Mesh-free Lagrangian modelling of fluid dynamics

David LE TOUZÉ, Ecole Centrale Nantes

CFD 2011, keynote lecture

Mesh-free Lagrangian methods in CFD

Smoothed-Particle Hydrodynamics (SPH)

Fast-dynamics free-surface flows

Multi-fluid flows

Fluid-Structure Interactions

HPC: massive interactive simulations

Mesh-free Lagrangian methods

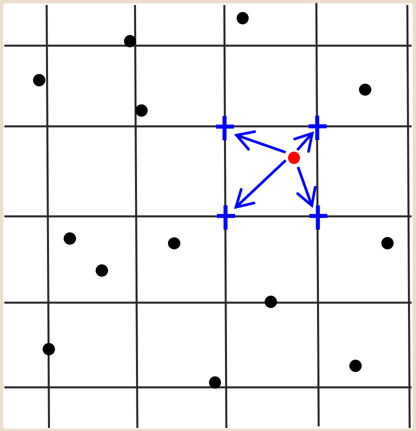
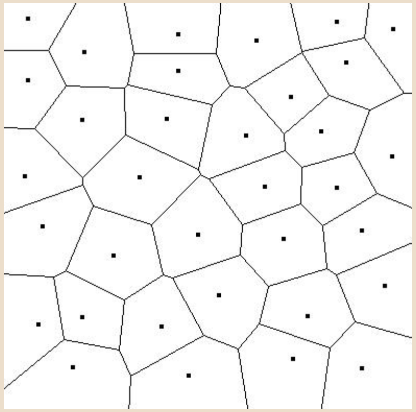
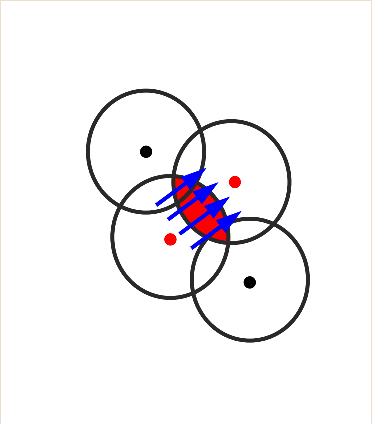
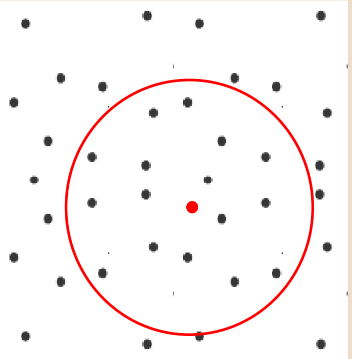
Methods for discrete media	**Methods for continuous media**
A « particle » is an atom, a molecule, a grain, etc.	A « particle » is an elementary mass of considered continuous medium
Methods directly model laws of interaction between two elements	Methods model local PDEs describing the medium mechanics/physics
MD, DEM, DSMC…	SPH, MPS, FPM, RKPM…

Numerically the discretized local equations can take the form of a Lagrangian set of material points (« particles »), implying intrinsic properties: conservation of linear and angular momenta, etc.

=> 2 complementary *numerical* visions:
standard discretized PDEs / particle system

Mesh-free Lagrangian methods for fluid dynamics

Mesh-free?

Projection	Tesselation	Face reconstruction	True mesh-free
			
PM…	PFEM	FVPM	SPH, MPS…

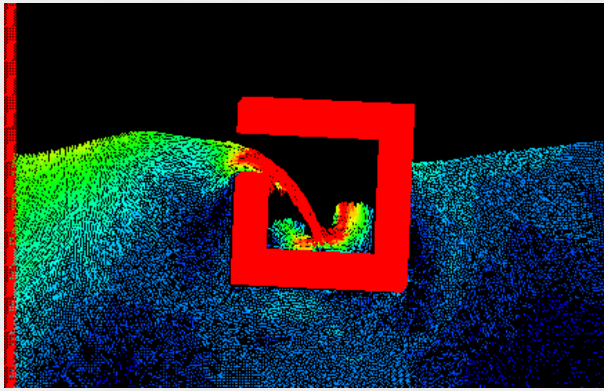
Mesh-free Lagrangian methods for fluid dynamics

Mesh-free methods, for what purpose?

- to solve situations where meshes have difficulties: very **large deformations** of the fluid domain / **large motions** of multi-bodies (especially contact between 2 bodies in the flow)
- to avoid costly mesh generating/handling (human cost especially)

Lagrangian methods in fluid, for what purpose?

- to naturally and accurately follow **interfaces** deformations:
  - Complex free surfaces
  - Multi-fluid/-phase interfaces
  - Fluid-structure interfaces
- to solve **fast dynamics** flows: no convection term



Lagrangian meshless methods, NOT for what?

- for all problems where mesh-based methods are already reliable (e.g. high-Re turbulent flows, steady flow, etc.)

