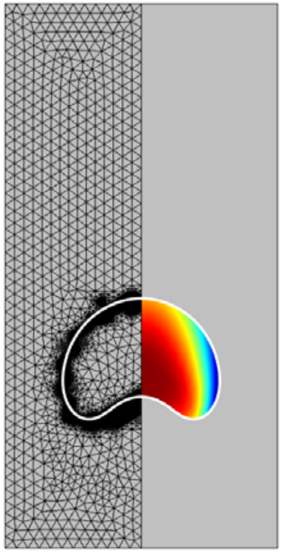
SIMTEC

Numerical modeling
that predicts, optimizes and innovates

Adaptive mesh refinement: Quantitative computation of a rising bubble using COMSOL Multiphysics®



T.Preney, J.D. Wheeler and P.Namy

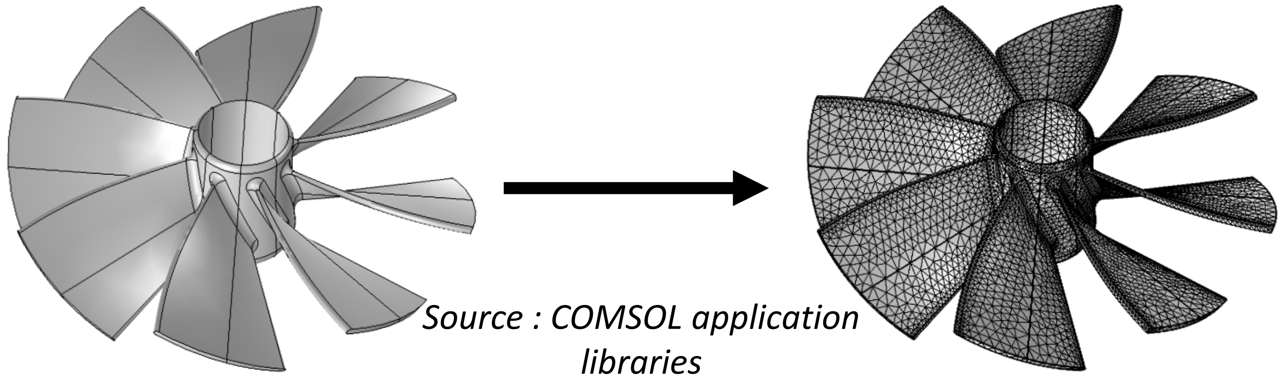
SIMTEC- (+33) 9 53 51 45 60

patrick.namy@simtecsolution.fr

COMSOL CONFERENCE 2018 BOSTON

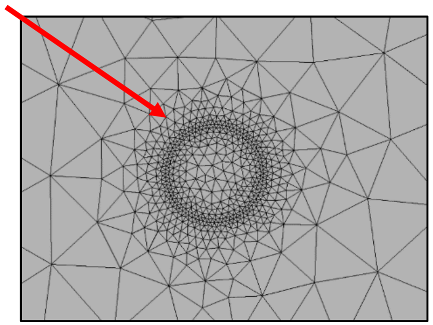
Need to accelerate your calculations ?

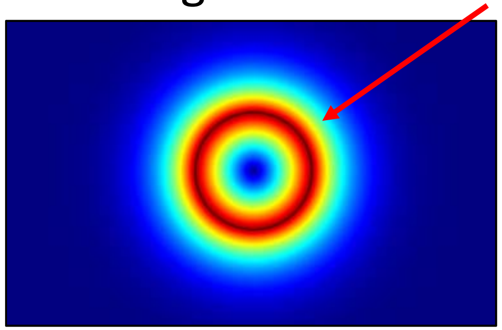
- The mesh : fundamental pillar of numerical computation on which the approached solution is built



Source : COMSOL application libraries

- A high concentration of nodes is needed where the gradients are important

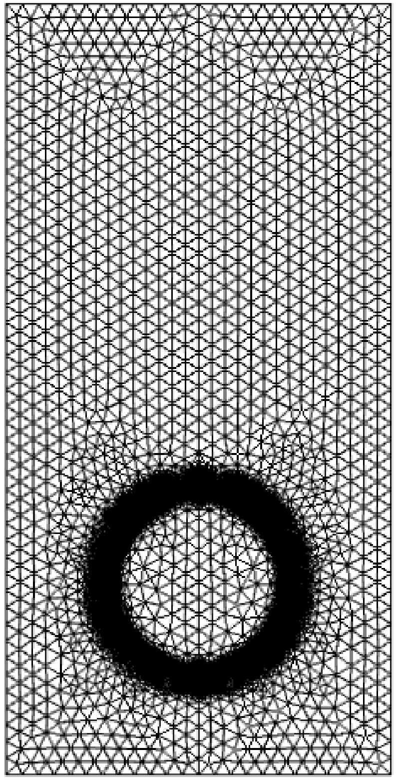




→ May induce large computational times !

Introduction

Definition : To adapt the mesh to the solution as time goes by



→ More efficient computation

Fast   Precise

Working with SIMTEC

Industry Challenges

- R&D sections: experts in their field
  → Expertise in numerical modelling?
- Lack of time
- FE modelling performed by a small group of people

SIMTEC's Solutions

- Numerical modelling project
  - → SIMTEC's member as your colleague
  - → Help improve your modelling knowledge!
  - → Cost-effective outsourcing

Our team & Our clients

Numerical modelling / simulation consultants

COMSOL Certified Consultant

6 members all EngD + PhD

- Extensive research background
- Complex problems / various fields of expertise

Successful track record:

- Big compagnies
- Government laboratories

Involved in research consortia

- EU funded projects (REEcover / SHARK)
- PhD projects supervision.

SHARK INDUSTRIALISING LASER FUNCTIONAL TEXTURING

www.simtecsolution.fr/en

Plan

I. General principle

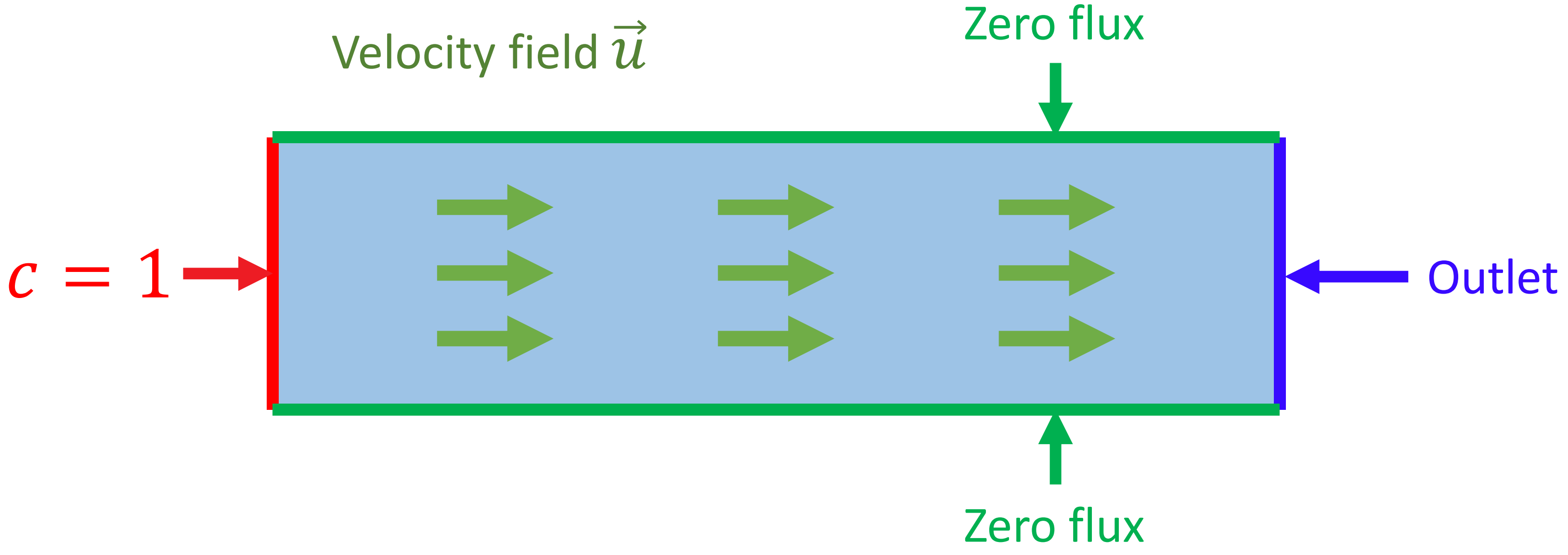
II. 2D validation study

III. 3D validation study : comparison with other softwares

Introduction

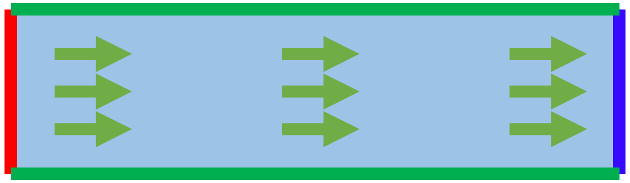
Especially useful for a time-dependent study !

