Improved VoF algorithm to simulate multiphase flows

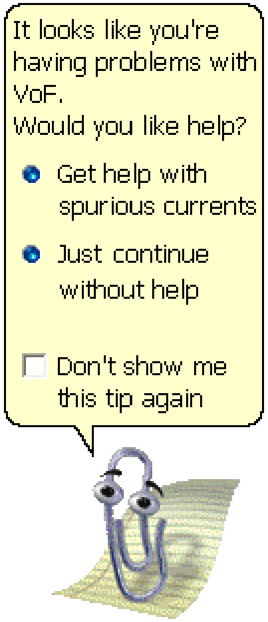
Anton Kidess

Introduction

Modeling of immiscible two phase flows

- VoF methods suffer from spurious currents – unphysical currents that appear without external forcing
- Reduction of spurious currents / improvement of interface tracking a hot research topic
- CLS-VoF, geometric reconstruction, …

- Here: Algorithm presented in Journal of Computational Physics, April 2012, by Raeini, Blunt, Bijelijc
  - SSF = sharp surface forces



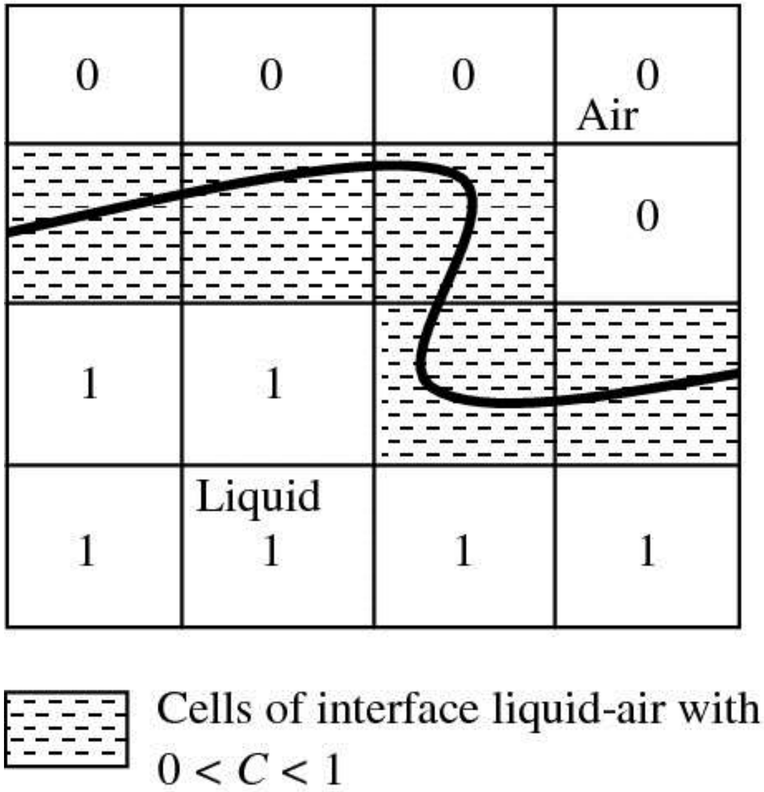
Outline

Modeling of immiscible two phase flows

- Combatting numerical diffusion in VoF methods
- Problems in determining interface curvature with a “sharp” VoF interface
- Combination of smoothing and sharpening in the SSF formulation
- Treatment of the capillary pressure
- Semi-implicit coupling of the volume fraction and momentum solution
- Static test case – spherical bubble equilibrium
- Dynamic test case – Thermocapillary driven cavity

Introduction

The Volume of Fluid method (VoF)



$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (U\alpha) = 0$$

- VoF field is directly coupled to flow field via advection term

VoF problems and solutions

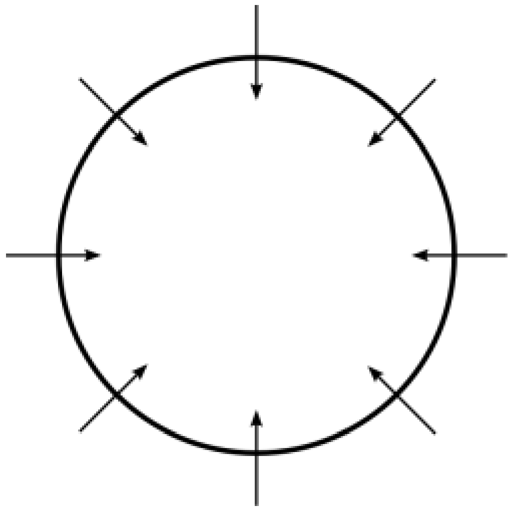
Artificial diffusion of the sharp interface

- A perfect solution to a volume tracking problem will retain a sharp interface shape
  - This is important e. g. to accurately determine drag force on a bubble
  - Strong diffusion would eventually to result in mixing, completely changing the physics
- Pure advection problems will always be accompanied by numerical diffusion
- This can be countered by sophisticated advection schemes incorporating downwinding, and artificial compression
- Dilemma: Too much sharpening will lead to noise in the curvature calculation