Mesh-free Lagrangian modelling of fluid dynamics

David LE TOUZÉ, Ecole Centrale Nantes

CFD 2011, keynote lecture

Mesh-free Lagrangian methods in CFD

Smoothed-Particle Hydrodynamics (SPH)

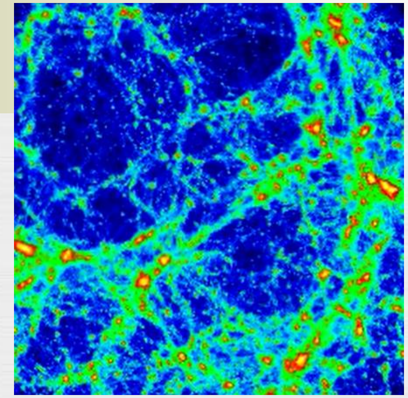
Fast-dynamics free-surface flows

Multi-fluid flows

Fluid-Structure Interactions

HPC: massive interactive simulations

SPH method: origin



1977 : Gingold & Monaghan / Lucy: SPH

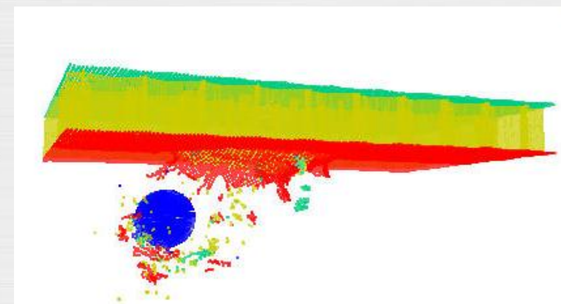
application field: astrophysics

methodological background: statistics

80's : mainly applied in astrophysics

star formation, self-gravitating clouds, galactic shocks…

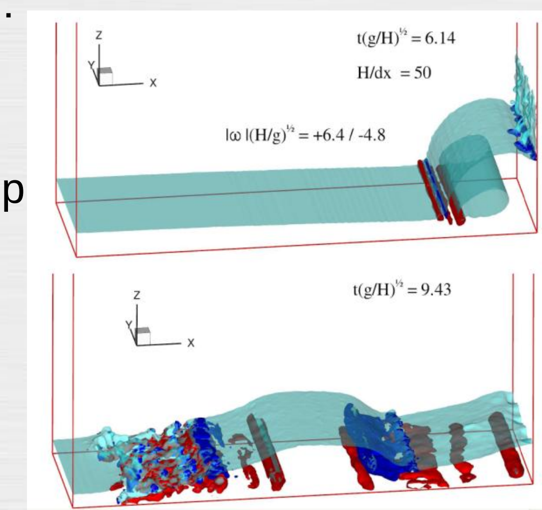
then for modeling structures



90's : astrophysics, structure;

1994: free-surface SPH (Monaghan)

2000's : astrophysics, structure, fluid dynamics, polymers, surfactants, explosions, lava, porous media, geotechnical problems, biomechanics, metallurgy…

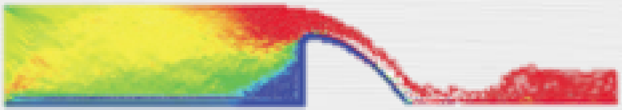


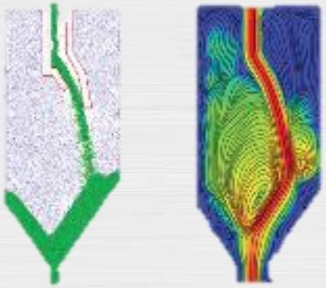
SPH in fluid dynamics

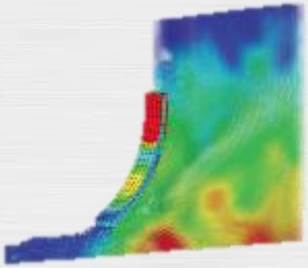
- fast development: **SPHERIC** ERCOFTAC Special Interest Group
  created in 2005, today: 91 international member entities
- a similar method (MPS) is also in fast development in Japan

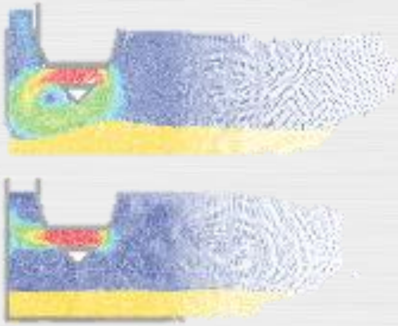
SPH in the world

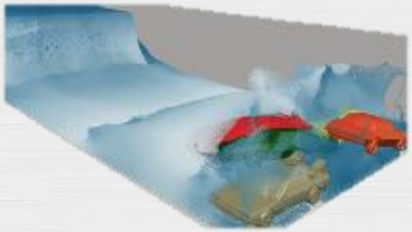
- SPHERIC group : 95 entities of 29 countries worldwide
  Annual workshop, benchmarks, bibliography and job databases, codes…