Example: transport of a concentration in water

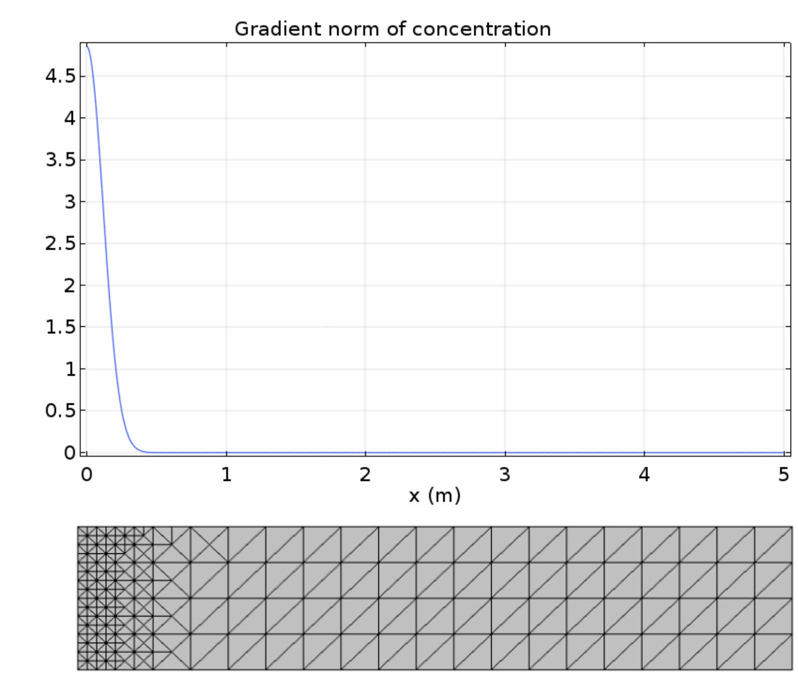


→ Concentration front propagation

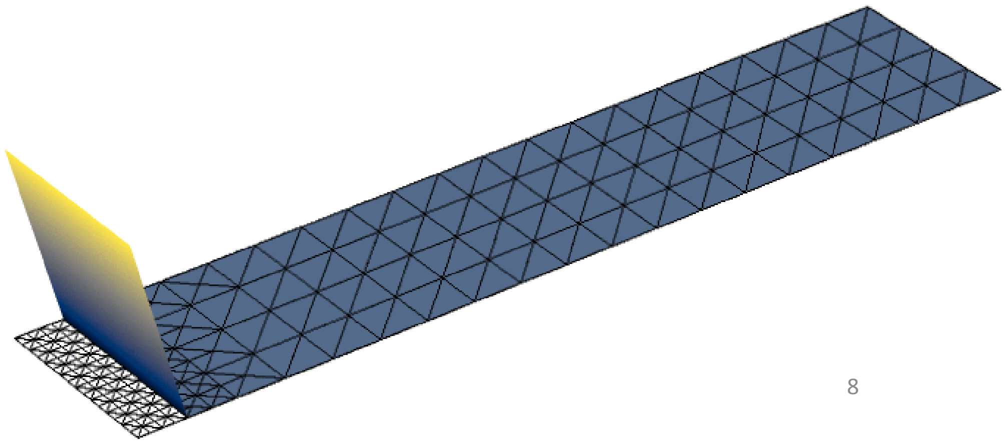
Introduction



Idea : Refine the mesh where the concentration gradient norm is important



About the concentration:



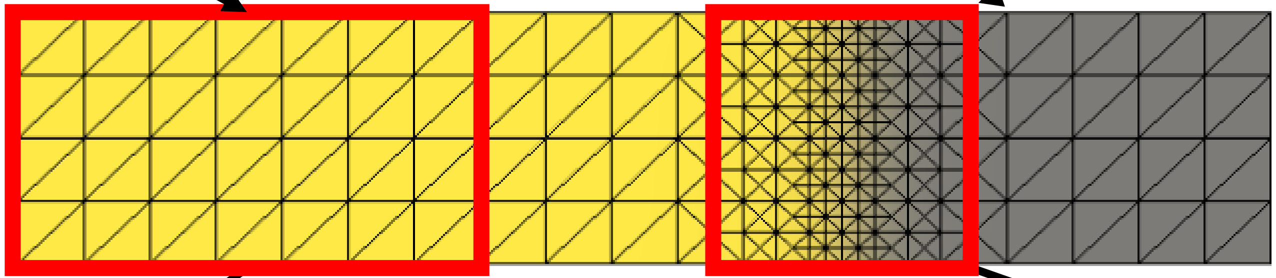
I – General principle

Avantages

2 fundamental points

Coarse mesh

Refined mesh



Few nodes : low impact on computational time

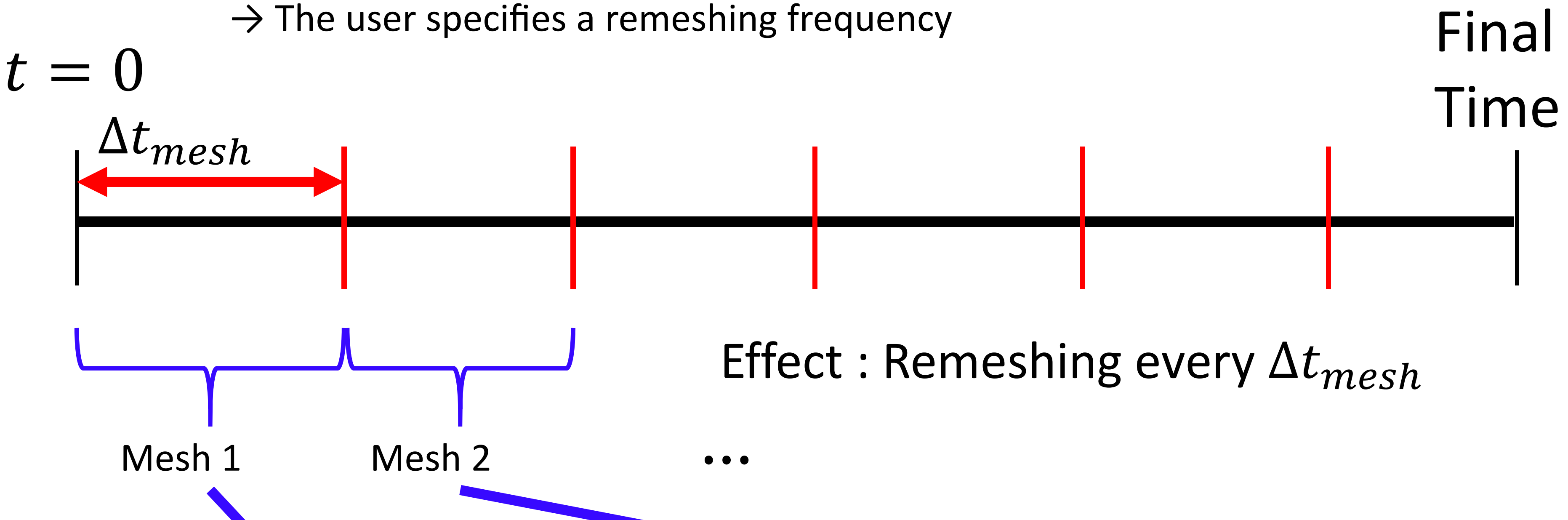
Gain in accuracy : the mesh is finer where it matters

Calculation both fast and precise !

