Bubble Dynamics using Free Surfaces in a VOF framework

Sevilla Meeting: Numerical Challenges in two-phase flows
27 October 2014

Léon MALAN

Supervisor:
Prof. Stéphane ZALESKI

DE LA RECHERCHE À L'INDUSTRIE
cea

ccrt
Centre de calcul recherche et technologie

UPMC
SORBONNE UNIVERSITÉS

Contents

- Introduction: Simulation problem
- PARIS code
- Free surface condition
  - Pressure treatment and surface tension
  - Velocity extrapolation
  - Ensuring mass (volume) conservation
- Test cases
- Conclusion: Future work

Problem Background

- Experiments performed on “micro-spalling”
- Metal sample subjected to intense shock wave
  - Laser irradiation, explosive detonation
- Cavitation in melted metal dominant failure mechanism, (Signor et al, 2008).

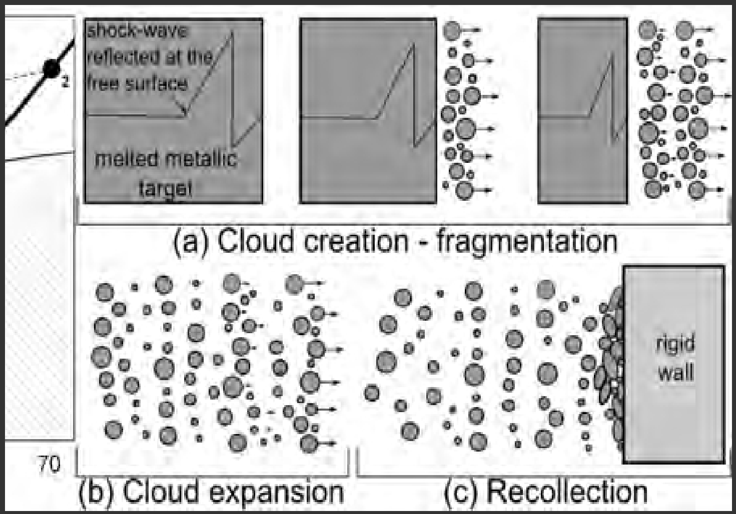
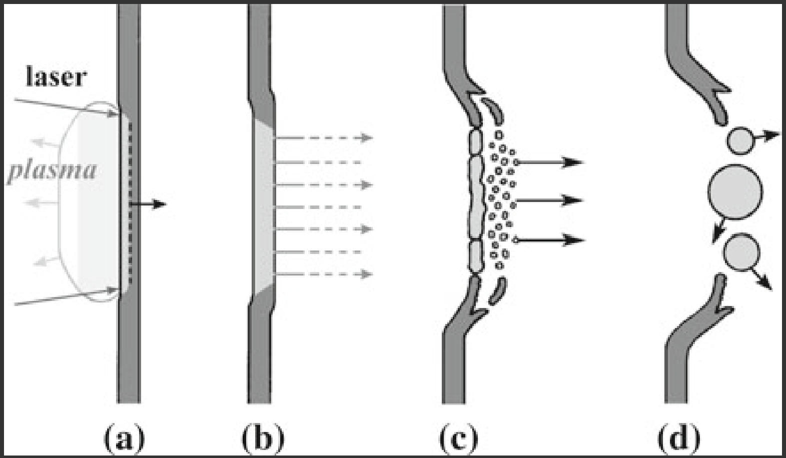


Illustration of shock loading on metal sample. Signor et al. (2008).



Schematic representation of response of metallic foil to laser shock. De Rességuier et al. (2010).

Problem Background

- Experiments performed on “micro-spalling”
- Metal sample subjected to intense shock wave
  - Laser irradiation, explosive detonation
- Cavitation in melted metal dominant failure mechanism, (Signor et al, 2008).

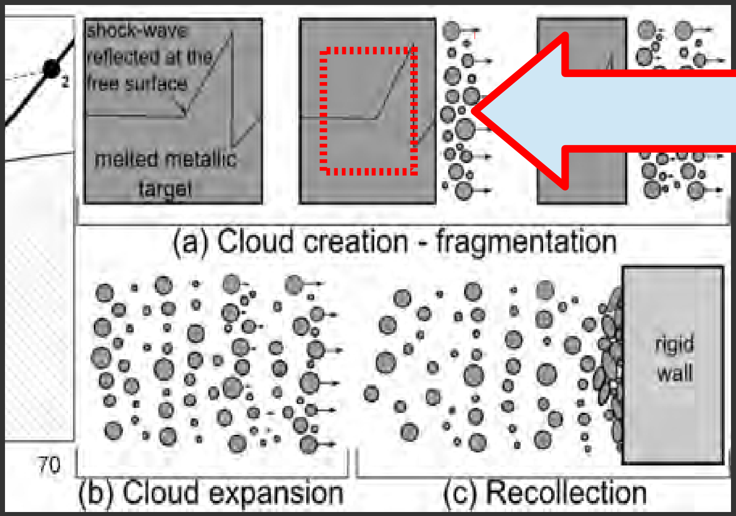
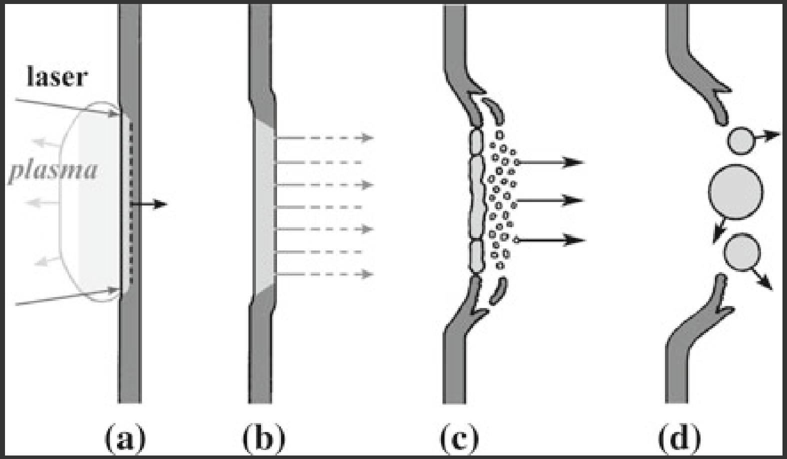


