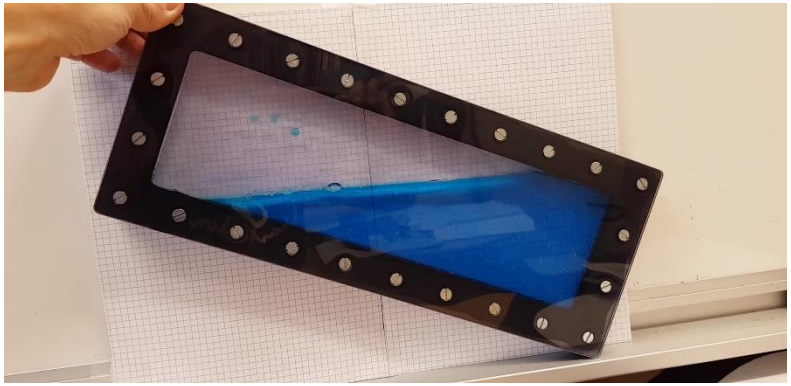
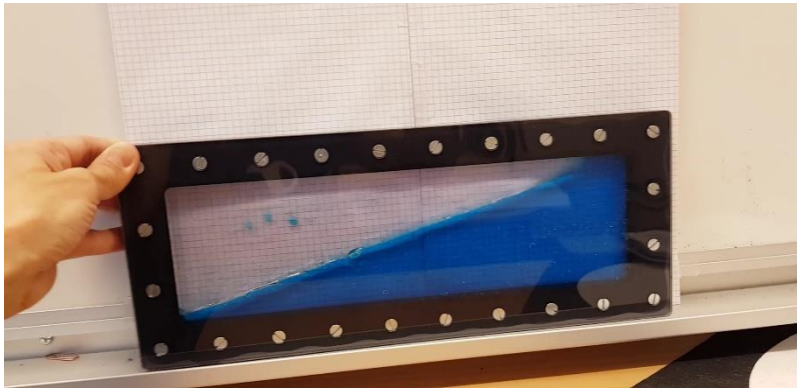
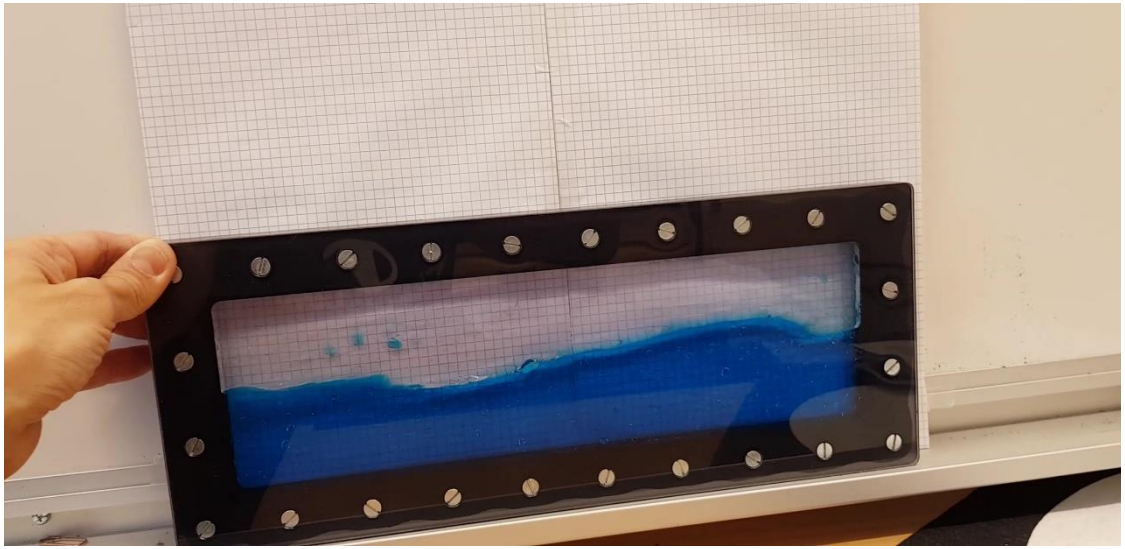
Computational Fluid Dynamics
Comsol Lab 1
Lecture 5