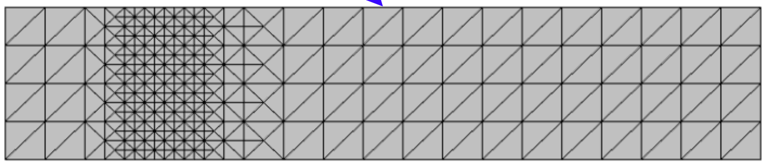
I – General principle

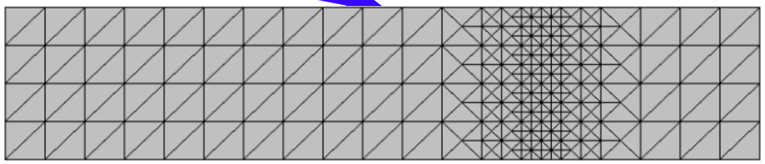
Question : How will the mesh evolve?

→ The user specifies a remeshing frequency



Effect : Remeshing every Δt_{mesh}



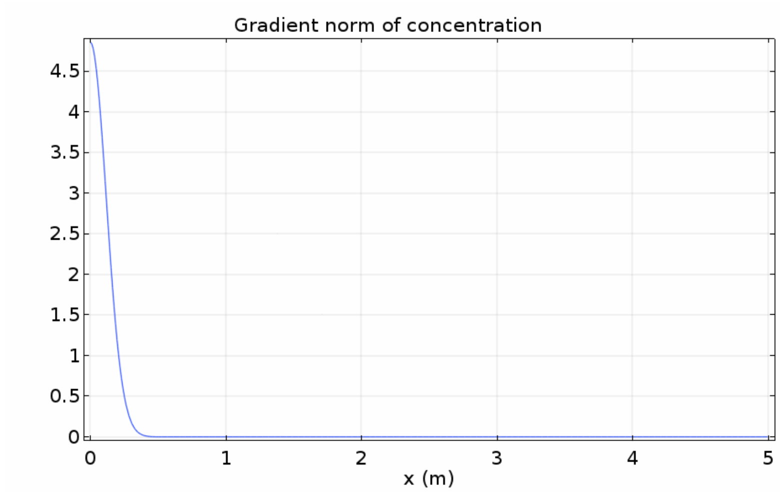


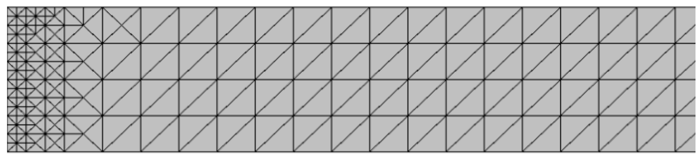
I – General principle

Question : Where will the mesh be adapted?

→ The user specifies an error indicator (usually a gradient norm)

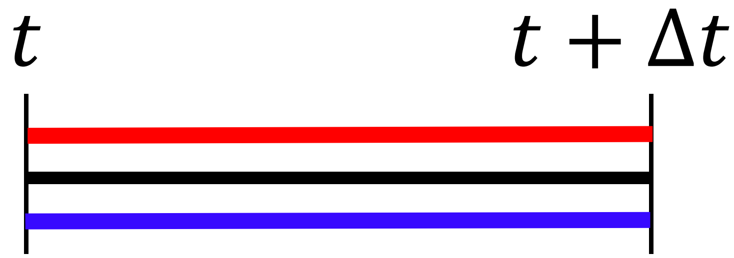
Effect : Mesh refinement where the error indicator function is important





I – General principle

Double calculations sweep



1. First calculation : estimation of the error indicator on the coarse mesh
   To determine spatial areas where the indicator is important

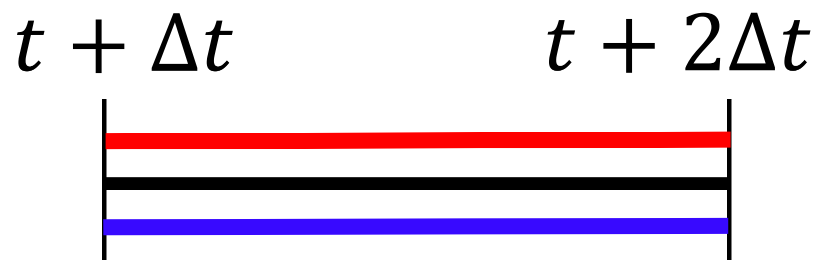
2. Mesh refinement on those areas

3. Second calculation : computation of the solution on the (now) refined mesh

4. Back to step 1

I – General principle

Double calculations sweep



1. First calculation : estimation of the error indicator on the coarse mesh

2. Mesh refinement where the error indicator is important

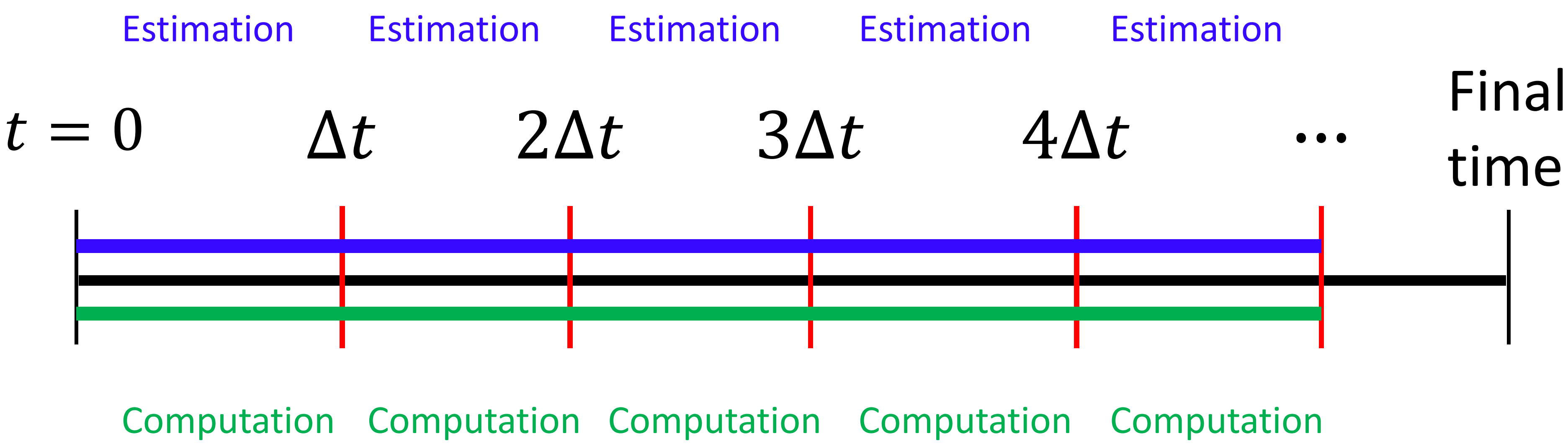
3. Second calculation : computation of the solution on the refined mesh

4. Back to step 1 at the end of the time interval

I – General principle

Double calculations sweep

1. Estimation : low precision calculation on coarse mesh
2. Mesh adaptation
3. Precise calculation on refined mesh

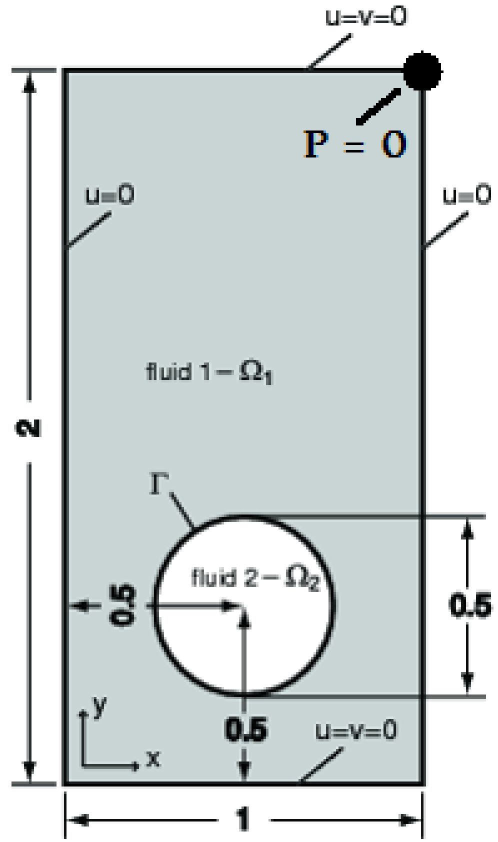


II – 2D Study

Public benchmark available at
http://www.featflow.de/en/benchmarks/cfdbenchmarking/bubble.html