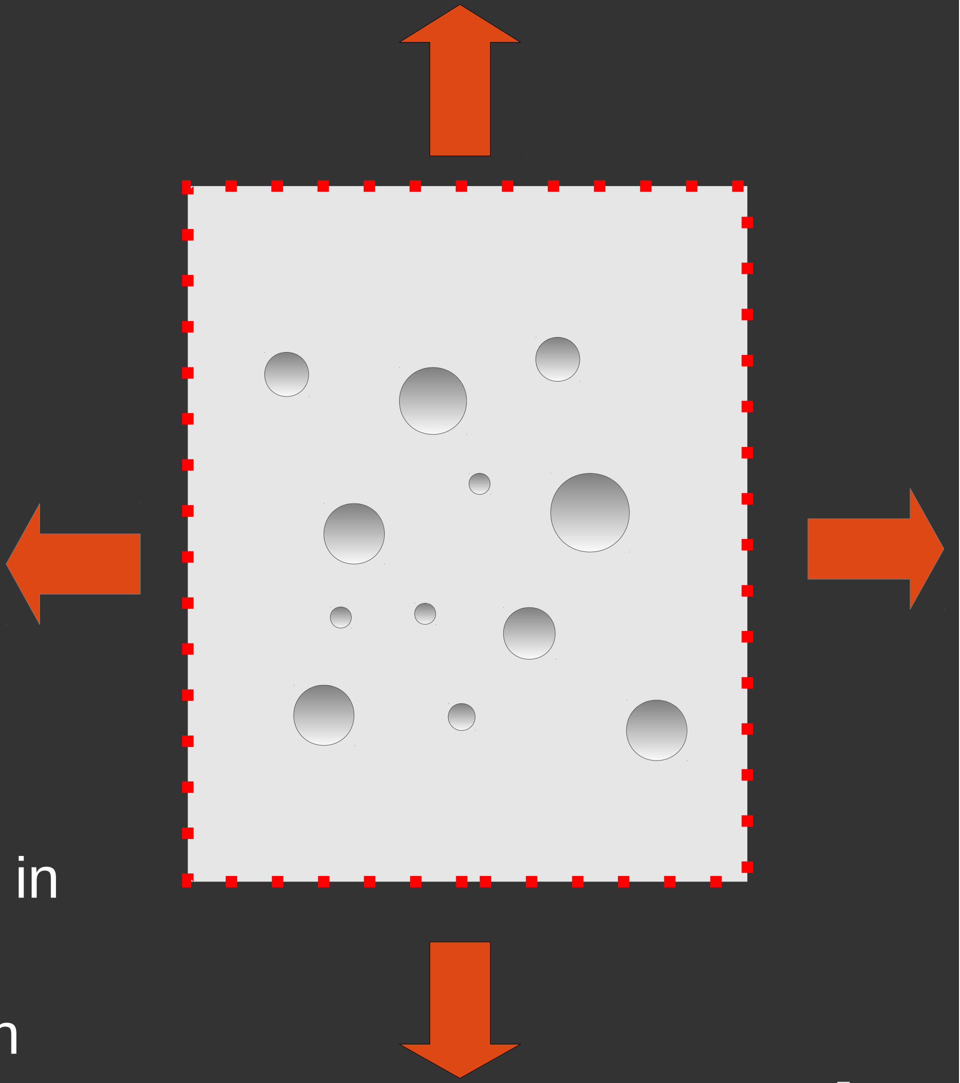
Illustration of shock loading on metal sample. Signor et al. (2008).



Schematic representation of response of metallic foil to laser shock. De Rességuier et al. (2010).

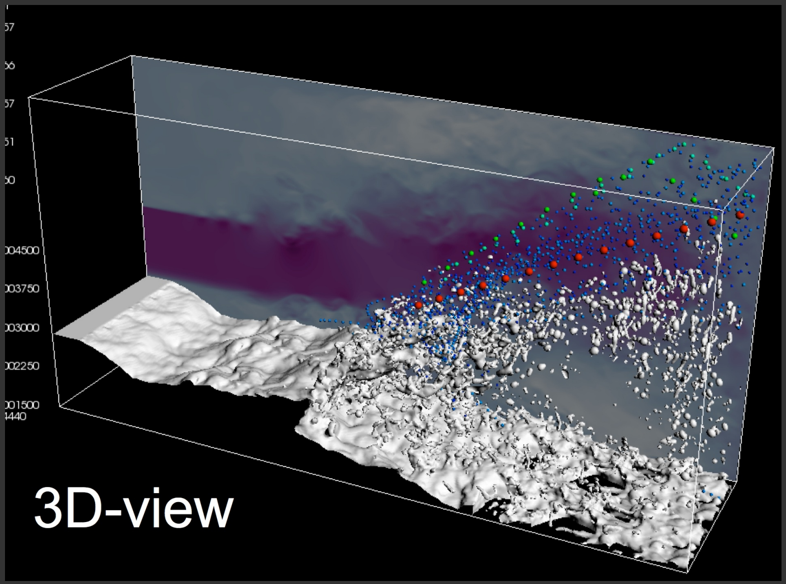
Problem Background

- Goal is to simulate the flow inside the liquid metal
- Flow characteristics:
  - Low Mach: incompressible
  - High Re: inviscid
  - High density ratio: no shear at interface
  - Adiabatic: no temperature effects
  - No mass transfer at interface
  - Void fraction < 20%: Not interested in mousse
  - Random bubbles already present in liquid