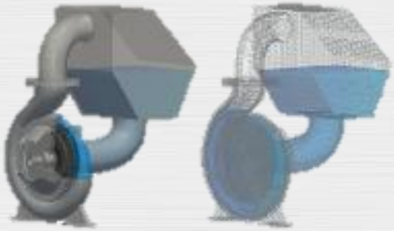


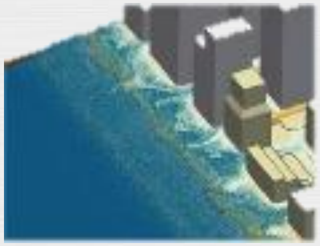


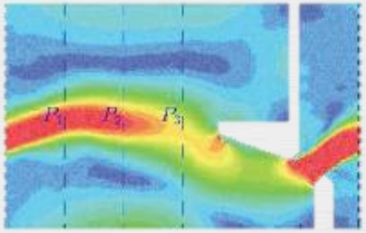












SPH method: origin

Smoothed Particle Hydrodynamics

Regularizing function

Mesh-free Lagrangian method

Liquids in motion

A fluid dynamics problem numerically seen as a system of masses (particles) using a regularizing function.

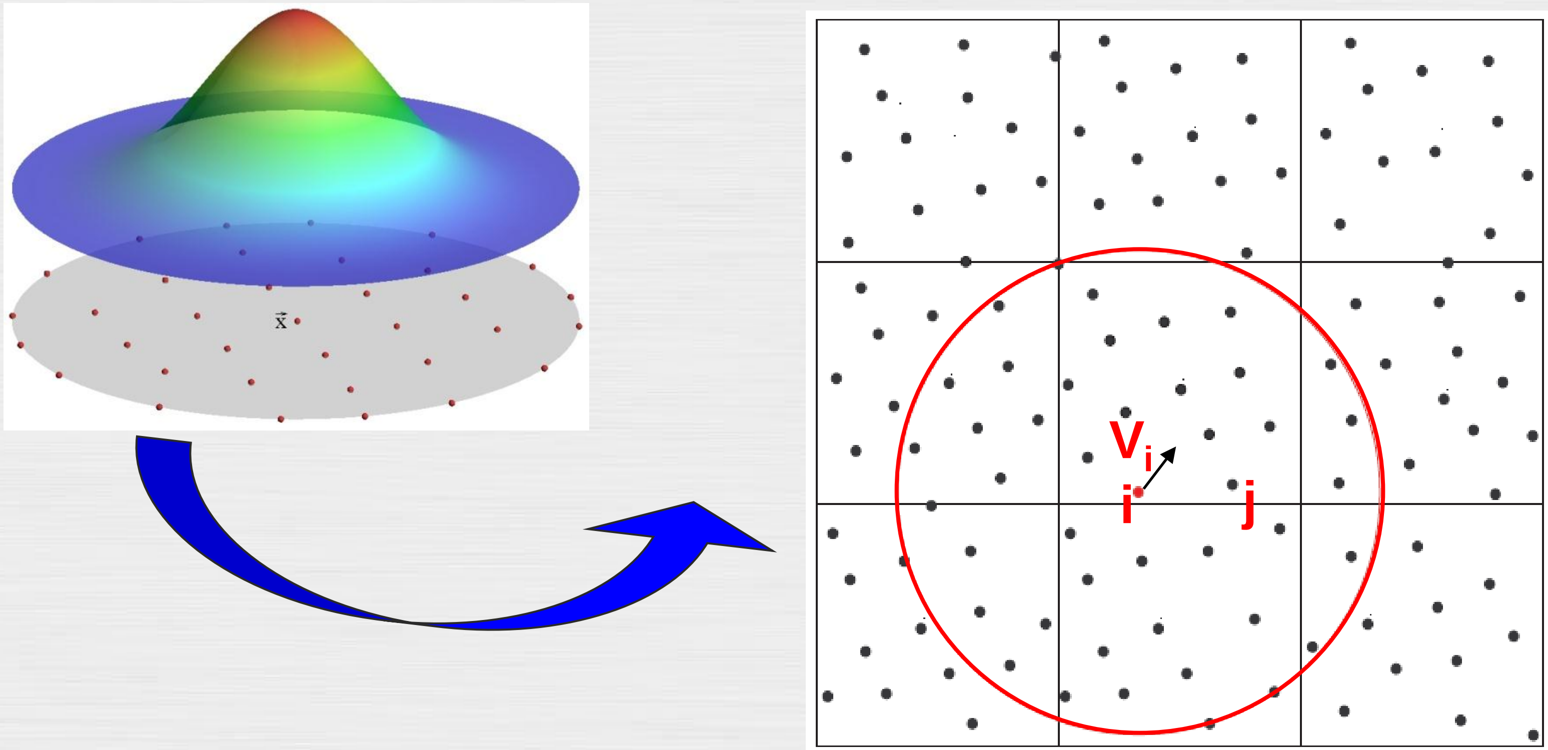
SPH method: characteristics

- meshless: no need for handling a mesh
- explicit: « easy » and efficient parallelization
- Lagrangian: no convective term modeling:
  => accurate for fast dynamics flows (violent impacts, shocks, explosions…)
- naturally handles large deformations of the domain:
  - large motion of (multi-)bodies
  - arbitrary complex free surface
- perfectly non-diffusive interface
- permits to include different species (materials, phases…) in a monolithic way