Reference paper:
Hysing, S.; Turek, S.; Kuzmin, D.; Parolini, N.; Burman, E.; Ganesan, S.; Tobiska, L.: Quantitative benchmark computations of two-dimensional bubble dynamics, International Journal for Numerical Methods in Fluids, Volume 60 Issue 11, Pages 1259-1288, DOI: 10.1002/fld.1934, 2009

II – 2D Study



Extract : reference paper

Configuration

Rise of a bubble of gas inside a fluid

- 2D geometry
- Laminar flow modelled by Navier-Stokes equations
- Two-phase flow with a phase-field approach

Study parameters

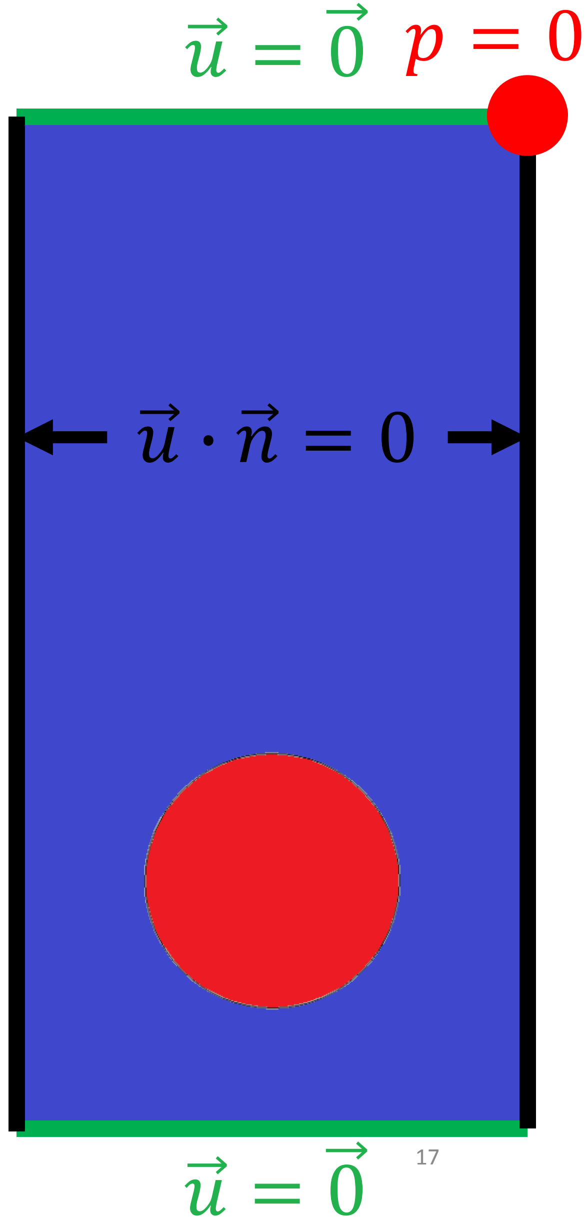
$\rho_1\ (kg.m^{-3})$	$\rho_2\ (kg.m^{-3})$	$\mu_1\ (Pa\cdot s)$	$\mu_2\ (Pa\cdot s)$	$g\ (m.s^{-2})$	$\sigma\ (N\cdot m^{-1})$
1000	1	10	0,1	0,98	1,96

II – 2D Study

Equations and boundary conditions

Laminar flow with Navier-Stokes equations

$$\begin{cases} \rho\left(\dfrac{\partial\vec{u}}{\partial t}+(\vec{u}\cdot\vec{\nabla})\vec{u}\right)=\rho\vec{g}-\vec{\nabla}p+\mu\Delta\vec{u} \\ div(\vec{u})=0 \end{cases}$$



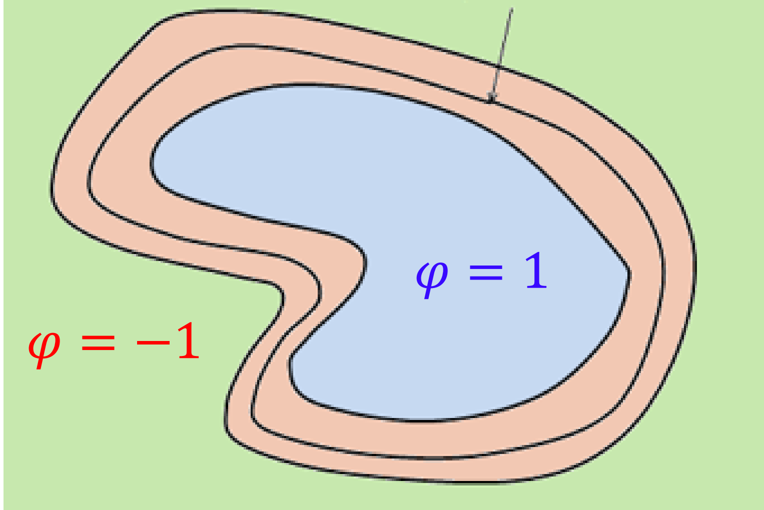
II – 2D Study

Phase-field method to simulate two-phase flow

Principle : Use a dimensionless phase field variable φ that can take values in {-1, 1} according to the phase represented

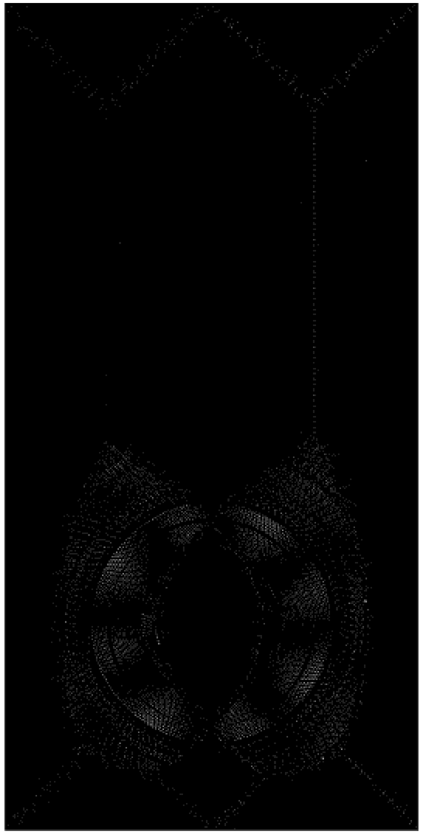
- Fluid 1 : $\varphi = -1$
- Fluid 2 : $\varphi = 1$

The physical interface is characterised by $\varphi = 0$

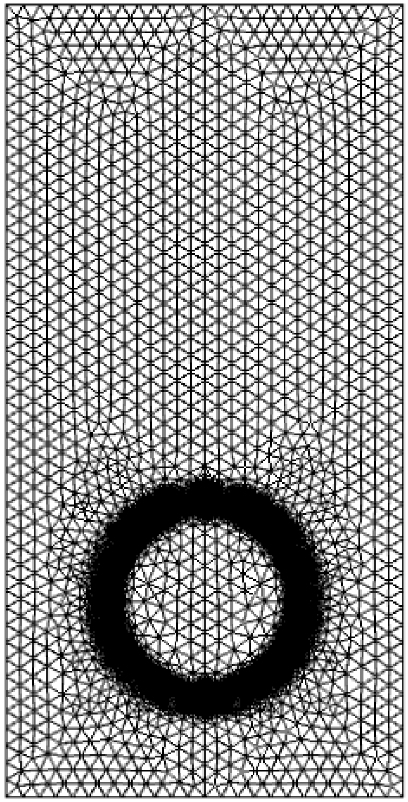


II – 2D Study

Two test cases: fixed mesh and adaptive mesh



Fine mesh



Adaptive mesh

II – 2D Study

Mesh type	Mesh element size	Number of degrees of freedom	Computational time
Fixed	$6{,}4 * 10^{-3}\ m$	260 000	75 minutes
Adaptive	$5{,}4 * 10^{-3}\ m$	250 000	15 minutes