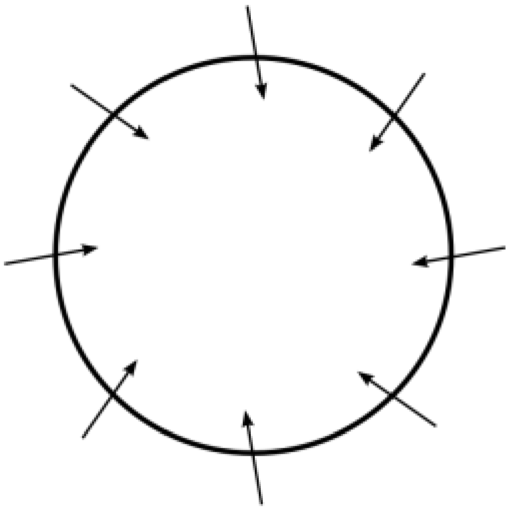
VoF problems and solutions

Noise in curvature due to sharp interface

- Flow field is coupled to VoF through surface tension forces - $\sigma\kappa|\nabla\alpha|$
- Inaccuracies in curvature determination lead to force field not balanced by pressure gradient => rotational velocity field



Irrotational force field, can be balanced by pressure gradient



Rotational force field, must be balanced by parasitic currents

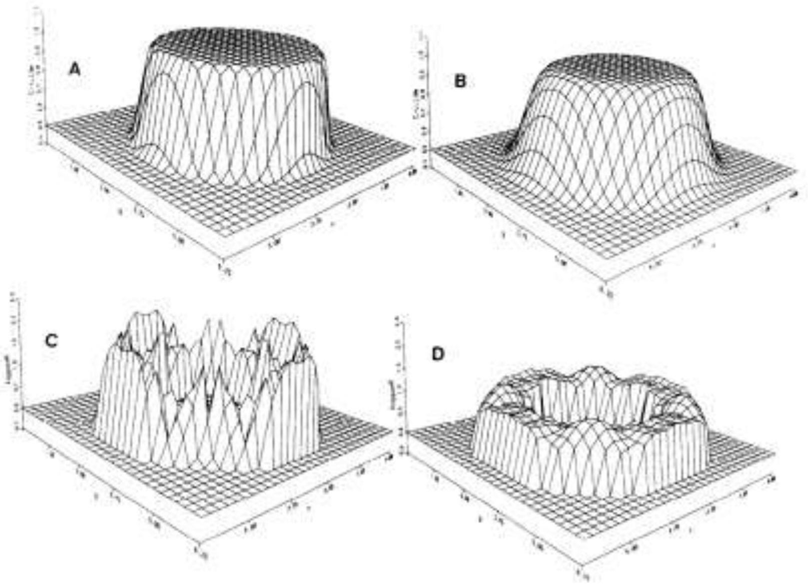
VoF problems and solutions

Noise in curvature due to sharp interface

- Noise can be mitigated by smoothing the volume fraction field

α:

κ:



- Dilemma: Too much smoothing will allow high frequency capillary waves to persist unphysically

VoF problems and solutions

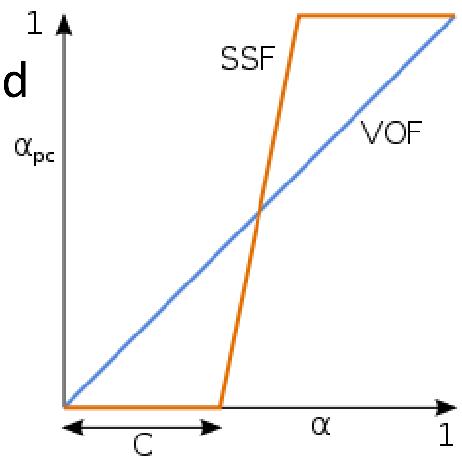
Smoothing in interFoamSSF (the new algorithm)

- Compute VoF field
- Determine curvature from smoothed volume fraction field
- Amount of smoothing determined by a blending coefficient
- Smooth curvature away from interface

VoF problems and solutions

interFoamSSF – sharp surface formulation

- Compute VoF field
- Determine curvature from smoothed volume fraction field
- Amount of smoothing determined by a blending coefficient
- Smooth curvature away from interface
- Apply the resulting surface tension force to a sharpened interface (sharp surface formulation - SSF)
    - $f_{c,f} = \sigma k_f \nabla_f (\alpha_{SSF})$
    - $\alpha_{SSF} = \frac{1}{1-2C}[\min(\max(\alpha, C), 1-C) - C]$
    - C=0 will give original field, C=0.5 a very sharp transition
    - ~0.48 good for static problems, ~0.25 good for dynamic problems



VoF problems and solutions

Notes on pressure

- It is important to determine the surface force at cell faces if the pressure is solved there (already done in interFoam)
    - Using cell values will not balance pressure
- Reference goes a step further and separates capillary pressure and gravity from total pressure
    - $p = p_d + p_c + \rho g$
- Remaining pressure field p_d will be smooth! p_c is calculated explicitly from the normal surface tension force

VoF problems and solutions

Are we done yet? Not quite.