BUT

- enhanced versions are needed to get accurate pressure fields
- implementation of boundary conditions must be considered with great care
- convergence of SPH as usually used is not theoretically proved, heuristically: order 1

SPH method: principle



- Lagrangian formalism
- equations discretized within a compact zone of influence
- differential operators obtained from quantity values at the points included in this zone of influence

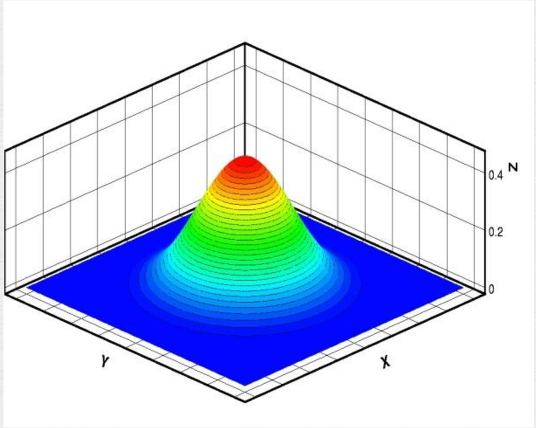
SPH method: discretization

interpolation

convolution using a kernel W (~test fonction)

$$f(\vec{r}) \approx \int_D f(\vec{x}) W(\vec{r}-\vec{x}) d\vec{x}$$

$$W\left(q=\frac{|\vec{r}|}{h}\right) = C \begin{cases} \frac{2}{3} - q^2 + \frac{1}{2}q^3 & \text{if } 0 \leq q < 1 \\ \frac{1}{6}(2-q)^3 & \text{if } 1 \leq q < 2 \\ 0 & \text{else} \end{cases}$$



interest

estimation of the gradient from the values of the considered quantity

$$\nabla f(\vec{r}) \approx \int_D \nabla f(\vec{x}) W(\vec{r}-\vec{x}) d\vec{x} = \int_D f(\vec{x}) \nabla W(\vec{r}-\vec{x}) d\vec{x}$$

analytical

SPH discretization: weakly-compressible variant

Numerical scheme - inviscid here

fully explicit

- Euler equation $$\frac{D\vec{v}}{Dt} = \vec{g} - \frac{\nabla p}{\rho}$$
- Free-surface condition $$p(\vec{x},t) = 0 \text{ et } \frac{D\vec{x}}{Dt} = \vec{v}_{\partial\Omega} \quad \forall \vec{x} \in \partial\Omega$$

=> intrinsic
Colagrossi, Antuono, Le Touzé, *Phys. Rev. E*, 2009

- Initial conditions $$\vec{v}(\vec{x},t_0) = \vec{v}_0 \text{ et } p(\vec{x},t_0) = p_0$$
- Compressible fluid $$\frac{D\rho}{Dt} = -\rho\, div(\vec{v})$$

$$P - P_0 = \frac{\rho_0 c_0^2}{7}\left[\left(\frac{\rho}{\rho_0}\right)^7 - 1\right]$$

$$\langle \nabla p_i \rangle = \sum_j (p_i + p_j)\,\omega_j \nabla W(\vec{x}_i - \vec{x}_j)$$

$$\langle div(\vec{v}_i) \rangle = \sum_j (\vec{v}_j - \vec{v}_i)\,\omega_j \nabla W(\vec{x}_i - \vec{x}_j)$$

for compatibility (energy conservation)

chosen at the weak-cpr. limit (Ma=0.1) when simulating incompressible flows = pseudo-compressibility

SPH discretization: weakly-compressible variant

Stabilization

Method 1

last equations + artificial viscosity added to the pressure term

Method 2

use of a density diffusive term (proportional to a Rusanov flux) in the continuity equation

$$\frac{D\rho_i}{Dt} = -\rho_i \sum_j \left(\boldsymbol{u}_j - \boldsymbol{u}_i\right)\cdot\nabla_i W\left(\boldsymbol{r}_j\right)V_j + \delta h c_0 \sum_j \boldsymbol{\psi}_{ij}\cdot\nabla_i W\left(\boldsymbol{r}_j\right)V_j$$

Method 3

use of an ALE form (Vila, 1999) and of Riemann solvers between each pair of particles (standard in FVM for compressible flows)

$$\begin{cases} \left.\dfrac{d\overline{x_i}}{dt}\right|_{v_0} = \overline{v_{0i}} \\ \left.\dfrac{d\omega_i}{dt}\right|_{v^0} = \displaystyle\sum_{j\in P(\Omega)} \omega_i\omega_j\left(\overline{v_{0j}} - \overline{v_{0i}}\right).\nabla_i W_{ij} \\ \left.\dfrac{d\omega_i\overline{\phi_i}}{dt}\right|_{v^0} + \displaystyle\sum_{j\in P(\Omega)} \omega_i\omega_j \underbrace{\left(\overline{\overline{F_i}} - \overline{\phi_i}\otimes\overline{v_{0i}} + \overline{\overline{F_j}} - \overline{\phi_j}\otimes\overline{v_{0j}}\right)}_{\text{given by the Riemann problem solution}}.\nabla_i W_{ij} = \omega_i\overline{S_i} \end{cases}$$