PARIS code introduction

- **<u>PA</u>**rallel **<u>R</u>**obust **<u>I</u>**nterface **<u>S</u>**imulator
- VOF and Front Tracking code based on SURFER and FTC3D2011
- Written entirely in Fortran90
- Released as a free code under GPL
- Code website:
  - http://www.lmm.jussieu.fr/~zaleski/paris/index.html



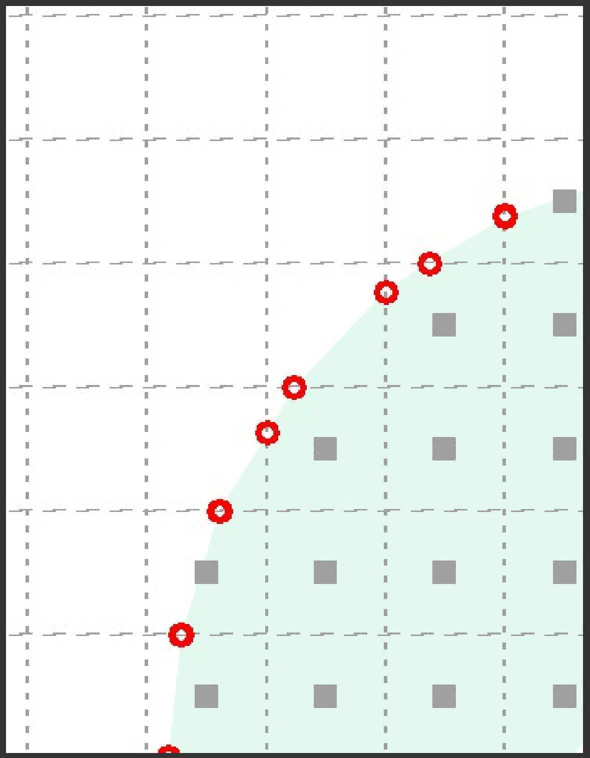
3D visualization of atomization sim.
Yue Ling using PARIS (2014).

PARIS code: Numerics

- Finite Volume/ Finite Difference on a Cartesian Grid
- MAC (staggered) grid.
- Time schemes : 1) first order Euler 2) second order Runge-Kutta for NS part
- Advection terms by QUICK, ENO or Upwind
- Explicit or implicit viscous terms.
- VOF – PLIC for interface reconstruction and advection either with CIAM or the conservative VOF of Yue & Weymouth.
- Surface tension by height functions (as in Gerris).
- Parallel computation with MPI: 2 ghost layers

Code modification: Free Surface

- For bubble dynamics problem, implement a free surface interface condition (Harlow & Welch, 1966)
- Constant pressure inside bubbles
- Incompressible liquid
- Surface tension at interface: replace CSF with pressure jump



Quick re-cap: Projection method

$$\rho^n \left(\frac{\boldsymbol{u}^* - \boldsymbol{u}^n}{\Delta t} + \boldsymbol{u}^n \cdot \nabla^h \boldsymbol{u}^n \right) = \nabla^h \cdot (2\mu_n \boldsymbol{D}^n) + (\sigma \kappa \delta_s \boldsymbol{n})^n$$

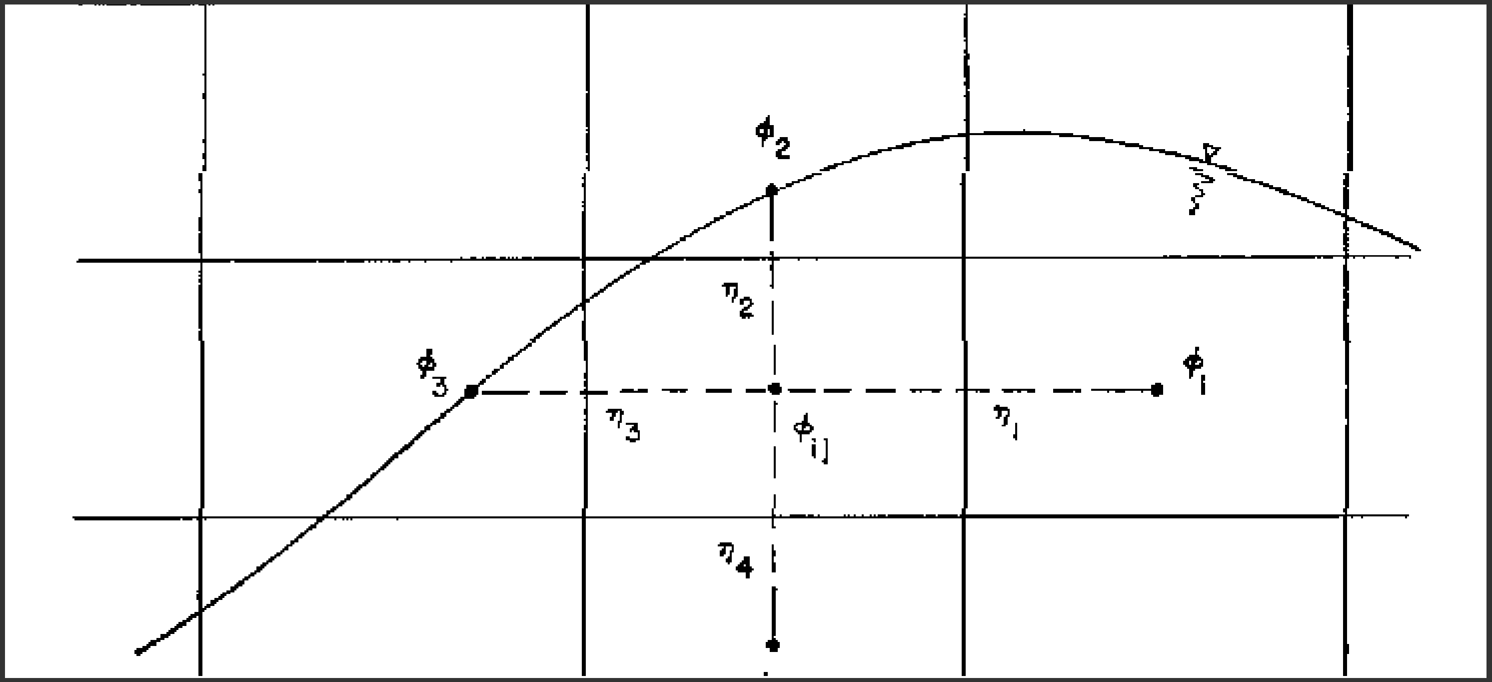
$$\frac{\boldsymbol{u}^{n+1} - \boldsymbol{u}^*}{\Delta t} = -\frac{1}{\rho^n} \nabla^h P^{n+1}$$