- Standard VoF implementations have explicit coupling of VoF field and velocity
  - Compute VoF field for a given time
  - Update velocity field for this time step based on new transport properties
  - Advance to next time step
- Updated velocity doesn't directly couple back to VoF equation!
- Severe time step criterion: $\Delta t = \sqrt{\rho/_{\sigma}}\,\Delta x^{3/2} \approx O(10^{-6}s)$

VoF problems and solutions

Semi-implicit solution

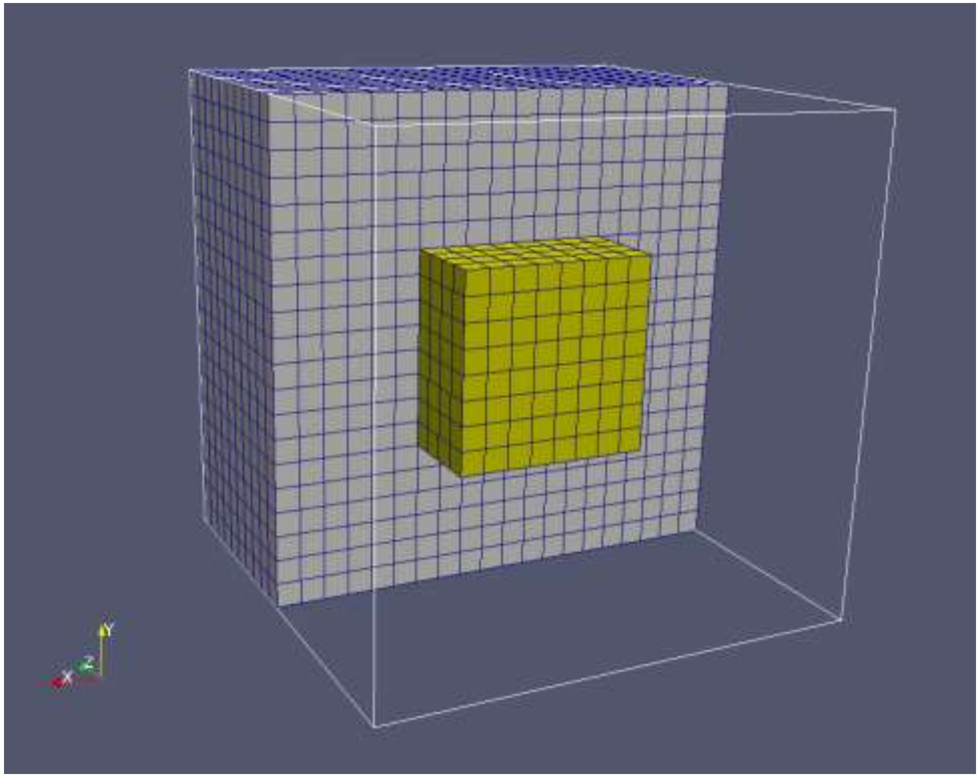
- Compute VoF field for time $t = t^{n-1} + {}^{\Delta t}/_{2}$
- For i
  - Update VoF field for time t^n
  - Update velocity and pressure
  - Relax velocity and surface force, EXCEPT for last iteration
  - Loop

- One loop: Half time step only for VoF field => computational efficiency
- Multiple loops: Semi-implicit update of the VoF field
- Larger time steps possible

SSF validation

Relaxing droplet

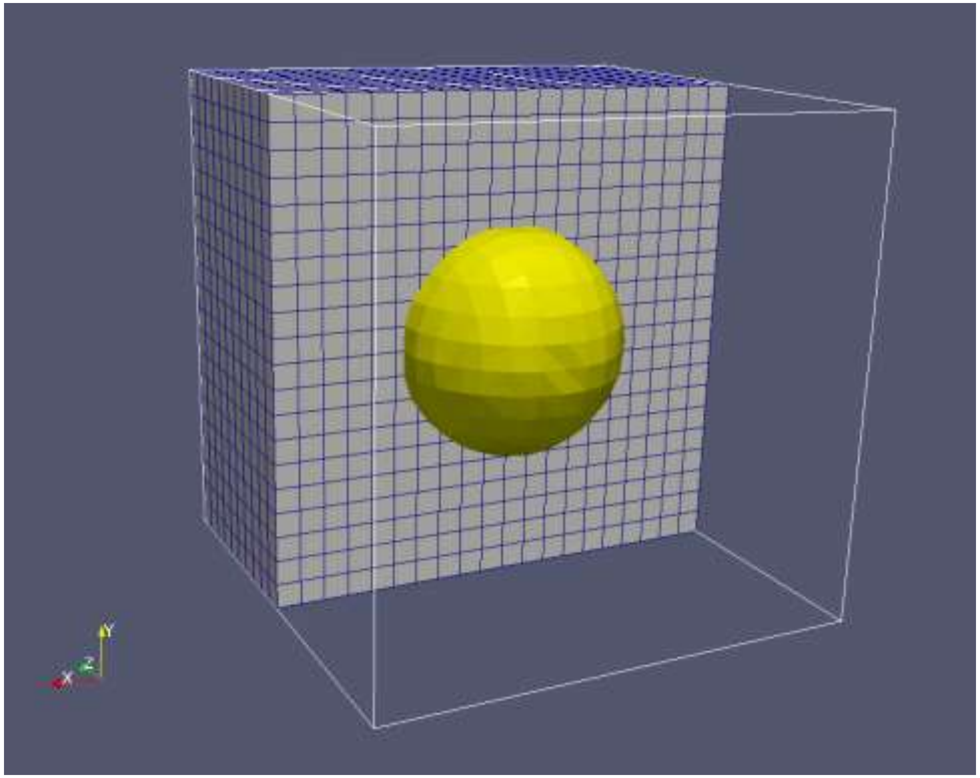
- 100 µm domain, 40 µm droplet diameter
- t=0s