SPH discretization: incompressible variant

Numerical scheme - viscous here

semi-implicit

$$\frac{Du}{Dt} = -\frac{1}{\rho}\nabla p + \upsilon\nabla^2 u + F$$

$$\nabla \cdot u = 0$$

$$u_i^* = u_i^n + \ \upsilon\nabla^2 u_i^n + F_i^n \ \ \Delta t$$

$$r_i^* = r_i^n + \Delta t u_i^*$$

$$\rho_i^* = \sum_j m_j \omega_{ij}$$

Pressure Poisson Equation

$$\nabla \cdot \left(\frac{1}{\rho}\nabla p^{n+1}\right)_i = \frac{1}{\Delta t}\nabla \cdot u_i^* \quad \text{or} \quad \nabla \cdot \left(\frac{1}{\rho}\nabla p^{n+1}\right)_i = \frac{\rho_0 - \rho_i^*}{\rho_0 \Delta t^2}$$

ISPH_DF ISPH_DI

$$u_i^{n+1} = u_i^* - \frac{\Delta t}{\rho}\nabla p_i^{n+1}$$

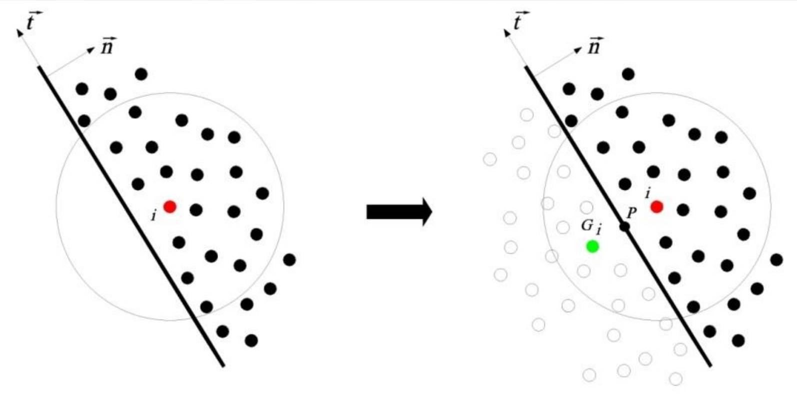
$$r_i^{n+1} = r_i^n + \Delta t\left(\frac{u_i^{n+1} + u_i^n}{2}\right)$$

- Solved by standard incompressible solvers (pre-conditioned Bi-CGStab…)
- Need for correctly imposing free-surface conditions