$$\nabla^h \cdot \boldsymbol{u}^{n+1} = 0$$

$$\nabla^h \cdot \left[\frac{\Delta t}{\rho_n} \nabla^h P^{n+1} \right] = \nabla^h \cdot \boldsymbol{u}^*$$

Pressure treatment, surface tension

- Pressure solved in liquid using asymmetric "branches"
  - Chan and Street, 1970, SUMMAC



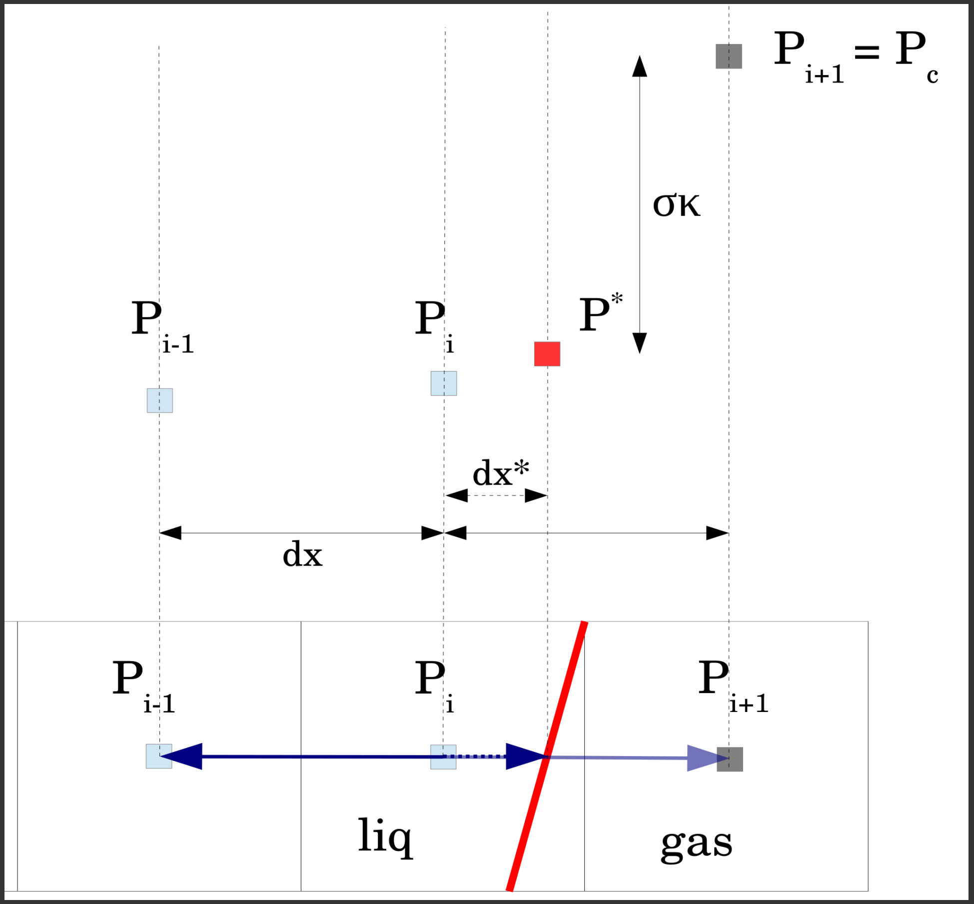
Pressure treatment, surface tension

- Pressure solved in liquid using asymmetric "branches"