SSF validation

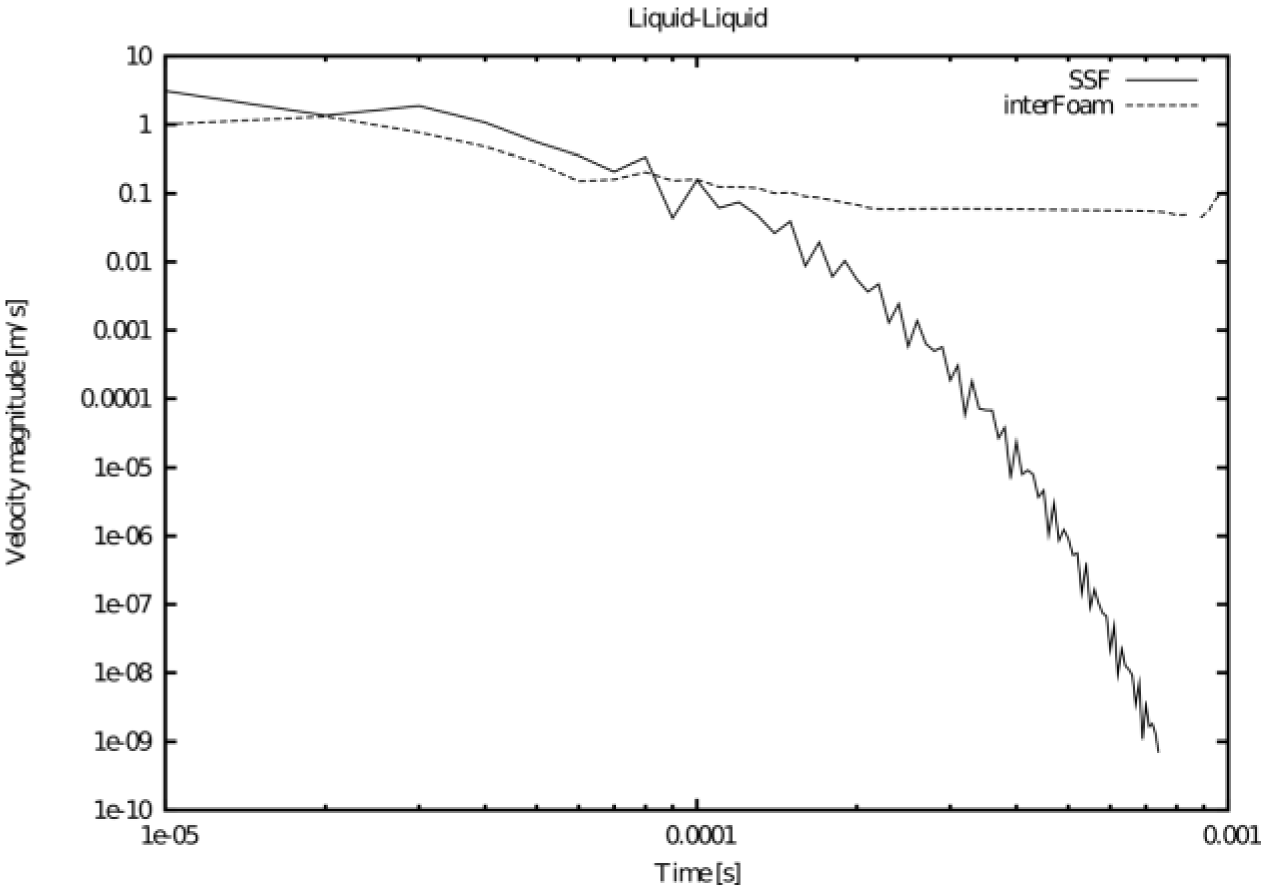
Relaxing droplet

- t=0.001s



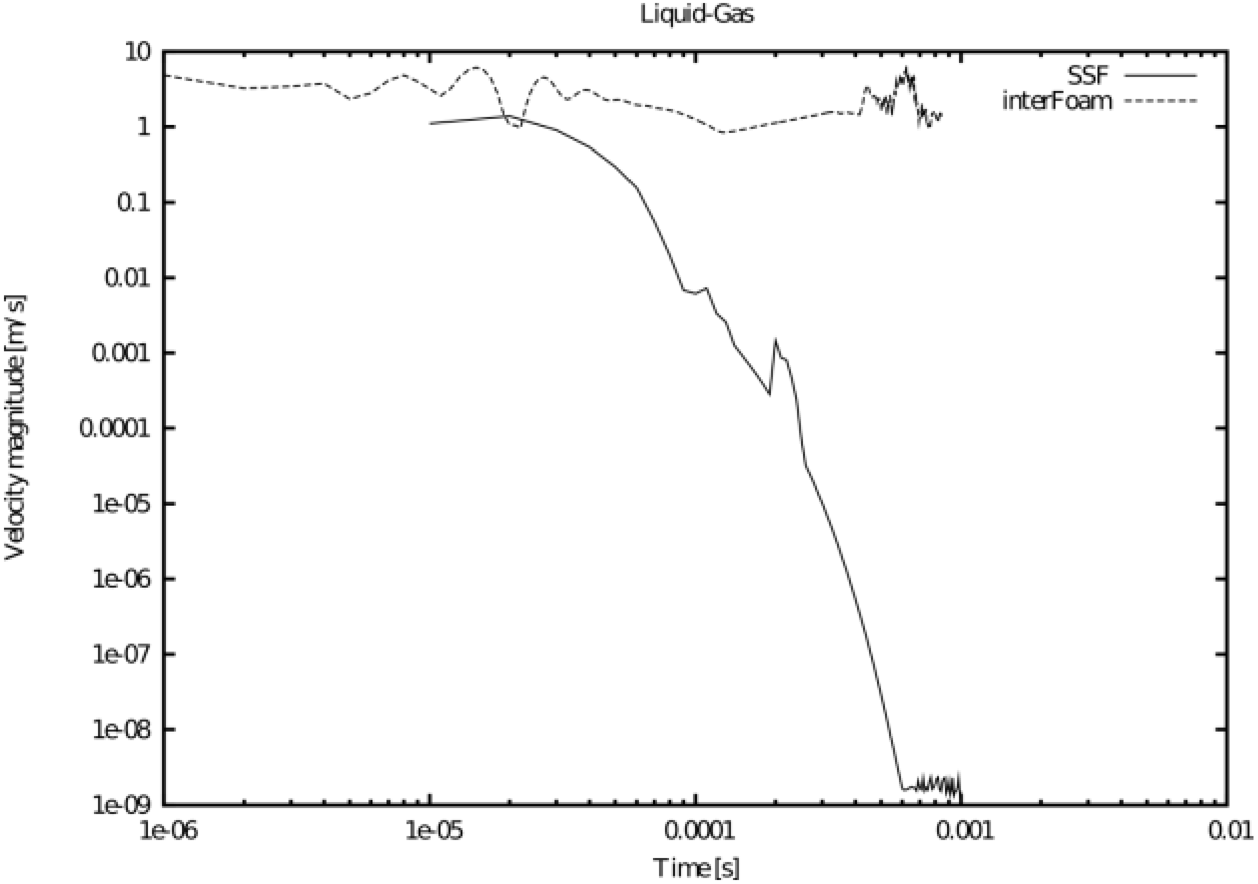
SSF validation

Relaxing droplet