Krister Wiklund
Department of Physics
Umeå University

Comsol Lab 1: Two-fluid Interface Dynamics Experiment (Low Re)

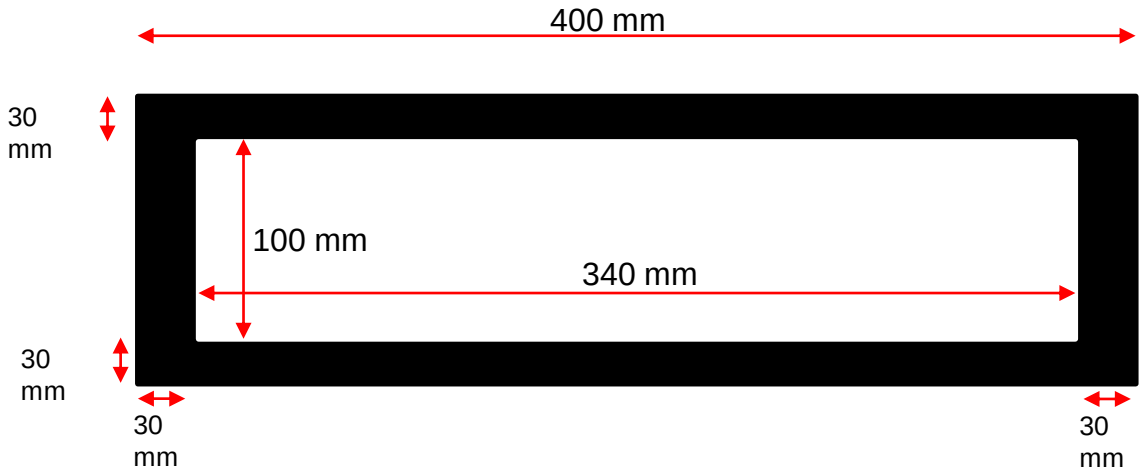






Comsol Lab 1: Two-fluid Interface Dynamics Experiment

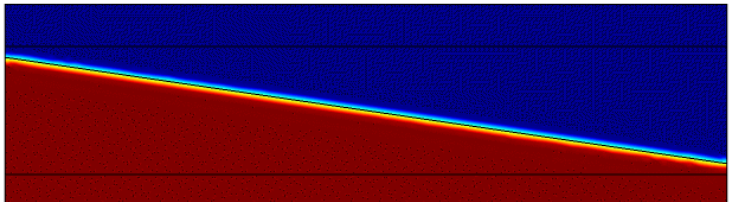
Wave motion machine



Task: Find a way to compare the experiment and the simulation

Two-fluid Interface Dynamics Simulation

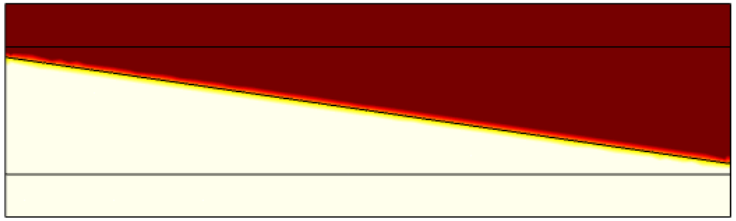
MultiPhase-interface in Comsol



This is Comsols pre-defined interface for multi-fluid simulations.

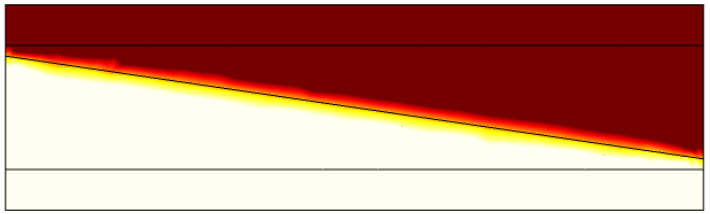
This is the same simulation where you have implemented the two-fluid equations.