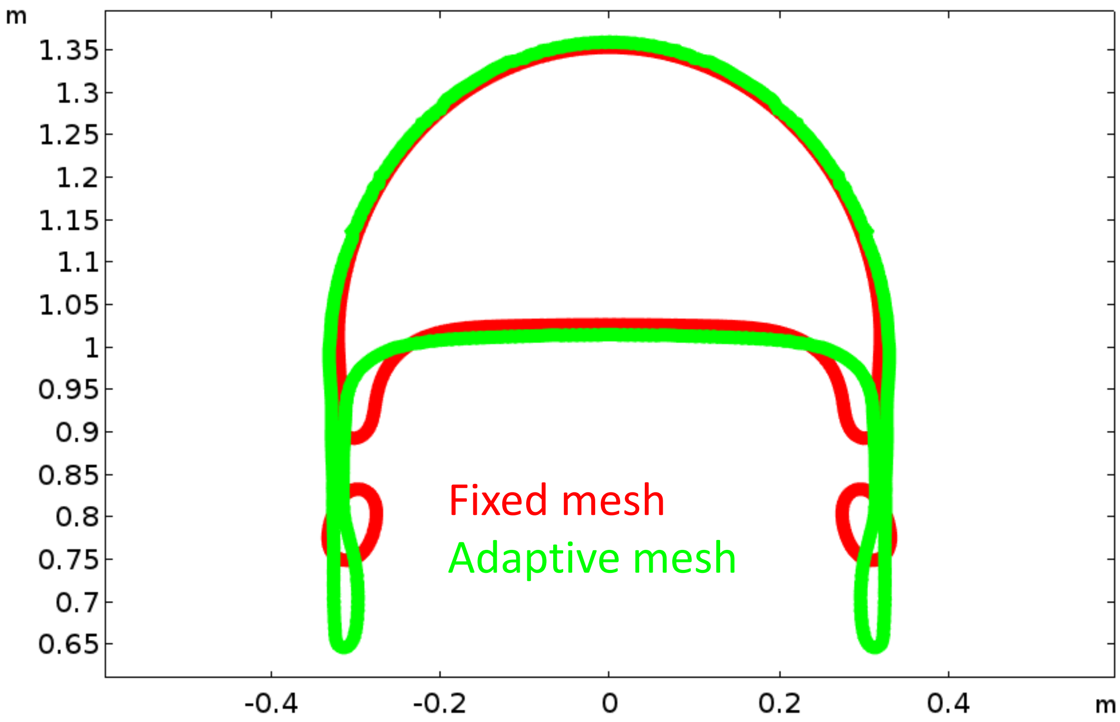
/5 !

→ Massive acceleration !

What about accuracy?

II – 2D Study

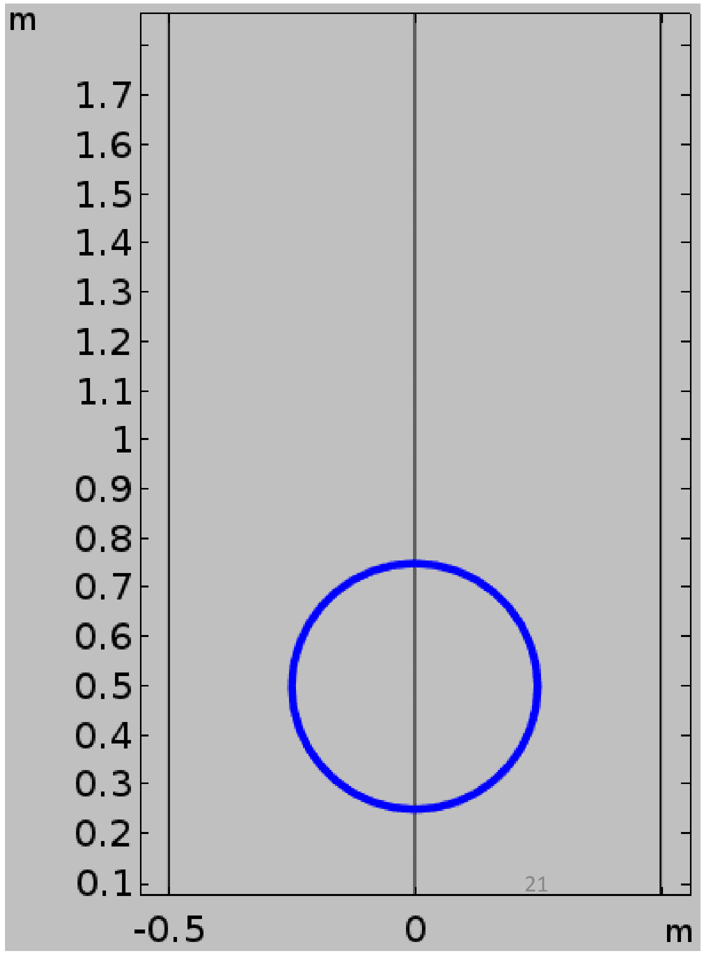
Results : bubble shape and comparison



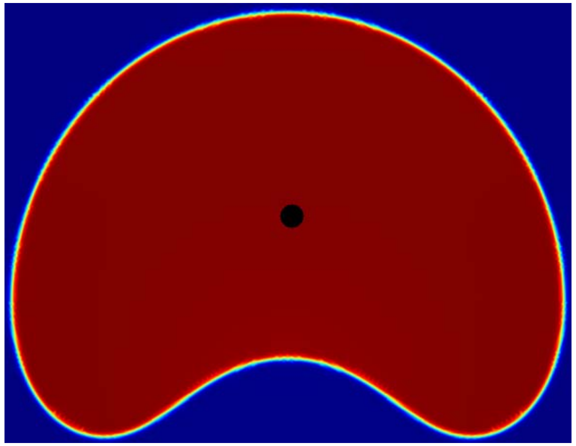
Bubble shape at t = 3s

→ Good adequacy generally …

… but some details vary (satellites)



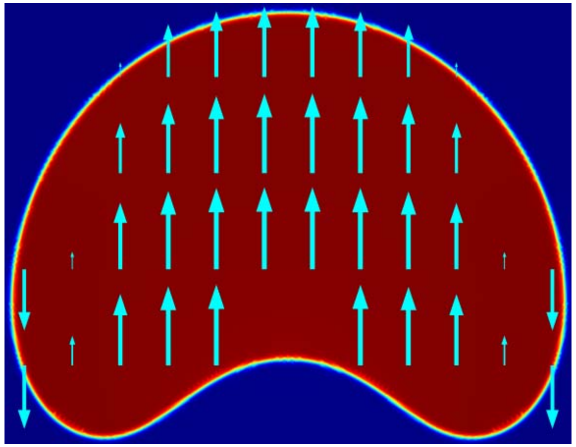
II – 2D Study



Quantitative comparison criteria

1. Position of centre of mass of the bubble $\bar{y} = \frac{1}{|\Omega|}\int_{\Omega} y\, dS$

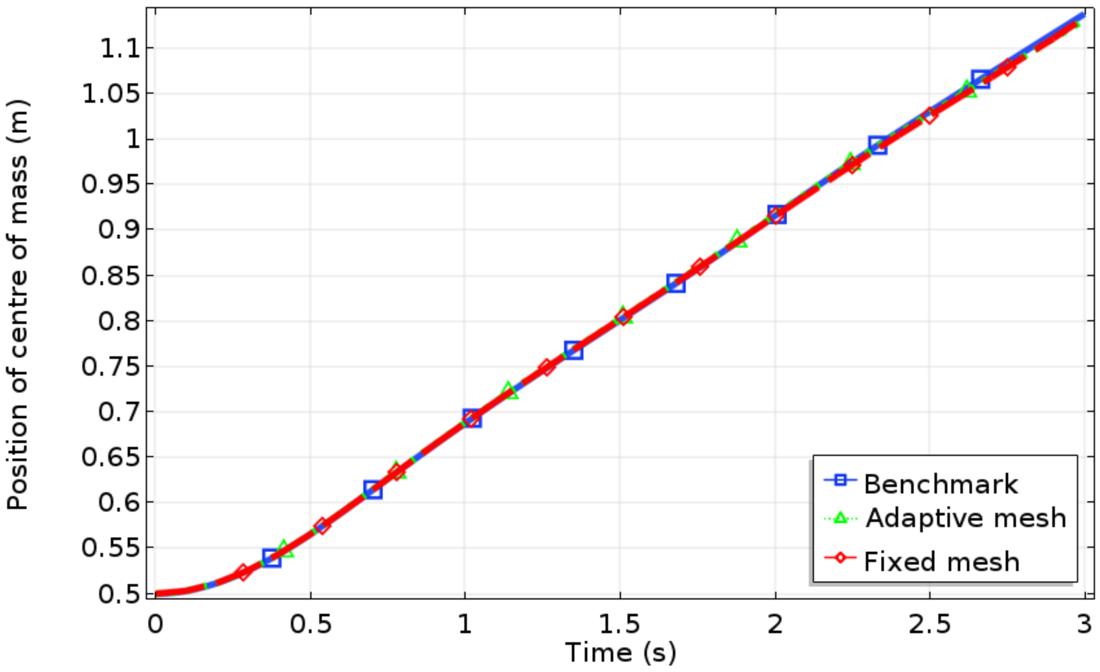
$$where\ \Omega = \{X \in \mathbb{R}^2 \mid \varphi(X) \geq 0\} \quad and \quad |\Omega| = \int_{\Omega} dS$$