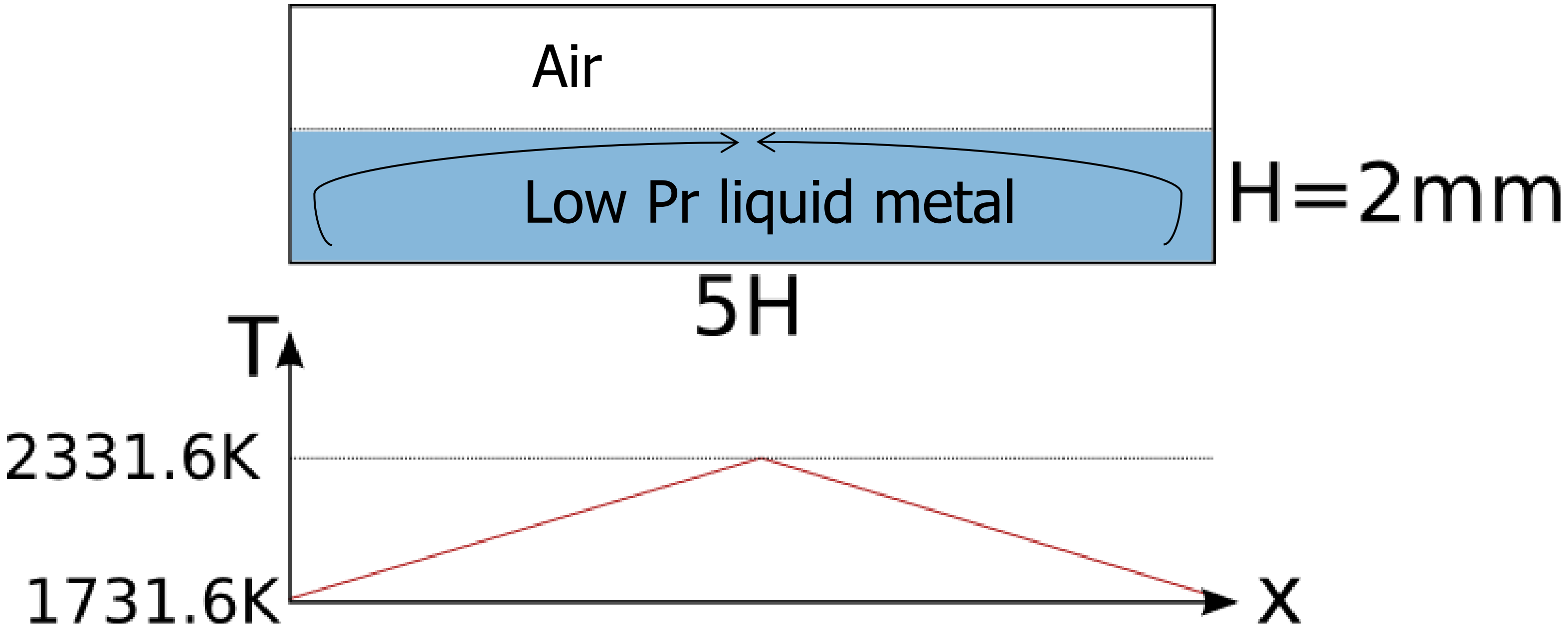
SSF validation

Relaxing bubble



SSF high surface tension test case

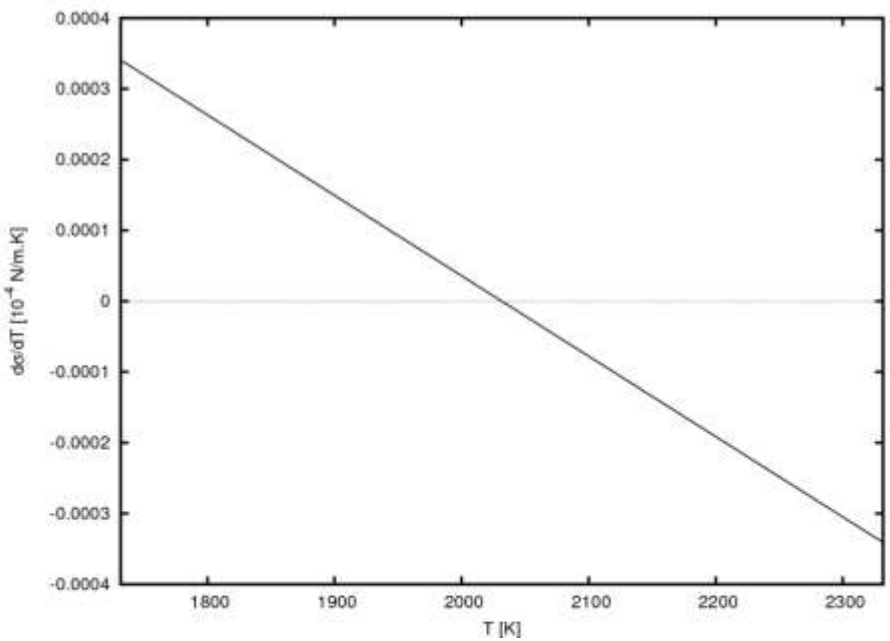