Single-phase-interface in Comsol



Fine mesh

Single-phase-interface in Comsol



Coarse mesh

Learning outcome the mandatory lab

- Experimental and simulation data comparison
- Experience with Level-set method
- Sensitivity study
- Mesh quality study
- Mesh convergence study
- Hands-on experience with stabilization techniques such as
  - Streamline diffusion
  - Cross-wind diffusion
  - Other

The Modeling and Simulation process

1. Problem formulation
Describe your interpretation of the problem formulation

2. Choice of measurement method
Formulate Strengths & Weakness of measurement methods.

3. Choice of simplified physical model (an abstraction of reality)
Formulate Strengths & Weakness of simplified model.

4. Choice of simulation approach
Formulate Strengths & Weakness of chosen simulation approach.

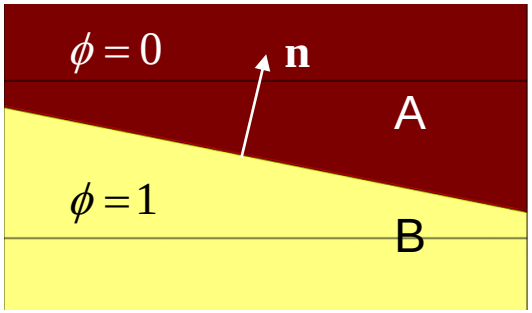
5. Expected simulation data
What are the expected data from your simulation?

6. Evaluation of simulated and measured data
How do you compare simulated data with measured data?

7. Back to 2

The Level-set method (a simple version)

Assume that we have a function, a field, that is constant on the interface (it moves with the interface):



$$\left(\frac{\partial}{\partial t} + \mathbf{u}\cdot\nabla\right)\phi = 0 \qquad \mathbf{n} = \frac{\nabla\phi}{|\nabla\phi|} \qquad \kappa = -\nabla\cdot\mathbf{n}$$

Curvature of interface

$$\Rightarrow \quad \mathbf{u}\cdot\mathbf{n} = \frac{\mathbf{u}\cdot\nabla\phi}{|\nabla\phi|} = -\frac{1}{|\nabla\phi|}\frac{\partial\phi}{\partial t}$$

Physical interpretation

$$\rho(\mathbf{r}) = \rho_A + (\rho_B - \rho_A)\phi(\mathbf{r})$$
$$\mu(\mathbf{r}) = \mu_A + (\mu_B - \mu_A)\phi(\mathbf{r})$$

\+ Navier-Stokes ⇒ The motion of a fluid with two densities and two viscosities can be represented by the standard N-S equations

Numerical implementation

$$\frac{\partial\phi}{\partial t} + \mathbf{u}\cdot\nabla\phi = 0 \longrightarrow \frac{\partial\phi}{\partial t} + \mathbf{u}\cdot\nabla\phi = \varepsilon\nabla^2\phi - \nabla\cdot\left(\phi(1-\phi)\mathbf{n}\right)$$