$$\nabla \cdot \left[\frac{dt}{\rho} \nabla P \right] = \nabla \cdot \vec{u}^*$$

$$\nabla^h P_{i+1/2} = \frac{P_{i+1} - P_i}{\delta x}$$

$$\nabla^h P_{i+1/2} = \frac{P^* - P_i}{\delta x^*}$$

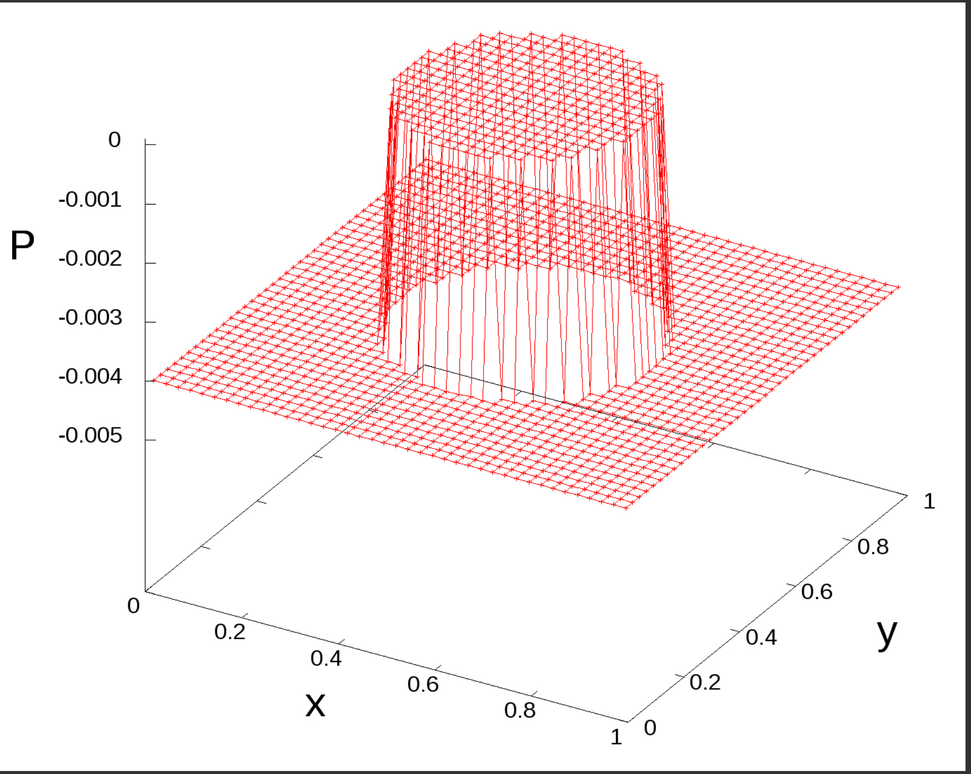


Pressure treatment, continued

- CSF not valid
- Surface tension added as an interface pressure jump

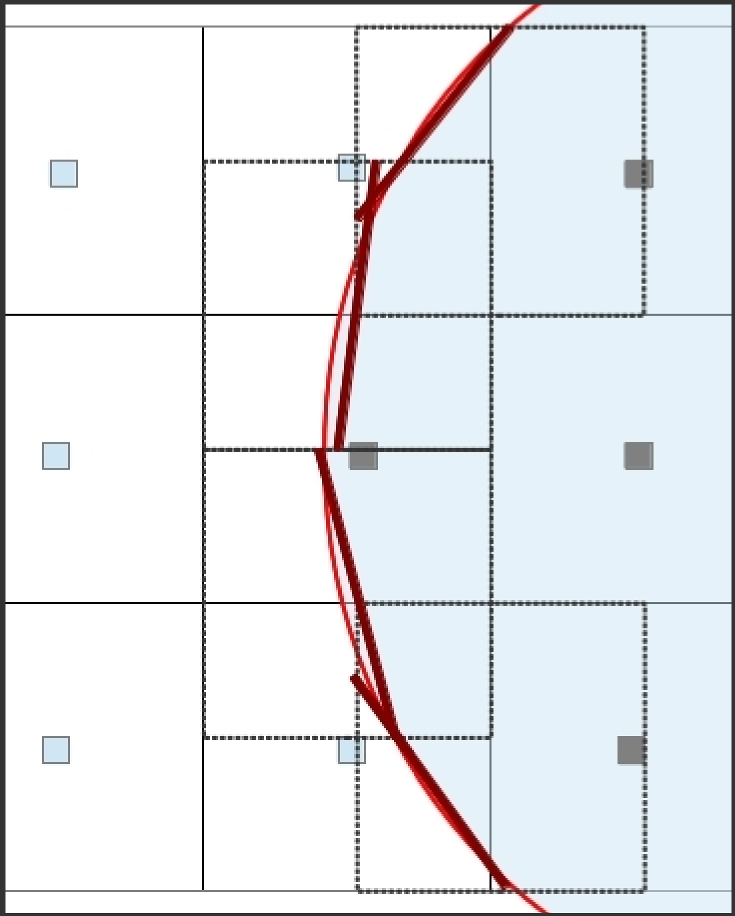