Thermocapillary cavity

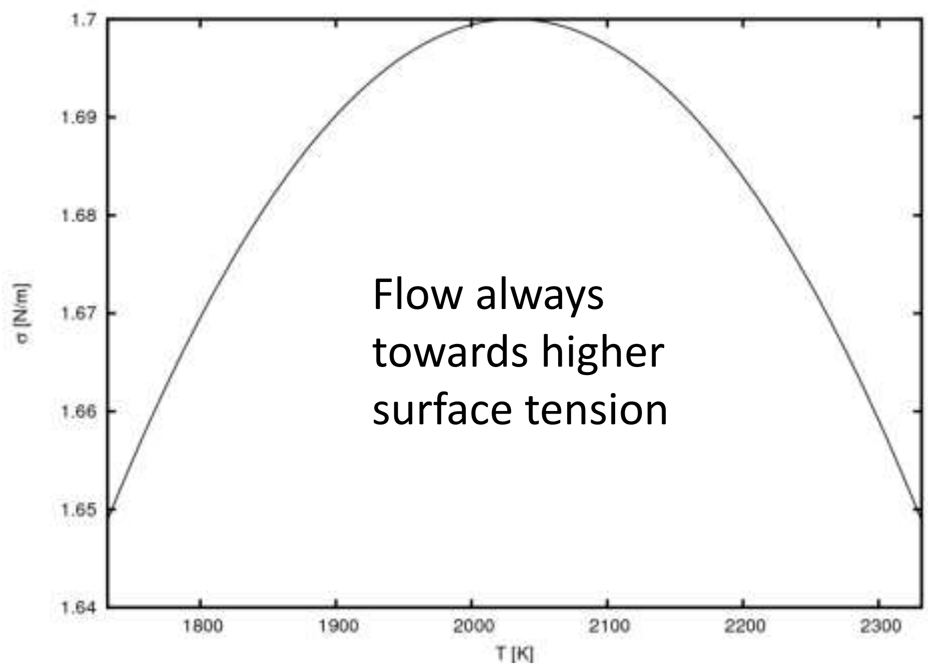


SSF high surface tension test case

Thermocapillary cavity

25x higher than water!