The Single-phase interface in Comsol

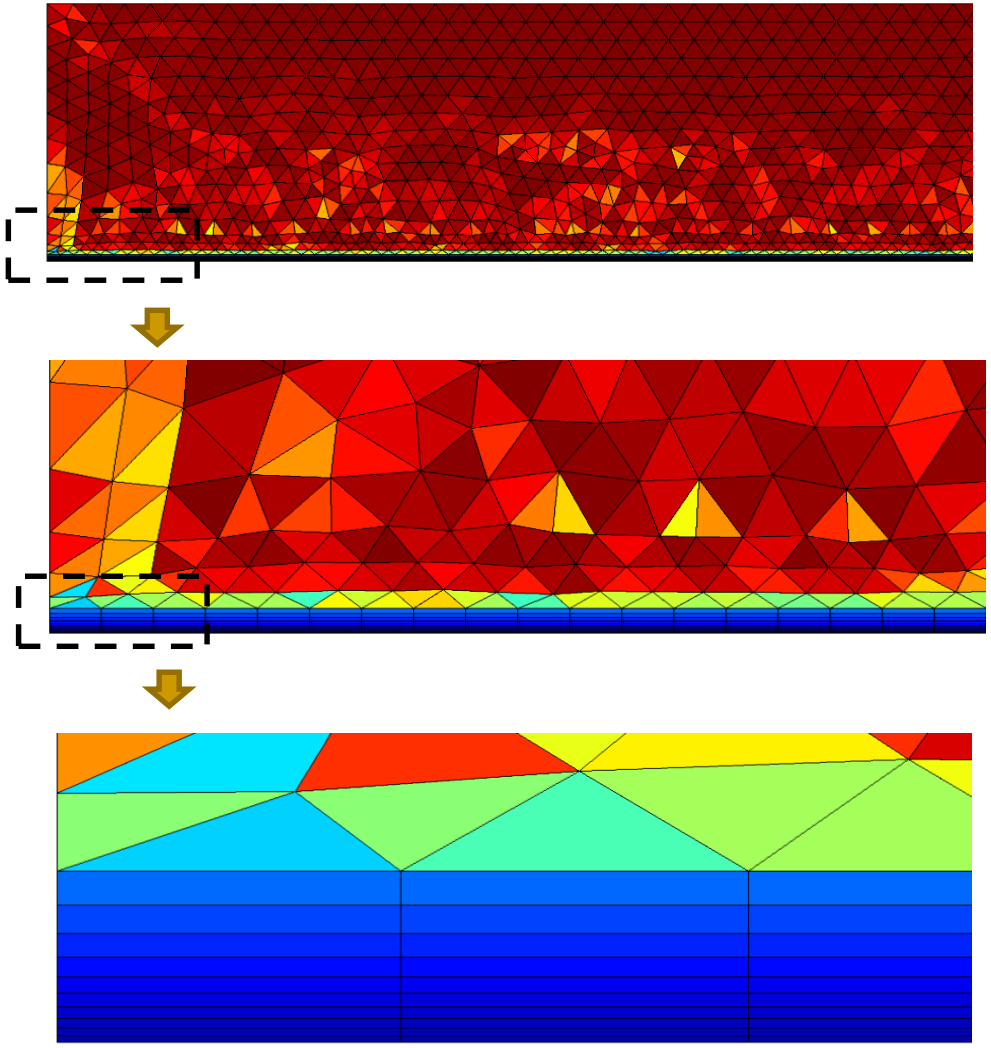
Numerical diffusion

Numerical sharpening of interface

Analysis of a simulation implementation

- **Sensitivity study**
  - Investigate how small changes in simulation parameters affects the simulation results
- **Mesh quality study**
  - Use the software built-in functions
  - In comsol one is *qual*
- **Mesh convergence study**
  - Consider how refining the mesh affects the result
  - An optimal setup has a mesh that is good enough, i.e. the the mesh gives the wanted accuracy without being too fine