$$P^* = P_c - \sigma\kappa$$

$$\nabla^h P_{i+1/2} = \frac{P^* - P_i}{\delta x^*}$$

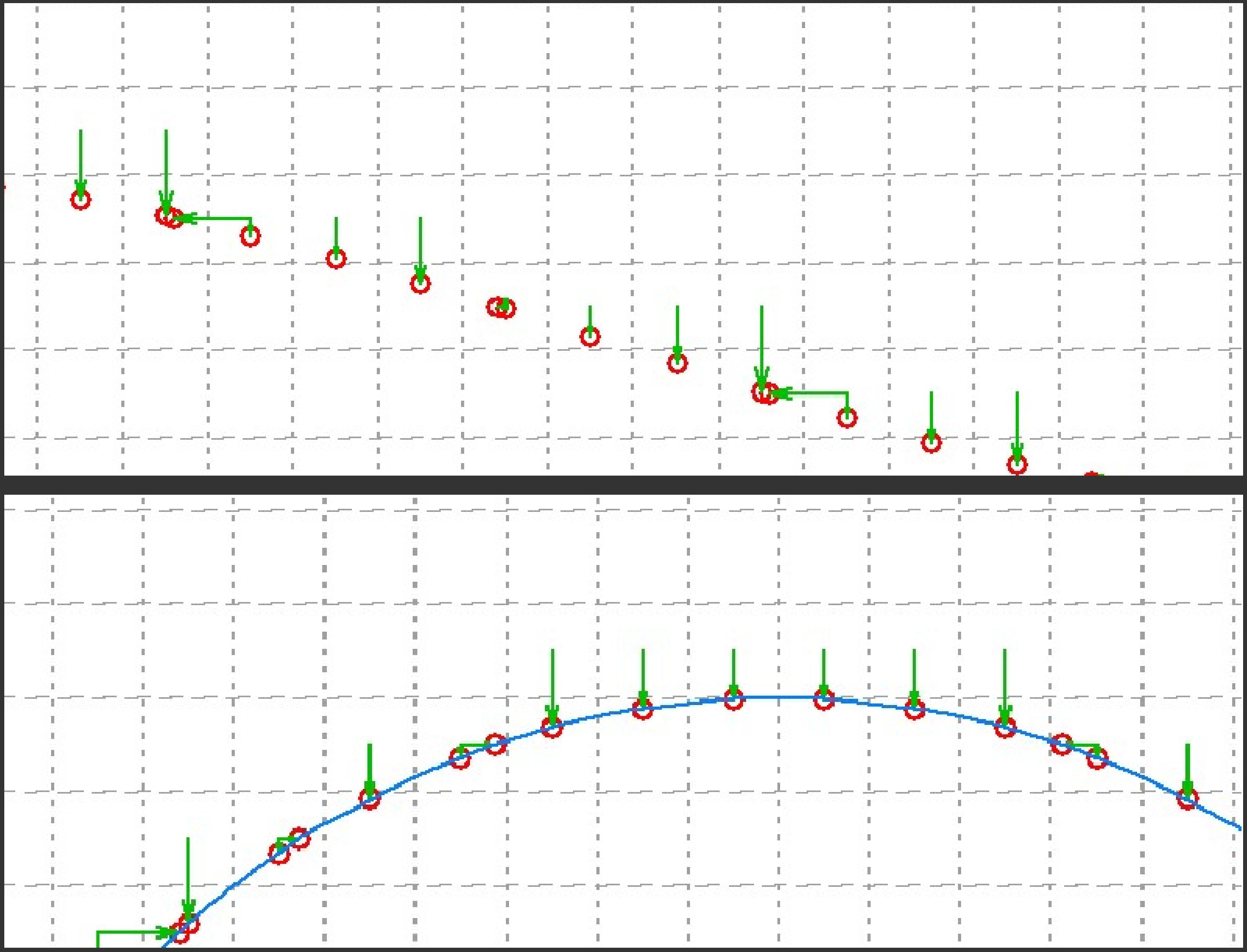


Locating the interface

- Asymmetric Poisson branches
  - Staggered VOF mesh used to calculate interface position
  - New VOF calculated using halves of neighbouring cells
  - Plane intersection with connecting branch taken to calculate modified length