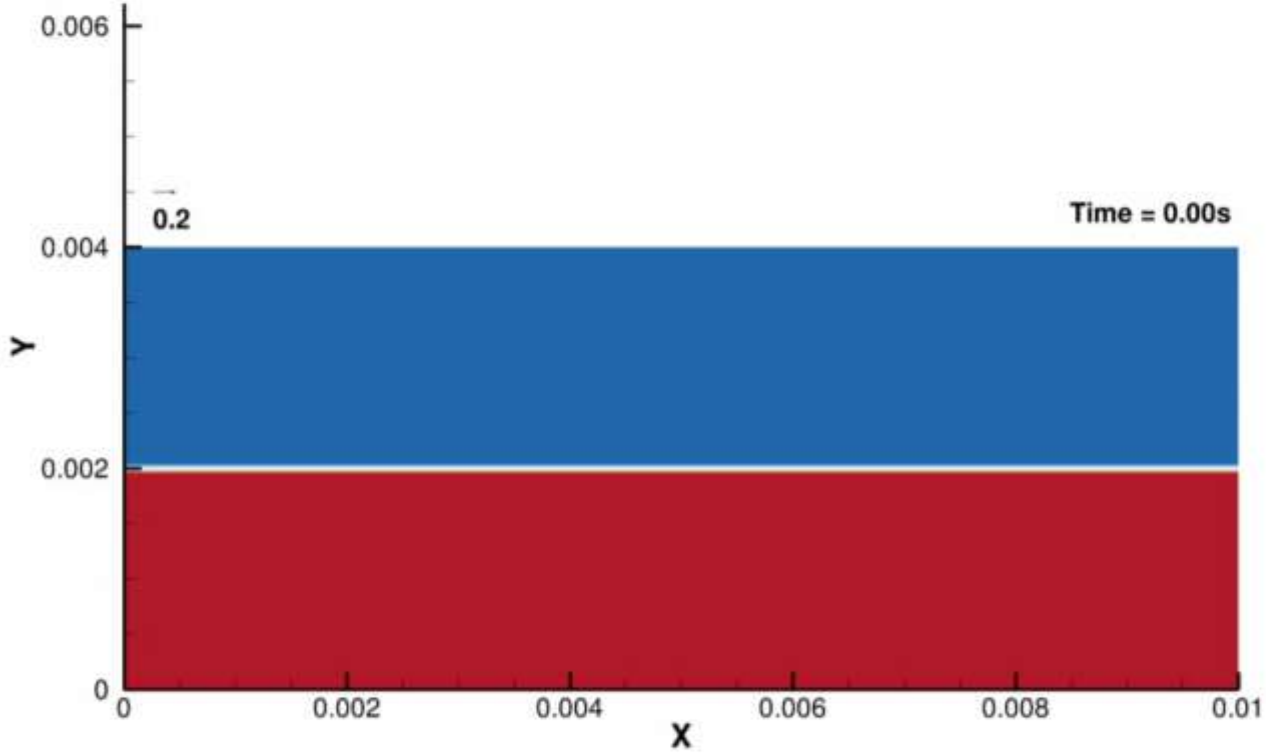




$$\vec{F} \sim \nabla \sigma$$

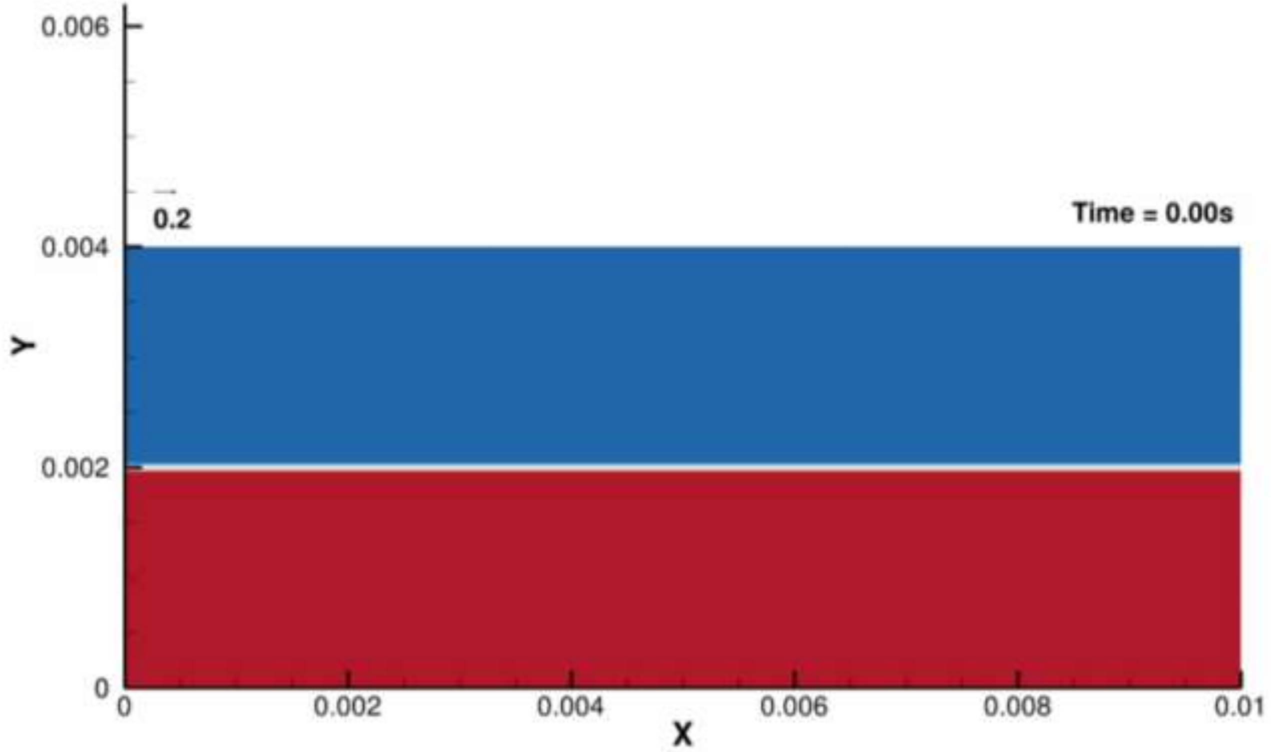
SSF high surface tension test case

Thermocapillary cavity results - interFoam



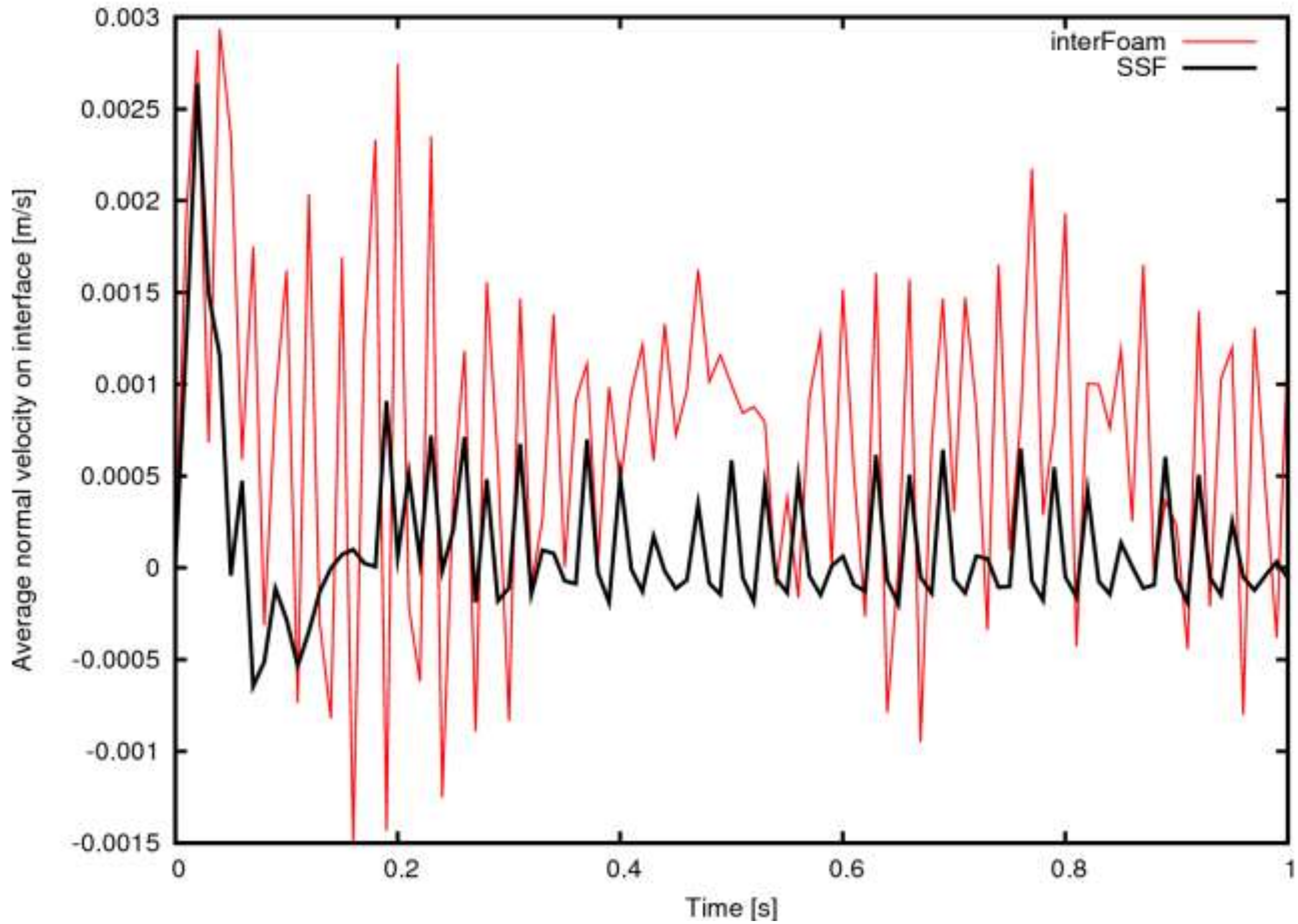
SSF high surface tension test case

Thermocapillary cavity results - SSF



SSF high surface tension test case

Thermocapillary cavity – spurious currents



Conclusion

Modeling of immiscible two phase flows

- VoF can be enhanced by a few relatively simple modifications
  - Smoothing
  - Sharp-Surface-Force
  - Explicit capillary pressure
  - Semi-implicit coupling of equations
- If you'd like to test the solver, check out my OpenFOAM version at http://code.google.com/p/interfoamssf/
- The official solver is available here: http://www3.imperial.ac.uk/earthscienceandengineering/research/perm/porescalemodelling/software/porefoam