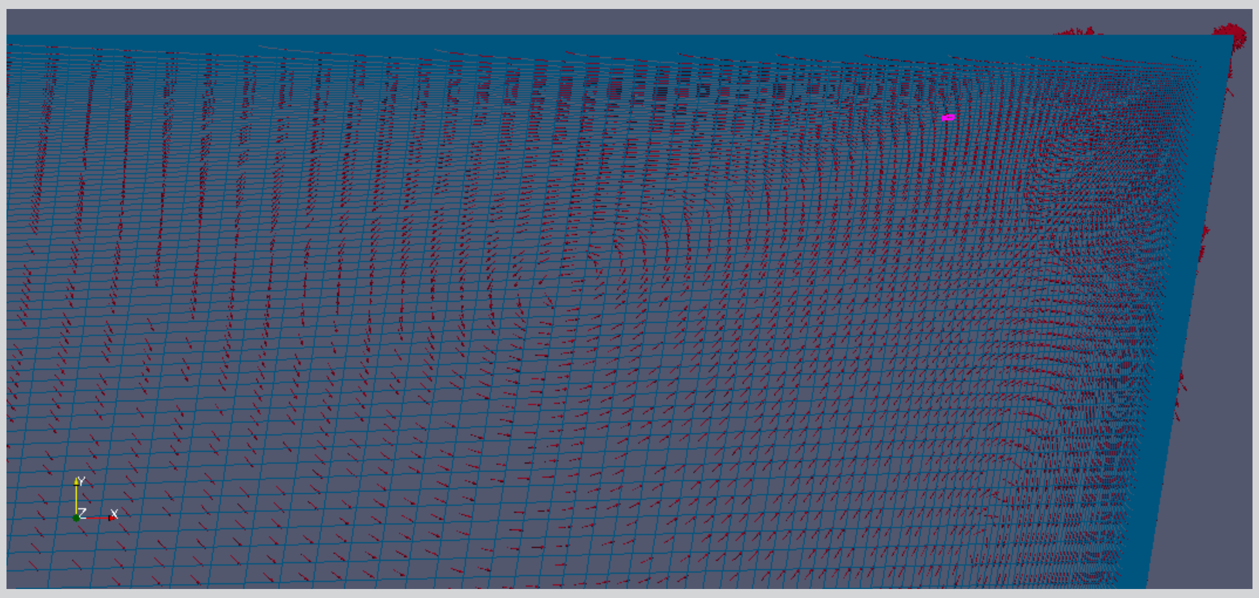
We want to see the circulation of fluid in a onephase flow of water. Refining the area in the domain marked with yellow, we find that the necessary mesh refinement is:

- horizontal: 180 cells, min.dl.=0.000014
- vertical: 300 cells, min.dl.=0.000028



The picture above shows the circulation at Reynolds number 50. By increasing the Reynolds number, the circulation area gets smaller, and vanishes at Reynolds number 300.

Simulations of angled oil boom are also ran. This shows that angling the plate in opposite direction of the flow increases the area of circulation drastically.

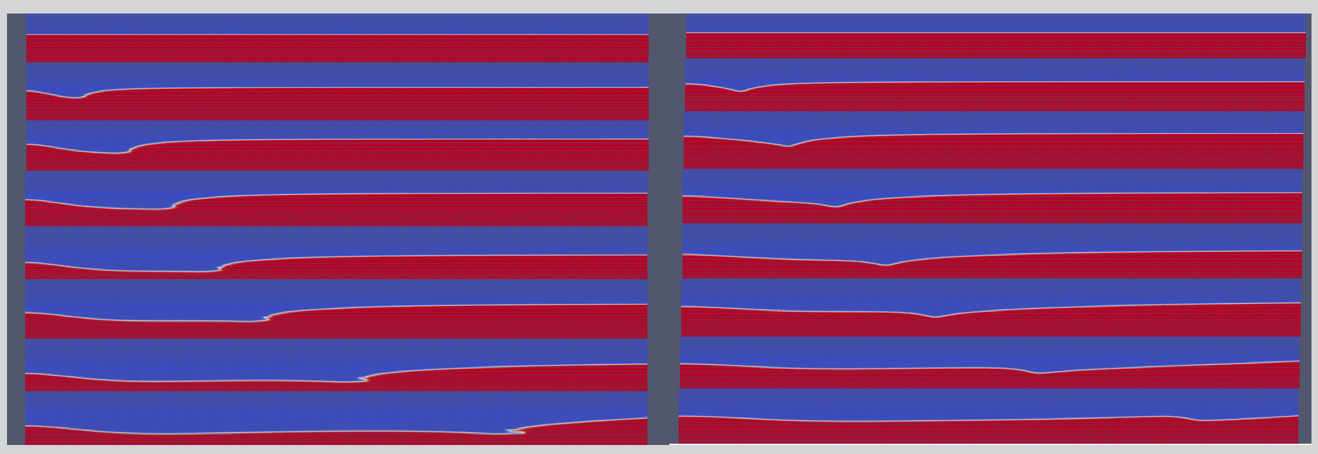


The pink box shows the location of lower left corner of the circulation area for angle = 0 degree.

Two-phase flow
Simulations of a channel case are done, with water at the bottom and oil on top (with equal proportions). The purpose of this case is to test sensitivity of the interface region when it comes to mesh refinement, skewness, aspect ratio, difference in density and Froude number.

The simulations show that the interface region is very sensitive to the mesh. aspect ratio far from 1 and skewness should be avoided. The simulations show that an irregularity, e.g. skewed, in the interface in the set-up will develop and become severe in the simulation. The interface region should be modelled as smooth as possible.

The interface region is highly sensitive to the Courant number. The simulation below shows the same simulation, but the simulation to the left is ran with CFL=1 and the simulation to the right with CFL=0.25 in the interface region. The simulation to the left clearly shows numerical diffusion in the internal wave that occurs at the interface.



Remaining work Next step is to run circulation simulations in a two-phase flow with the oil boom, using the experience from the one-phase flow simulations and the channel-case simulations.