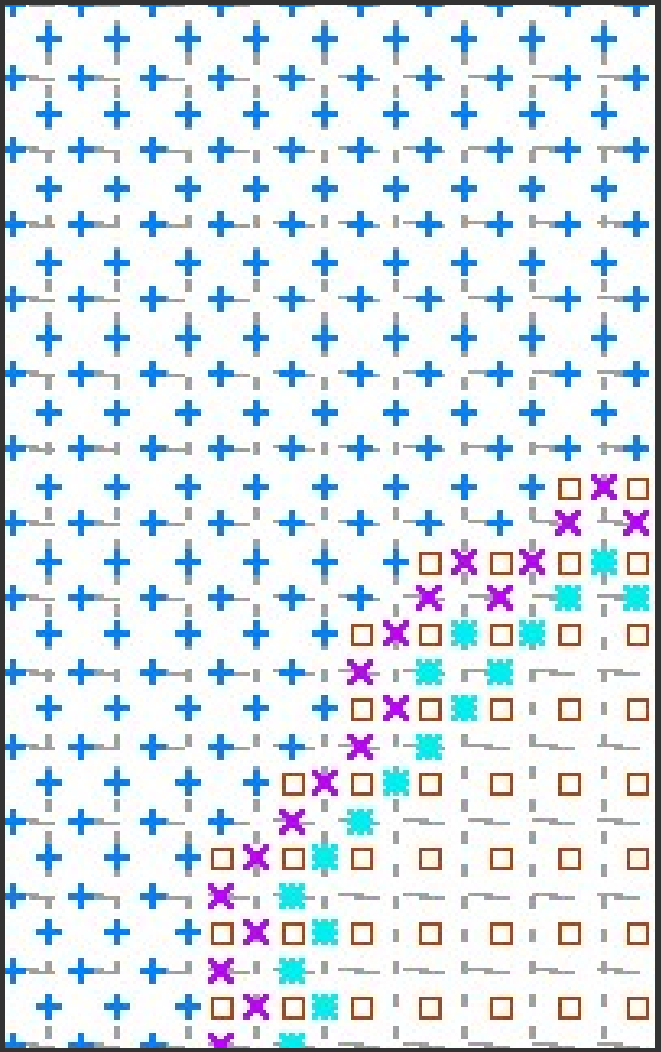


Examples: Modified lengths



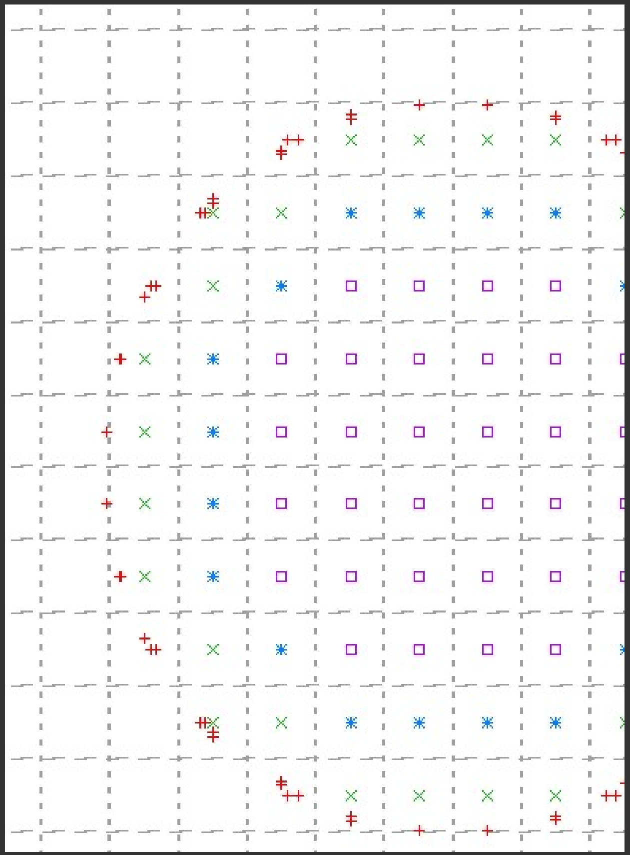
Boundary condition on velocity

- Needed for boundary cells in liquid:
  - Momentum advection
- Introduced topological description of domain
- All liquid and first neighbour (level 0) velocities are solved by standard projection method
- First order velocity extrapolation, topology wise.



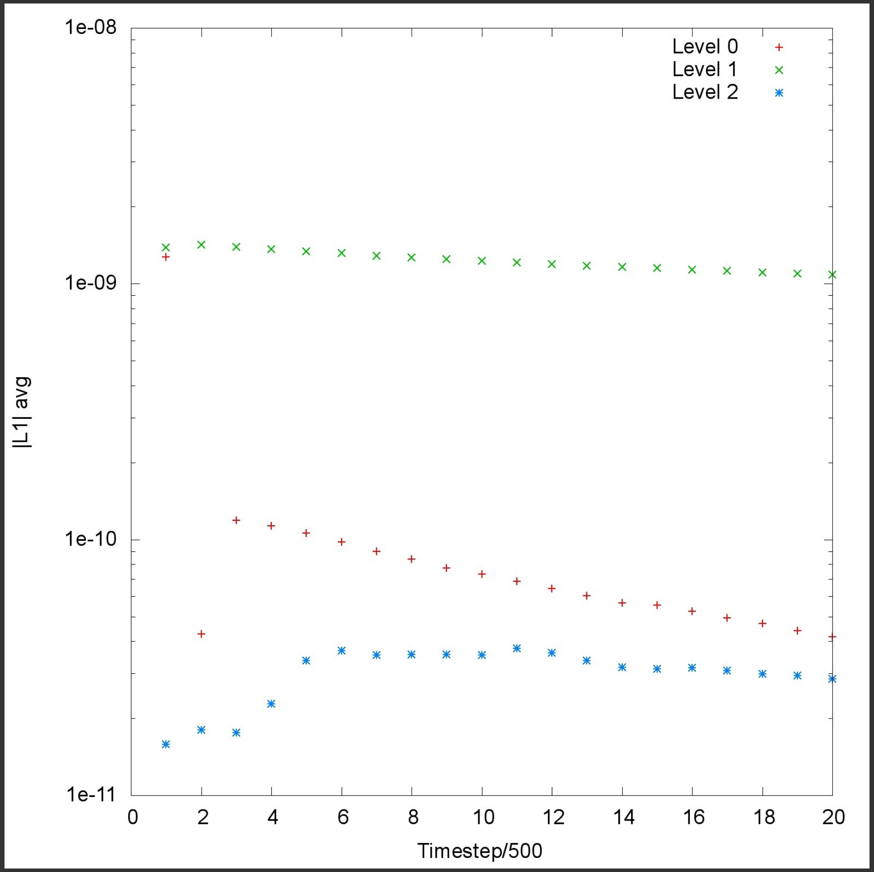
Volume conservation: 2nd projection

- Implemented 2nd projection step to ensure volume conservation to within defined limit
  - A “phantom” pressure is obtained inside first two cells inside cavity
  - Velocities level 1 and 2 are corrected using this pressure