Solid boundaries handling: ghost particles

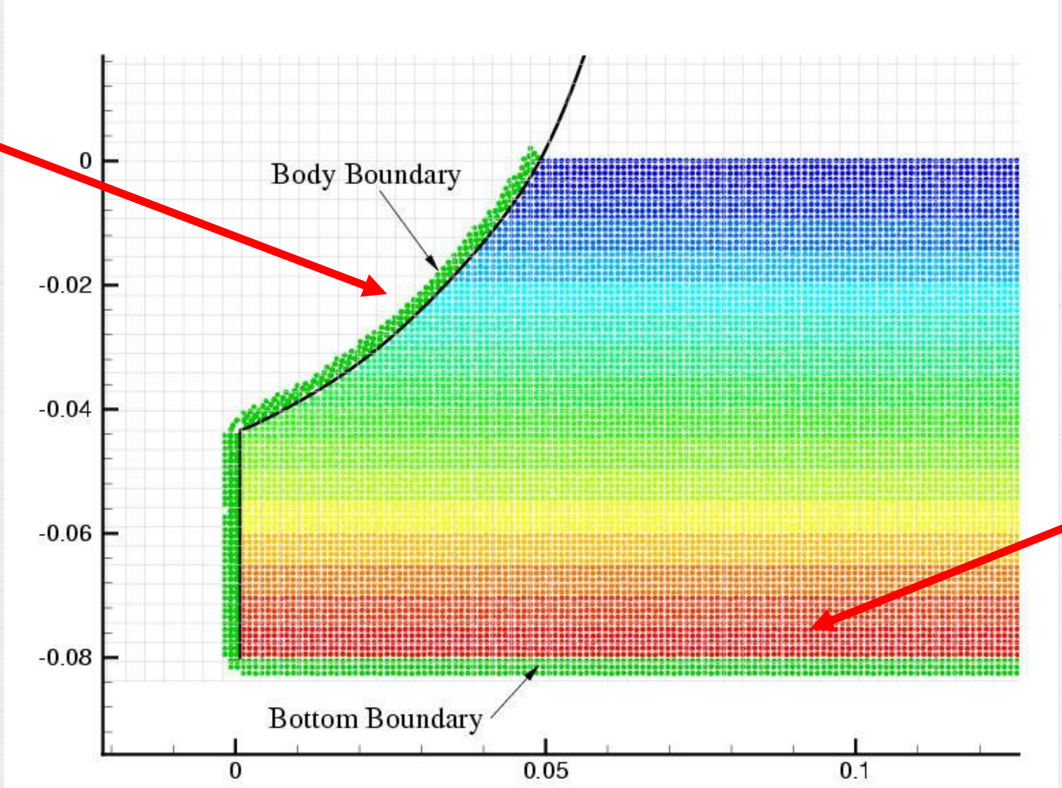
principle

solid BCs are imposed thanks to mirrored particles



$$V_{G_{in}} = 2V_{pn} - V_{in}$$
$$V_{G_{it}} = V_{pt}$$

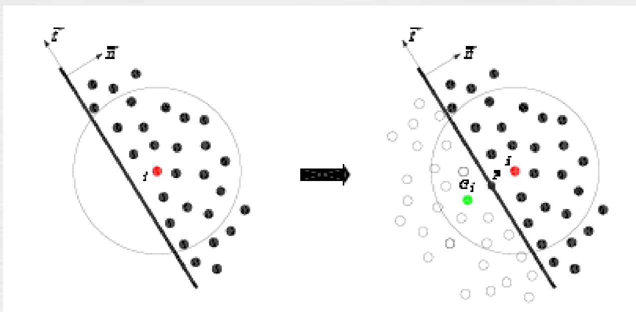
‘ghost’ particles

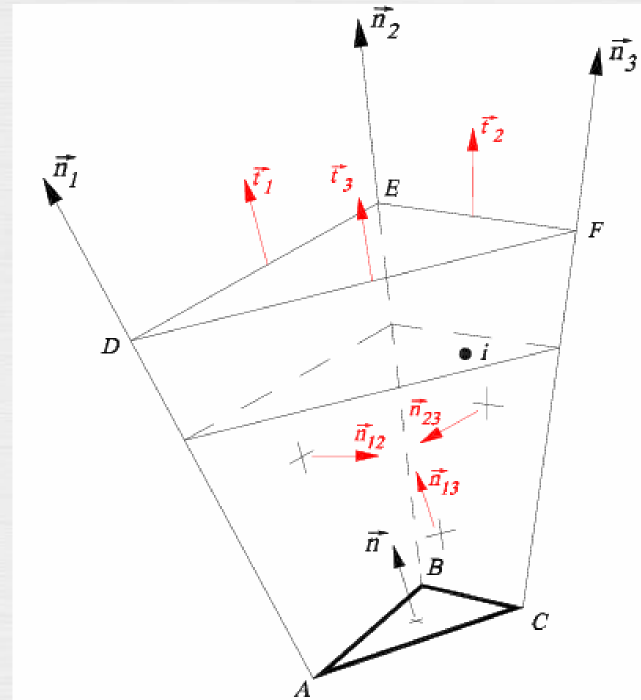


fluid particles

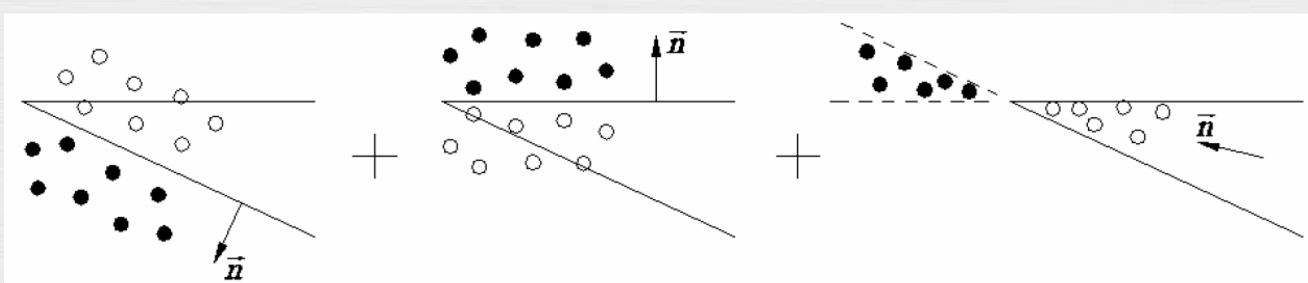
Solid boundaries handling: ghost particles

- 3D generic technique
  - from any surface (e.g. IGES format)





- Exact only for a flat panel, difficulties with geometrical singularities, especially sharp edges

