UMEÅ UNIVERSITY
Department of Physics
Lab 1

November 22, 2021

Lab 1

Computational Fluid Dynamics, Autumn 2021, 7.5 Credits

Two Fluid Interface Dynamics

Name	Elias Olofsson
user@student.umu.se	`elol0076@student.umu.se`

Teachers
Krister Wiklund, Erik Zäll

Contents

1 Results and Discussion

Having implemented the Level-Set method in two different ways in Comsol, first by using the Single-phase laminar flow interface and second using the native Multi-phase Level-Set interface for laminar flow, we can compare the simulated results with experimental data.

A video visually comparing the experimental flow with the Single-phase simulation can be found at Youtube (link: `https://youtu.be/cZ7xnbeu0Lk`) for a kinetic viscosity of 17.5 mm^2/s. As one can see, the behaviour and the general motion of the fluid in the simulation corresponds quite well to the real-life experiment. However, the velocities seemingly gets dampened a lot faster in the real experiment and an equilibrium is reached much quicker in real-life than in the simulated experiment. Possible reasons for this could either be that we have too low viscosities in the simulation, or that the frictional forces against the glass in the real experimental 3D-flow have not been modeled in a satisfactory way in our 2D-simulation. Additional factors like mismatching initial conditions and other artifacts of real physical experiments can also play a role in explaining these discrepancies.

After having manually extracted a time series for the evolution of the fluid interface from the video, we can numerically compare the experimental results with the simulated counterparts. Graphing the surface level a centimeter from the left-hand side of the domain, we can plot the time evolution in Fig.(1). Here we can see the discrepancy mentioned earlier, where the simulated versions reach a larger amplitude in the oscillations, and have a lesser dampening than the physical experiment.

Furthermore, we have in the same figure graphed the simulated time evolution of the surface for a range of different kinematic viscosities. Specifically, we have performed a parametric sweep for the kinematic viscosity of the fluid corresponding to the oil, where we used the range of values given in the attached data sheet for the oil used in the real experimental setup. As one can see in Fig.(1), the general behaviour is slightly altered within the sensitivity study, but the difference is in general much smaller than the difference to the real, observed measurements or the simulation employing the multi-phase laminar flow method.

Furthermore, we have performed a convergence study on the Single-phase simulation to ensure that the chosen mesh is well-suited. To this end, we have measured the velocity magnitude and the pressure in the fluid at the position (-14, -6) centimeters relative to the upper center point of the domain, at time 0.8 seconds into the simulation, for 5 different meshes. The results can be seen in Fig.(2), where these quantities are plotted against the

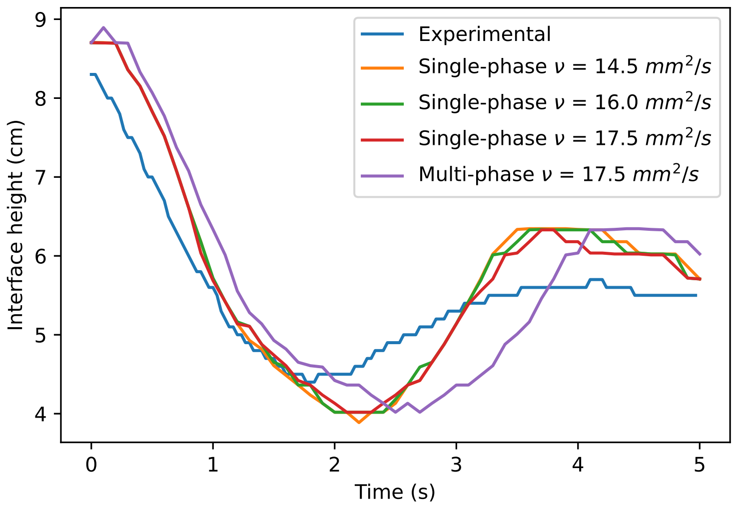

Figure 1 – Time evolution of the fluid interface, measured approximately 1 cm from the left-hand side of the domain. A sensitivity study have been conducted for the Single-phase fluid flow method, where a parametric sweep has been performed for the kinematic viscosity of the fluid corresponding to the oil.

number of degrees of freedom.