Mesh quality function in Comsol

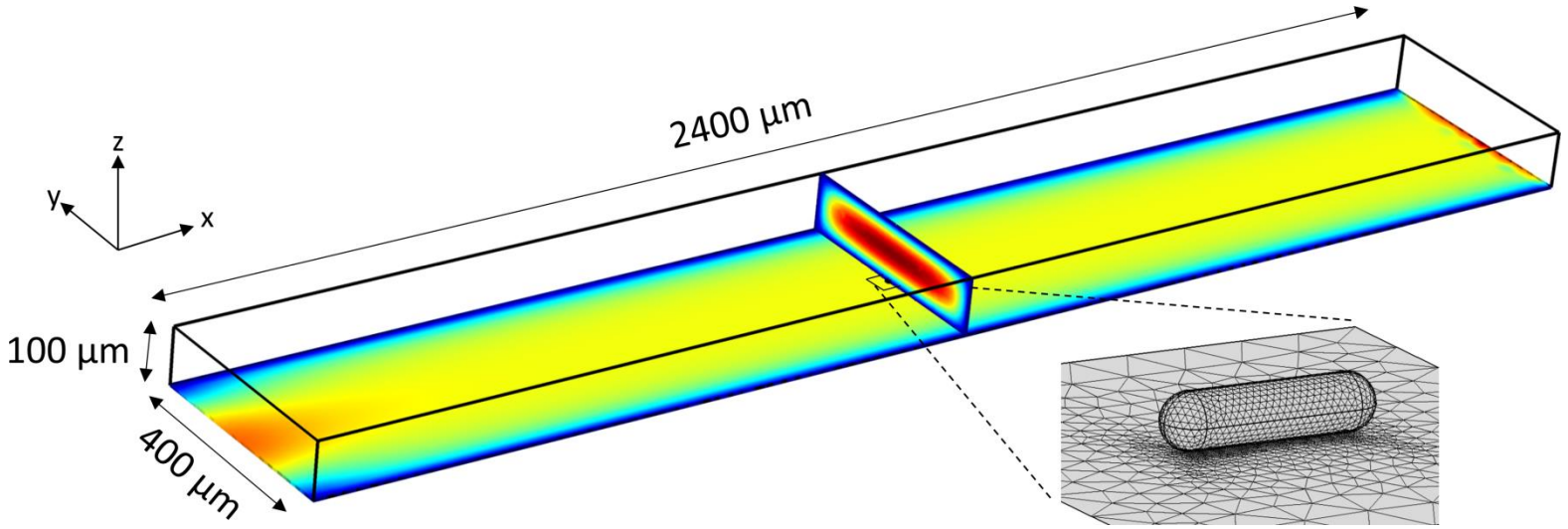


In Comsol you can display the quality of the mesh by creating a mesh-plot:

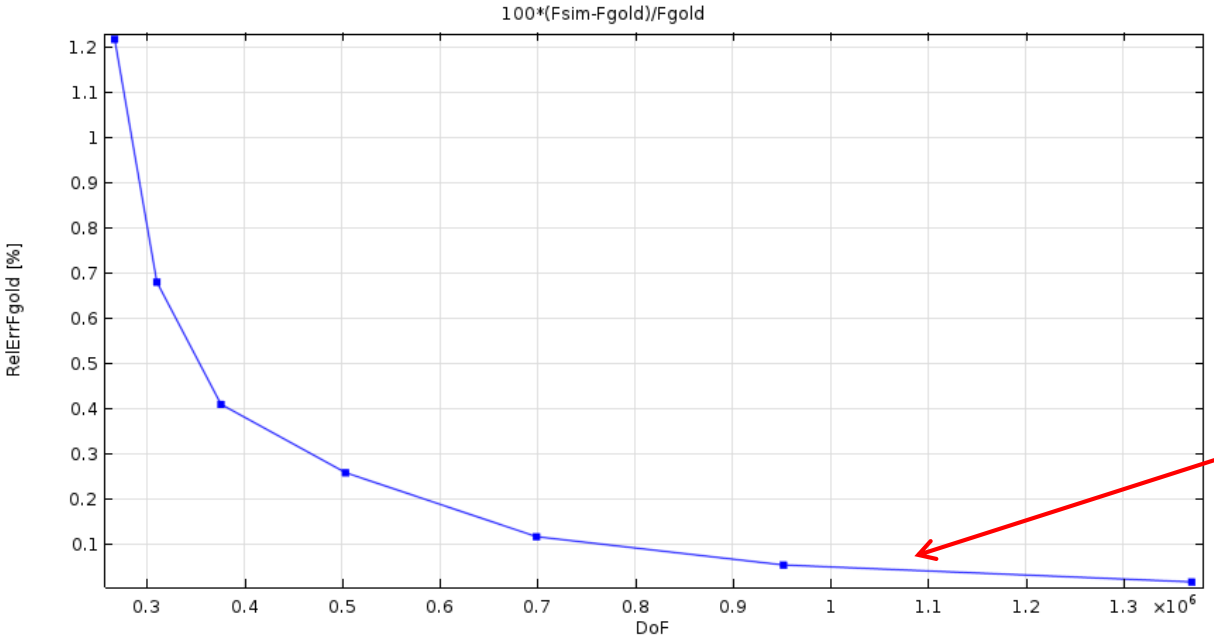
Results / 2D Plot / Mesh

and choosing Quality as Element color.

Mesh convergence study, Example



Mesh convergence study, Example (cont.)



Making the mesh finer does not improve the result much…

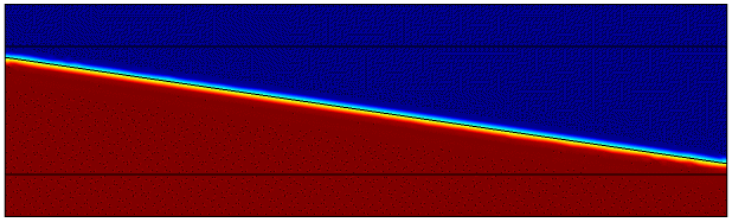
Mesh case	NoE on sphere	NoE in mesh box	NoE in channel	Total NoE	DoF [ ]	Force [pN]	RelErr in F * [%]
1	4030	64045	63793	127838	999	36.579	-0.068
2	4028	64319	37206	101525	731	36.604	-0.14
3	1828	34954	37313	72267	565	36.656	-0.33
4	560	16568	37235	53803	465	36.778	-0.93
5	560	16569	17758	34327	264	37.125	-1.21
6	560	16569	11394	27963	197	37.581	

What if you do not have any analytical model to compare with?

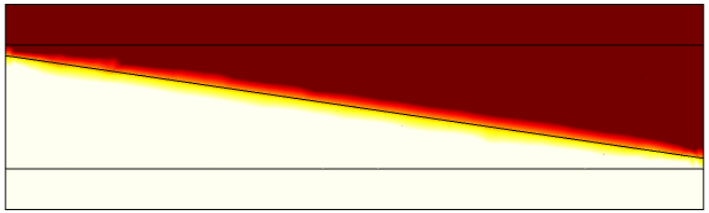
Just plot the Force vs DoF, it should also get better and better the finer mesh we use.
(You can of course plot any variable vs DoF)

Data analysis in the two-fluid simulation lab

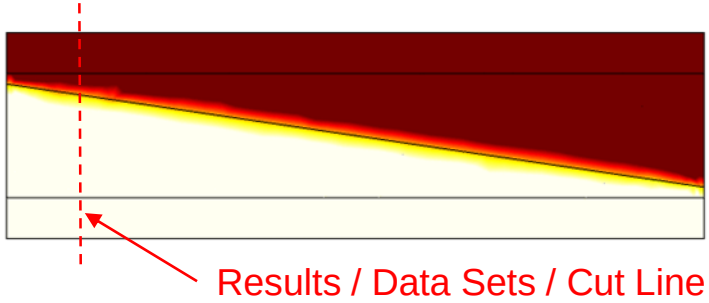
MultiPhase-interface in Comsol



Single-phase-interface in Comsol



Coarse mesh



How do we compare the experimental data with the simulation results?
Example:

- Movie on experiment => Height data
- Cut line in simulation => Height data

Comsol vs Matlab

- Ok to do the analysis within Comsol
- Efficient to do the analysis in Matlab by exporting simulation data as a text file

Sensitivity study
Example: How sensitive is the result on small changes in viscosity?

Note:
In the pre-made Comsol files (and their instructions) in Lecture folders you find hints and tricks useful in the mandatory labs and the project.

End of lecture