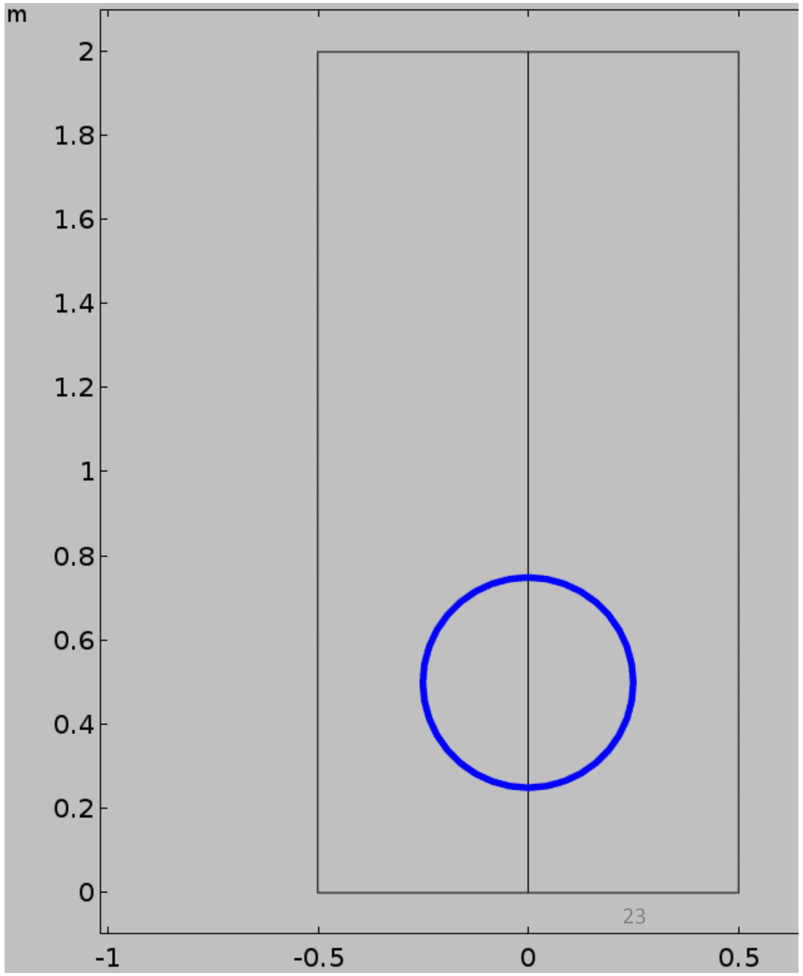


2. Mean rise velocity $\bar{v} = \frac{1}{|\Omega|}\int_{\Omega} v\, dS$

II – 2D Study

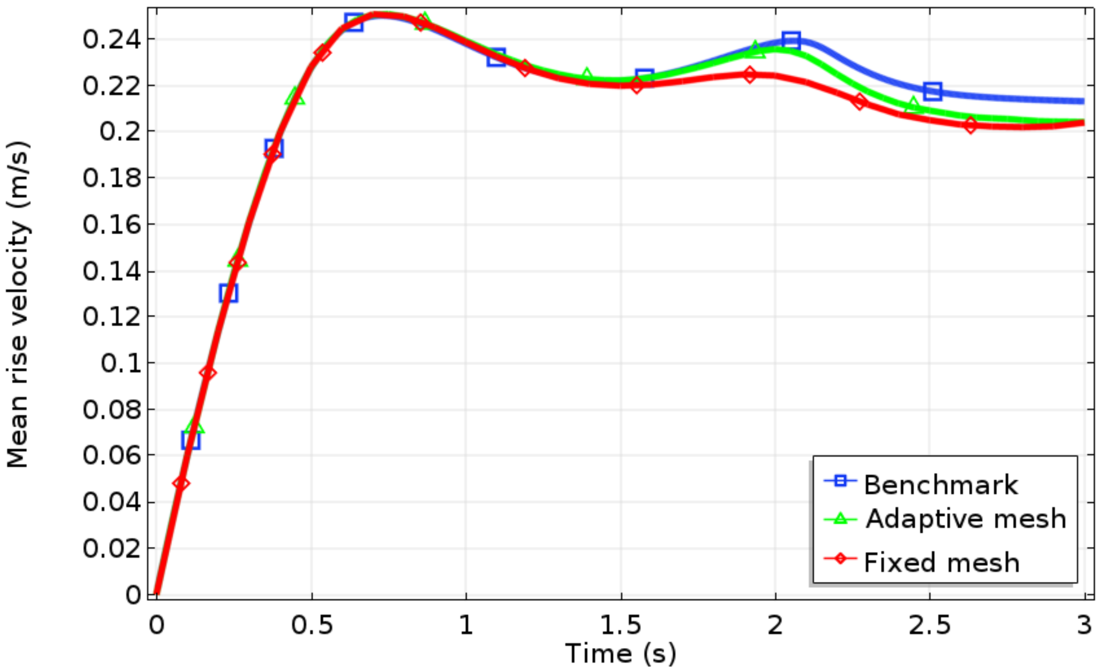
Results: comparison with the benchmark

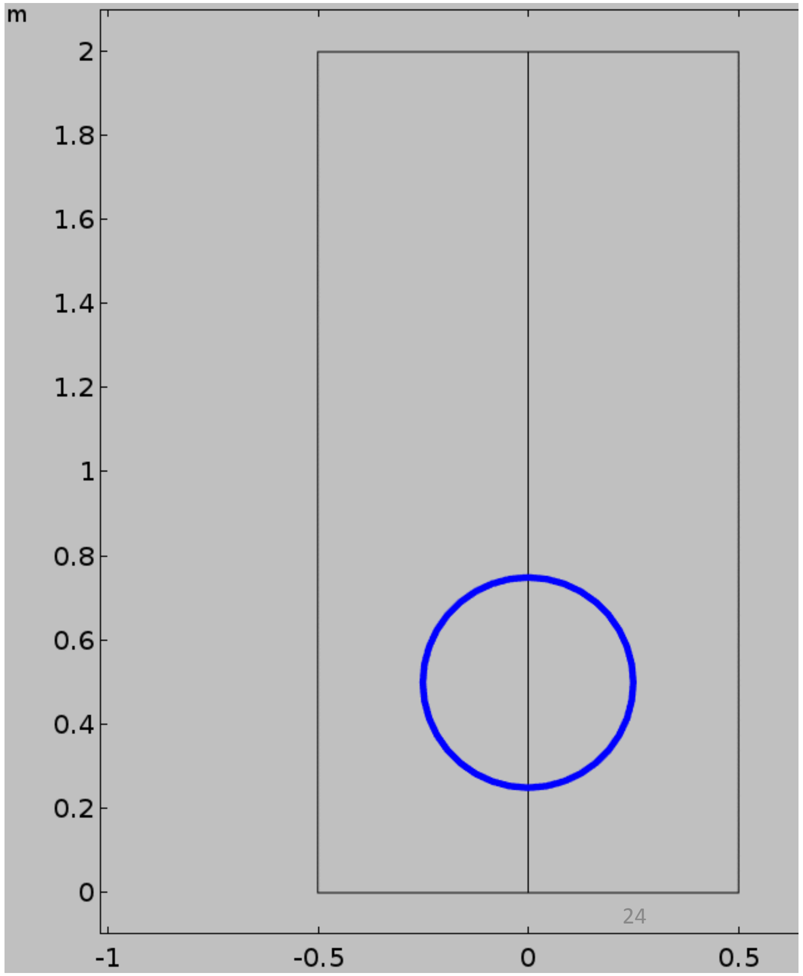




II – 2D Study

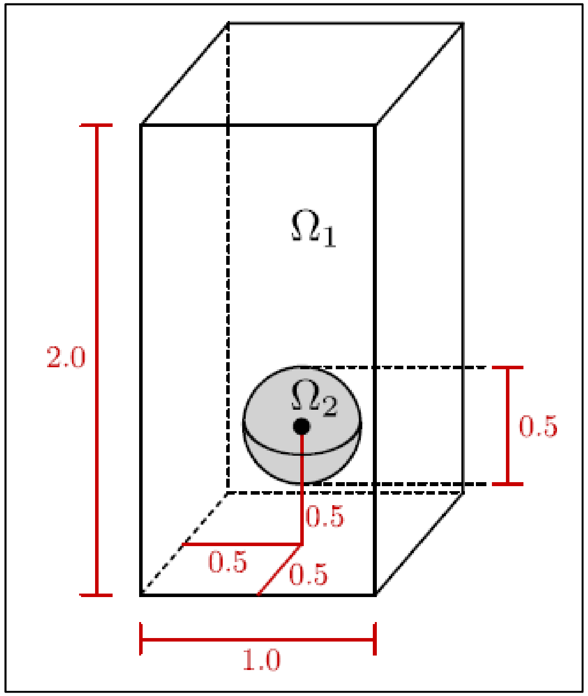
Results: comparison with the benchmark





III – 3D study

Scale in meters



Extract : reference article

Configuration

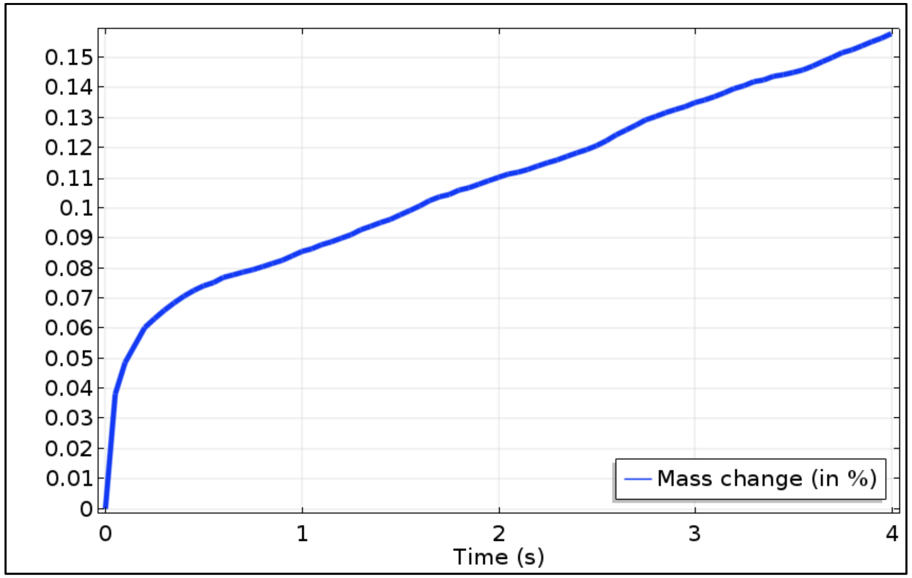
3D generalisation of the 2D case

From the reference paper

J. Adelsberger, P. Esser, M. Griebel, S. Groß, M. Klitz, and A. Rüttgers.