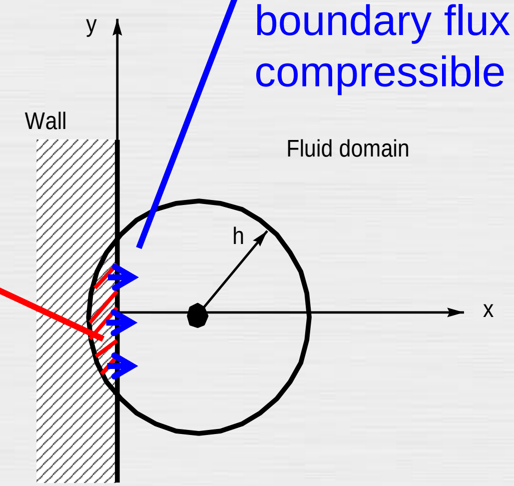


Solid boundaries handling: normal flux method

$$
\begin{cases}
\left.\dfrac{d\overline{x_i}}{dt}\right|_{v_0} = \overline{v_{0i}} \\[2ex]
\left.\dfrac{d\omega_i}{dt}\right|_{v^0} = \dfrac{1}{\gamma_i}\displaystyle\sum_{j\in P(\Omega)} \omega_i\omega_j\left(\overline{v_{0j}}-\overline{v_{0i}}\right).\nabla_i W_{ij} + \dfrac{1}{\gamma_i}\displaystyle\sum_{j\in P(\partial\Omega)} \omega_i s_j\left(\overline{v_{0j}}-\overline{v_{0i}}\right).\overline{n_j}W_{ij} \\[2ex]
\left.\dfrac{d\omega_i\overline{\phi_i}}{dt}\right|_{v^0} + \dfrac{1}{\gamma_i}\displaystyle\sum_{j\in P(\Omega)} \omega_i\omega_j\left(\overline{\overline{F_i}}+\overline{\overline{F_j}}\right).\nabla_i W_{ij} + \dfrac{1}{\gamma_i}\displaystyle\sum_{j\in P(\partial\Omega)} \omega_i s_j\left(\overline{\overline{F_i}}+\overline{\overline{F_j}}\right).\overline{n_j}W_{ij} = \omega_i\overline{S_i}
\end{cases}
$$