I believe it is quite hard to tell from these graphs alone which of the given meshes are well suited and which ones are not. However, what I can tell is that the three coarsest meshes seems to group quite closely to each other, while the finer meshes seem to diverge to a larger extent in the measured quantities. I am however not sure if this is due to the relative proximity in degrees of freedom, or if this is due to better convergence of the mesh.

To get some better insights into this, I have graphed the time evolution of these same quantities for the different meshes, as seen in Fig.(3). Here we can see the full evolution, instead of only observing a single fix point in time. As one can see, the three coarsest meshes are following each other quite closely, while in the finest mesh is diverging quite a lot from the rest. Especially looking at the pressure evolution for the finest mesh, there seems to be a lot of irregularities and discontinuities at the end of the simulation, where there from physical intuition should be less, seemingly indicating unwanted numerical behaviour from the mesh. Thus, it seems evident that the finest mesh is not necessarily the most well-suited one. For a more conclusive understanding, more mesh-sizes must be tested and evaluated, but from this convergence study alone I would guess that the middle option of the tried

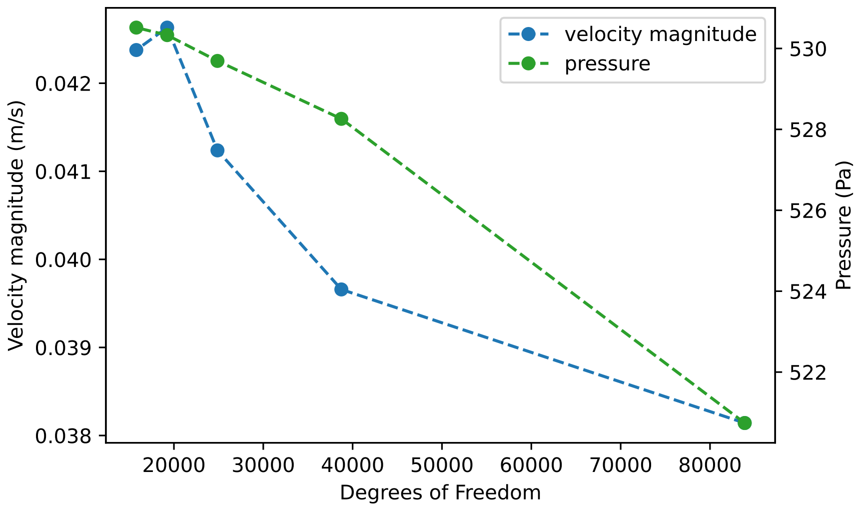

Figure 2 – Mesh convergence studied by graphing the velocity magnitude and the pressure in the Single-phase fluid simulation, measured at the point (-14, -6) centimeters relative to the upper central point of the domain, at time 0.8 seconds, for 5 different meshes.

meshes provides a reasonable trade-off between accuracy and computational speed.

Additionally, we have conducted a mesh quality study of the two most extreme meshes from the convergence study above. The coarsest mesh can be seen in Fig.(4) and the finest mesh tested can be seen in Fig.(5). The finest mesh corresponds to the preset 'extra-fine' used in Comsol, while the coarsest mesh is analogous to the preset 'normal'. Specifically, we used the preset 'extra-fine' to set the finest mesh, and scaled all other meshes by multiplying a integer in the range 1 to 5 to the maximum element size setting.