3D incompressible two-phase flow benchmark computations for rising droplets.
2014. Proceedings of the 11th World Congress on Computational Mechanics (WCCM XI), Barcelona, Spain, also available as INS Preprint No. 1401 and as IGPM Preprint No. 393.

III – 3D study

Numerical validation



Total mass variation < 0,2%

III – 3D study

Comparison with two CFD software

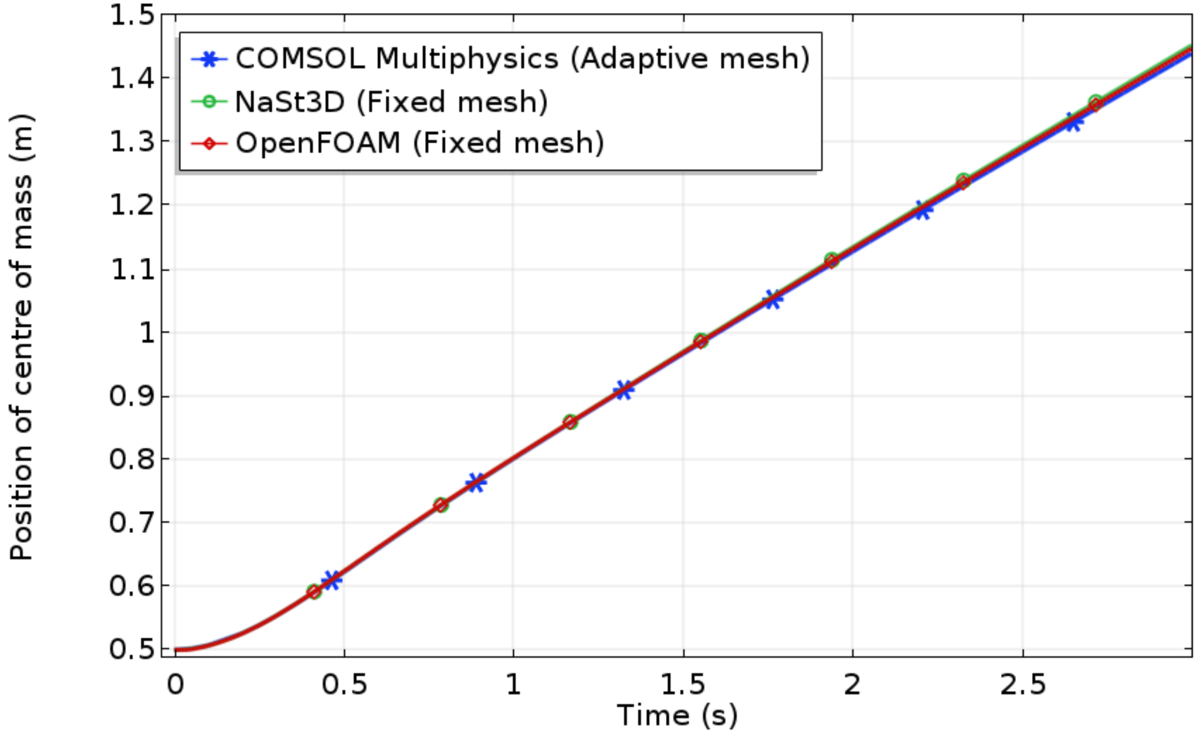
- NaSt3D
- OpenFOAM

Computational times

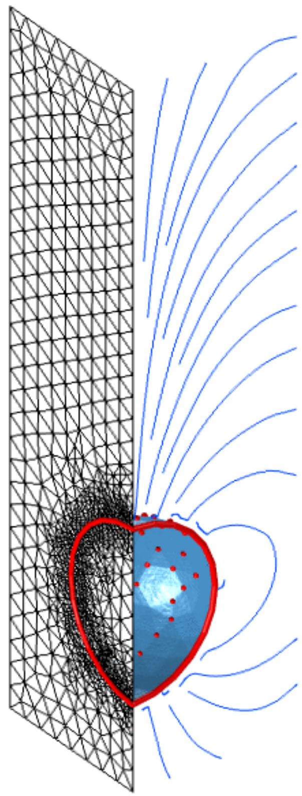
COMSOL Multiphysics® (adaptive mesh)	22 h on 2 cores at 4,16 GHz
NaSt3D (maillage fixe)	1 week on32 cores at 2,226 GHz
OpenFOAM (maillage fixe)	2,5 days on 32 cores at 2,226 GHz