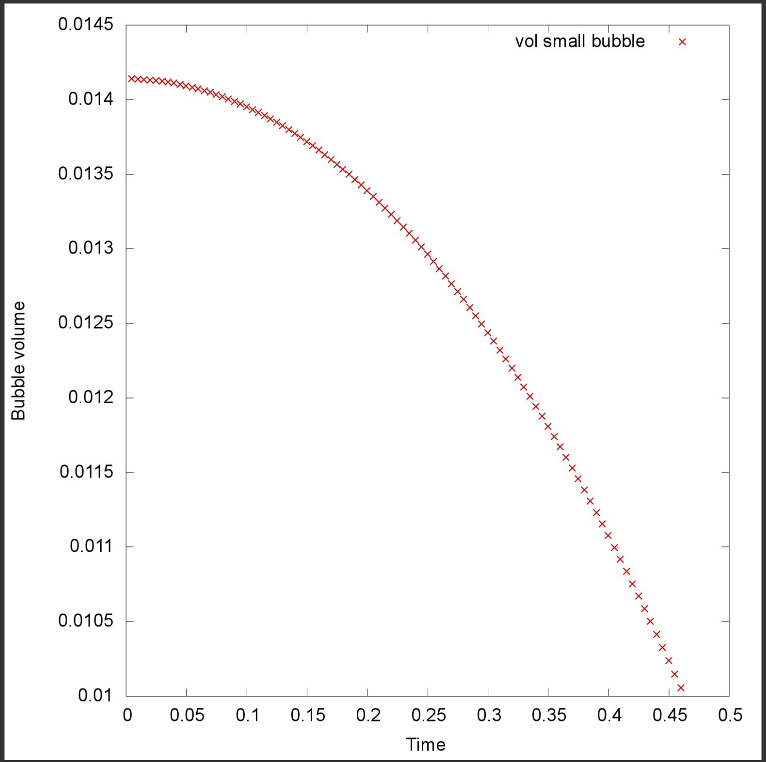
Test case: Volume conservation

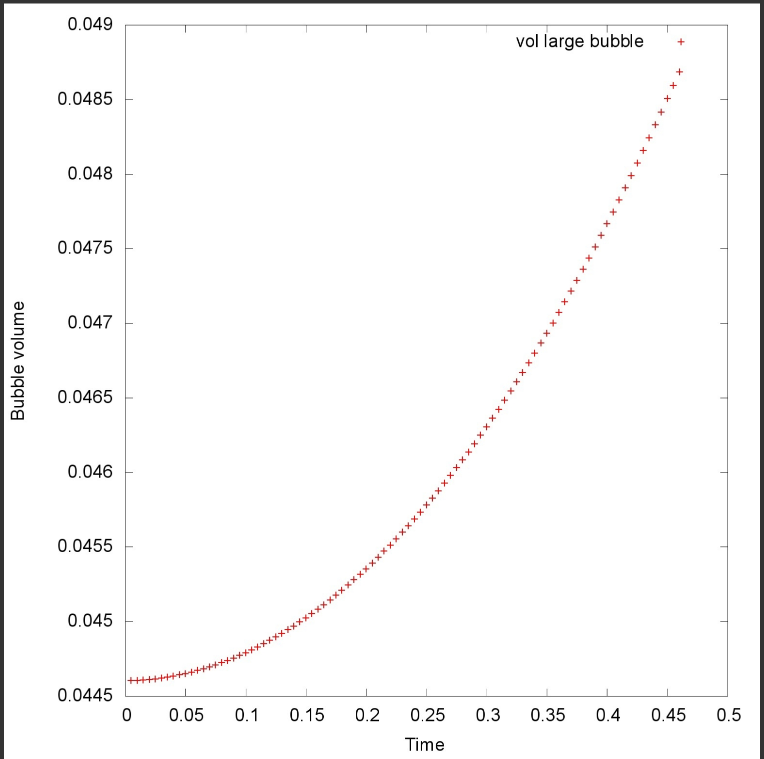
- Mass conservation on single expanding cavity in 3D
- Test run on Airain, 64 processors (4 per coord direction)
- Domain 128^3
- Running time: ~42 minutes
- Measured divergence at topological levels 0 (liq) 1 and 2 (cavity)



Two interacting bubbles

- Two different sizes bubbles initialised in periodic domain
- Bubble “competition” observed

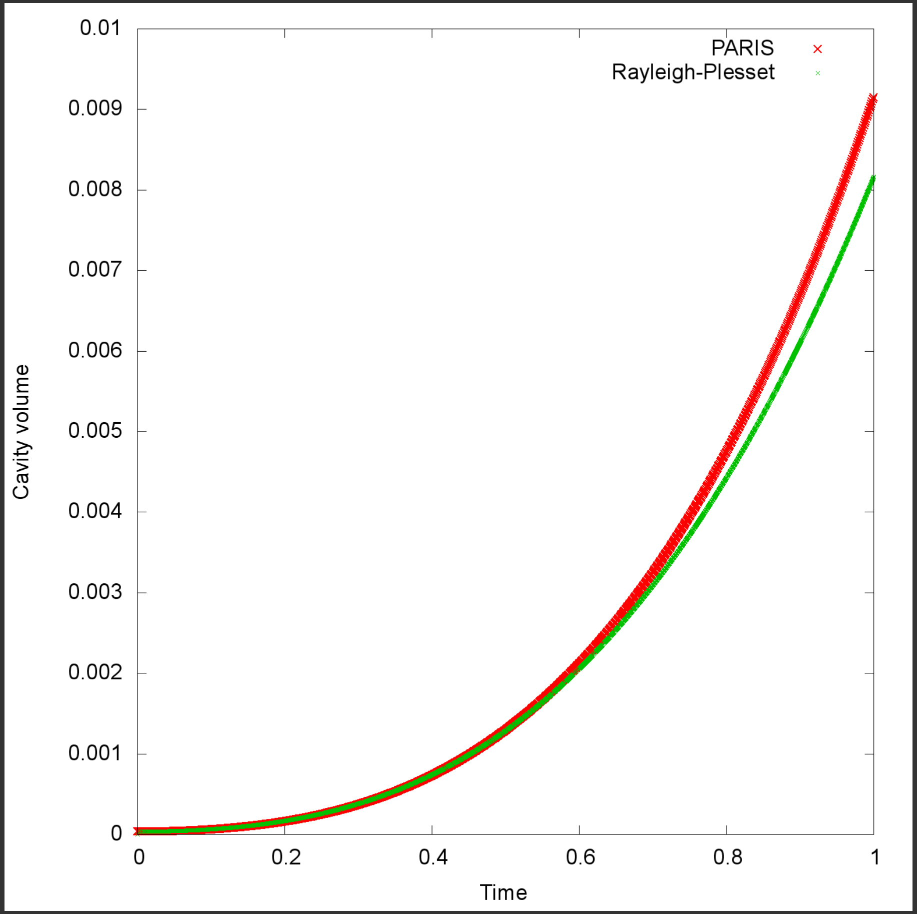




Rayleigh-Plesset bubble expansion

- Bubble radius versus time
- Constant gas pressure
- Fixed P_∞ at domain boundary

$$\ddot{R}R + \frac{3}{2}\dot{R}^2 = \frac{P_R - P_\infty}{\rho_l}$$
$$= \frac{P_c - \frac{2\sigma}{R} - P_\infty}{\rho_l}$$



Looking ahead

- Error quantification for Poisson solver
- 2^{nd} order velocity extrapolation
  - Least square fit through neighbouring cells
- Deal with imploding bubble: can add volume source when extrapolation layers are no longer available inside cavity
- Implement a coordinate change
  - Additional term(s) to be added to momentum equation
- Implement and test in *Basilisk* ?