2 Conclusion

In this lab we have reproduced the experimental result for a two-fluid interface. The time evolution in the simulation was quite similar in general behaviour to the physical experiment, but was was dampened to a lesser degree than the real experiment. The reason was discussed to be due to unsatisfactory modelling of the frictional forces from the glass wall in the real 3D-flow, not experienced in the simulated 2D-flow, among other things. A sensitivity study on the impact of changes in kinematic viscosity was

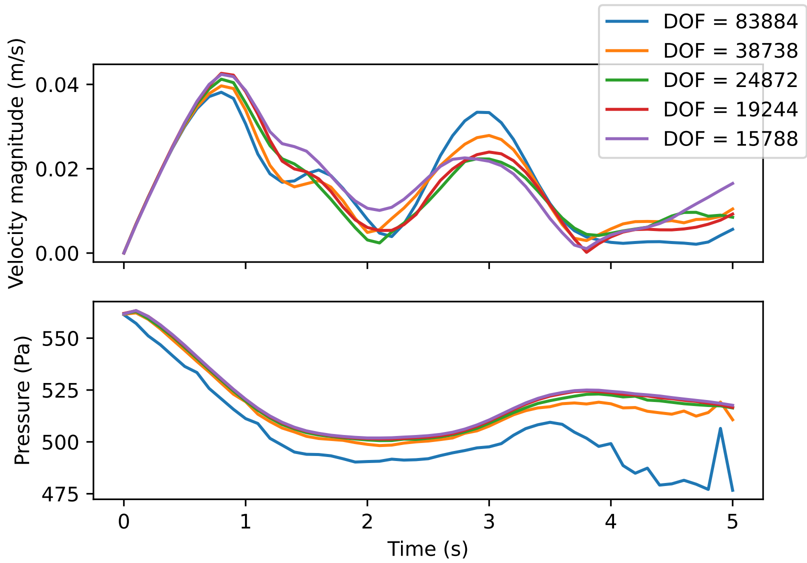

Figure 3 – Mesh convergence studied by graphing the time evolution of the velocity magnitude and the pressure in the Single-phase fluid simulation, measured at the point (-14, -6) centimeters relative to the upper central point of the domain, for 5 different meshes with corresponding degrees of freedom.

conducted, as well as a mesh convergence study and a mesh quality study.

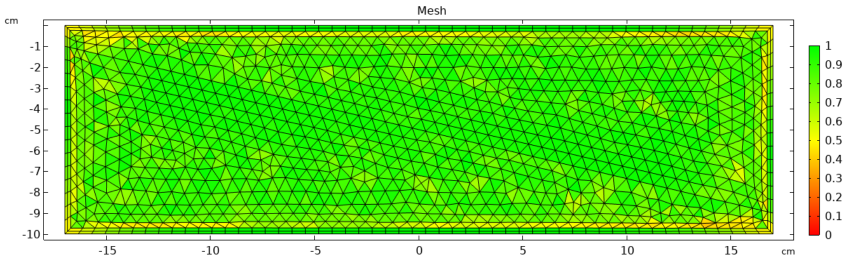

Figure 4 – Mesh quality study corresponding to the coarsest mesh used in the mesh convergence study from above, with approximately 15k degrees of freedom.

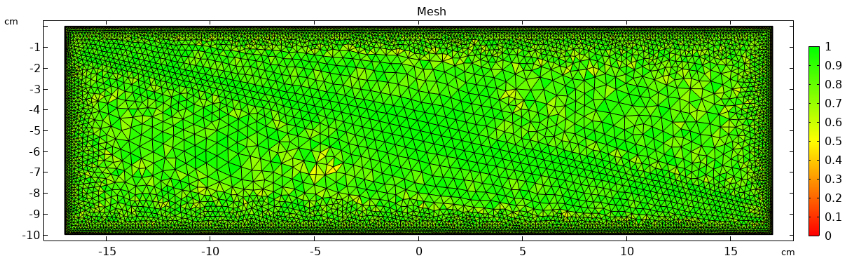

Figure 5 – Mesh quality study corresponding to the finest mesh used in the mesh convergence study from above, with approximately 83k degrees of freedom.