III – 3D study

<u>Results: comparison with the benchmark</u>

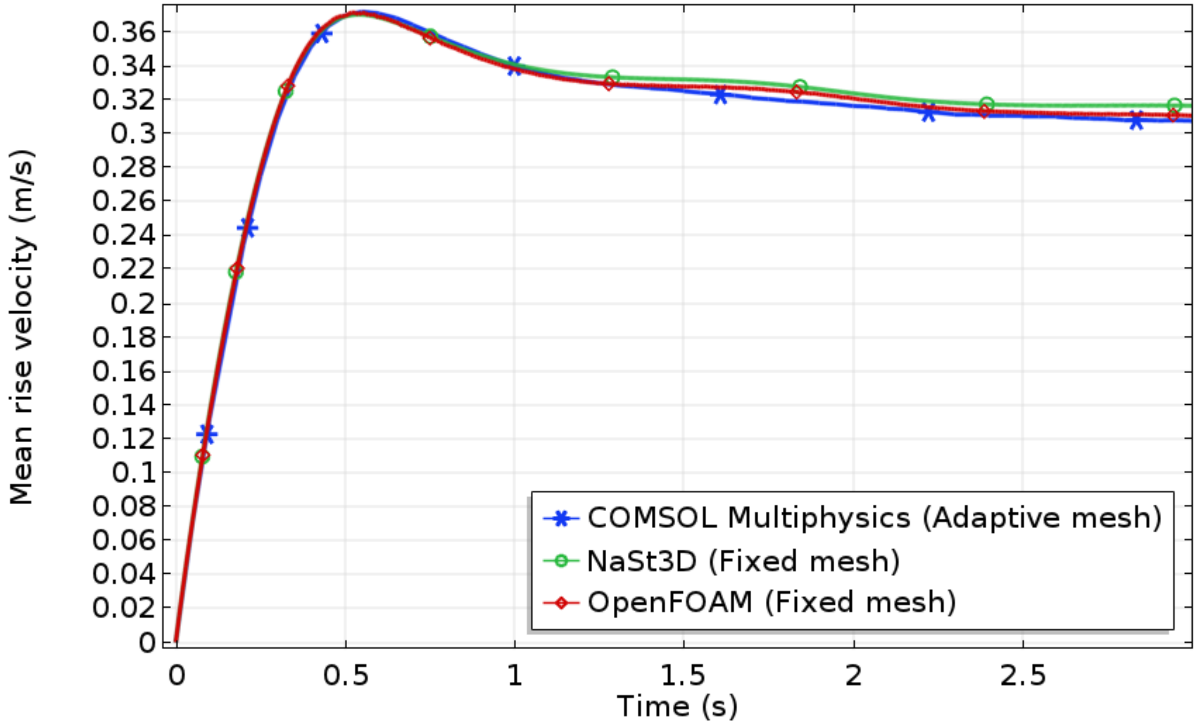


Mesh visualisation (left)
streamlines (right)
interface (in red)

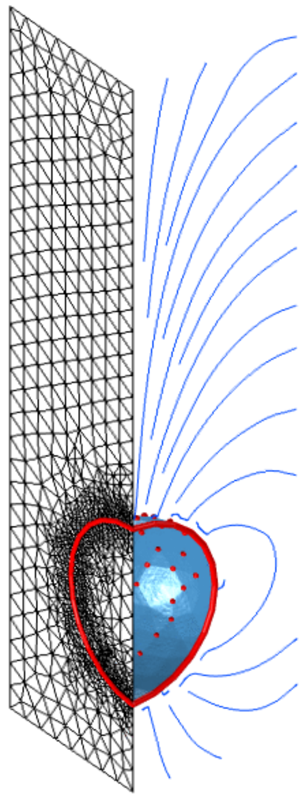


III – 3D study

Results: comparison with the benchmark



Mesh visualisation (left)
streamlines (right)
interface (in red)

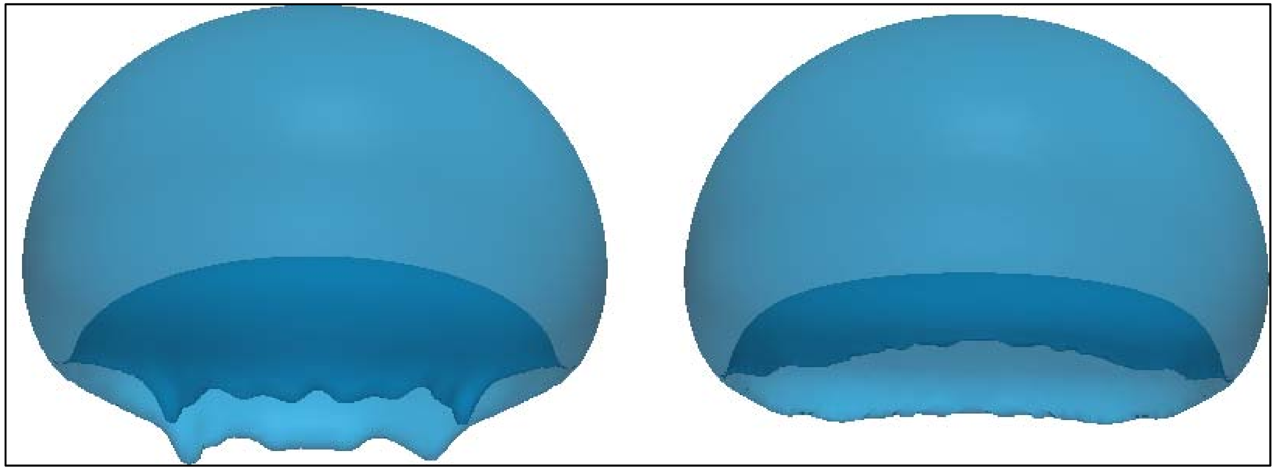


III – 3D study

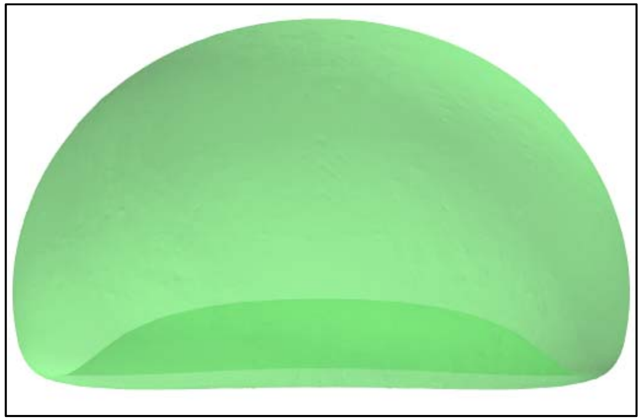
Results: comparison of the bubble shape at t=3,5s

NaSt3D | OpenFOAM

Extract: reference article

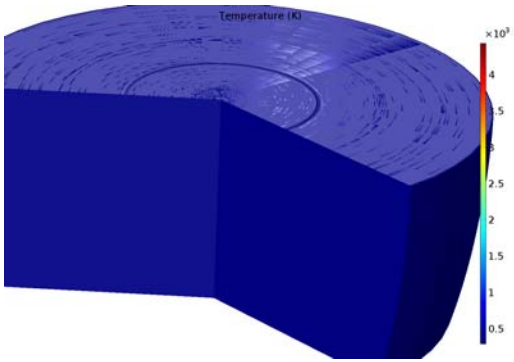


Adaptive mesh with COMSOL Multiphysics®

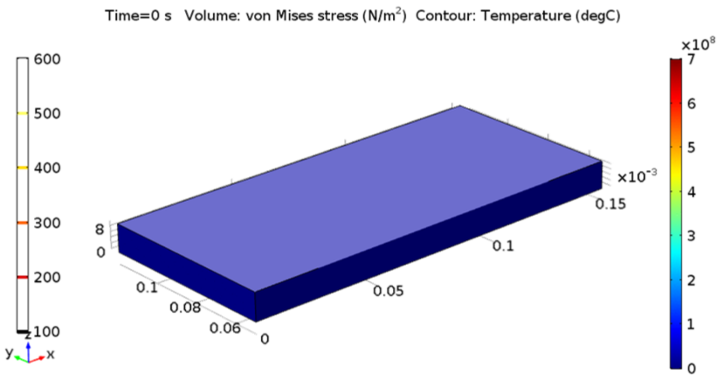


Conclusion

- Principle of the adaptive mesh refinement method : accelerate calculations while improving accuracy
- Comparison with results from literature and others CFD software : **validation of the method**
- Practical applications on industry topics:



Laser piercing



Additive fabrication

Thanks a lot!
Q&A

T.Preney, J.D. Wheeler and P.Namy
SIMTEC- (+33) 9 53 51 45 60
patrick.namy@simtecsolution.fr

COMSOL CONFERENCE 2018 BOSTON