compensates missing part of the kernel support

boundary flux obtained from compressible flow characteristics

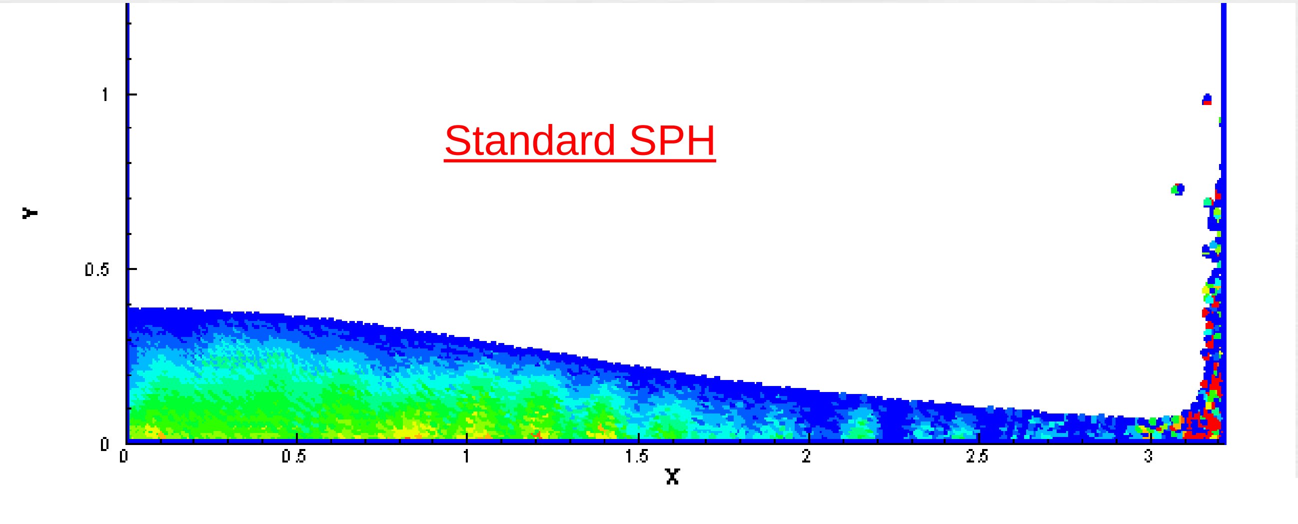


- **Fully general technique**

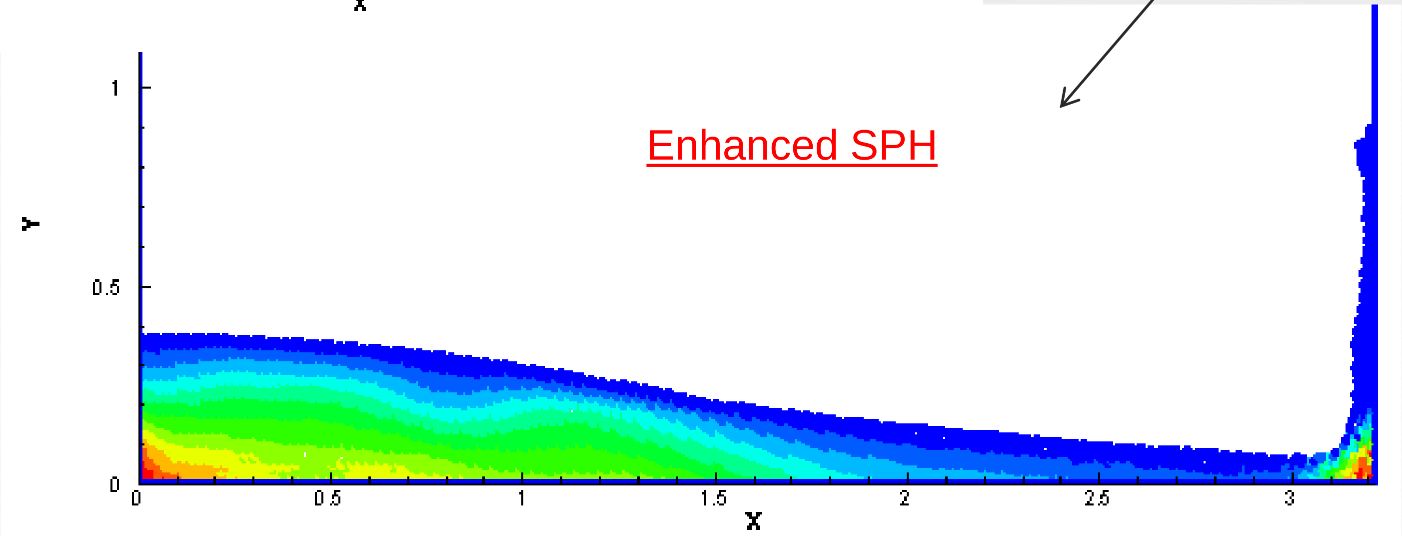
De Leffe, Le Touzé, Alessandrini, *submitted to J. Comput. Phys.*

SPH solver: accuracy

Pressure field quality

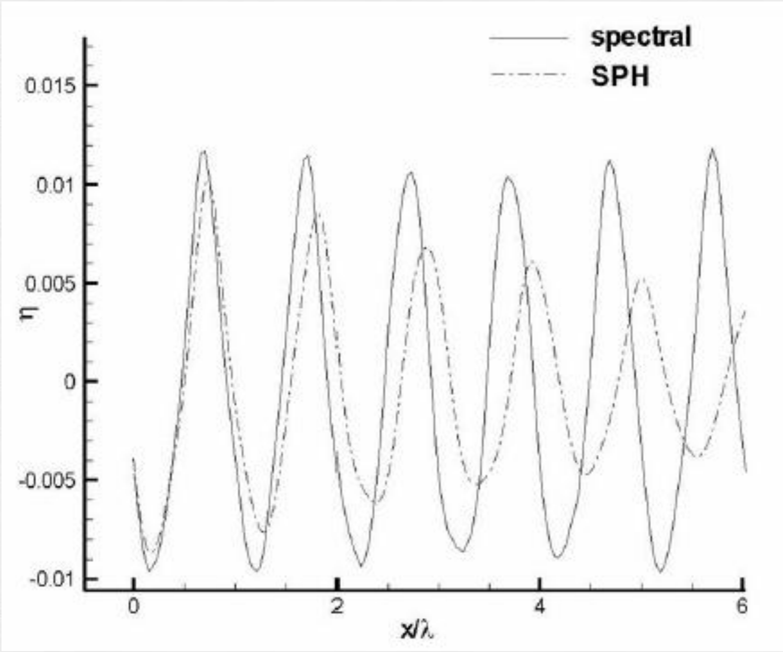


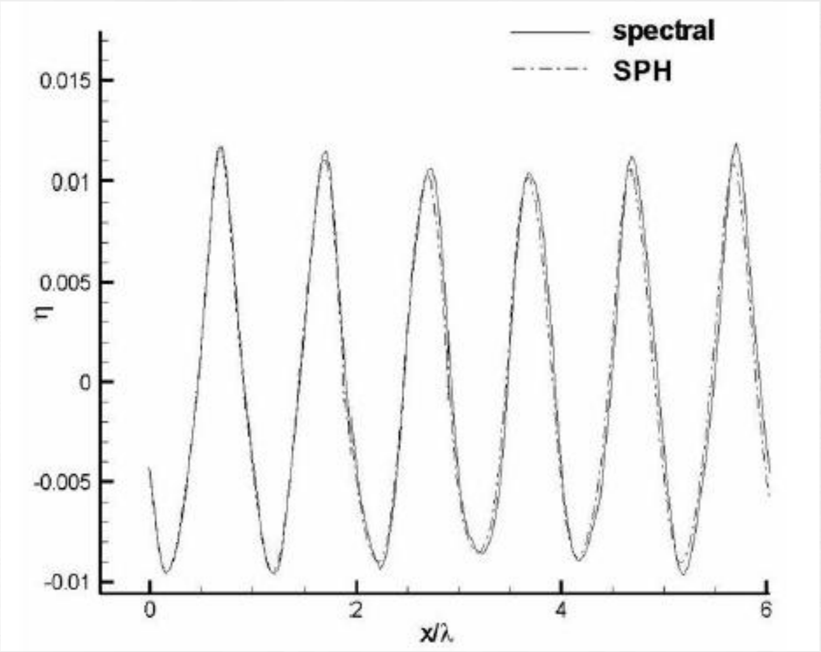
Use of density diffusive term, or Riemann solver ,or incompressible variant…



SPH solver: accuracy

Accuracy test: gravity wave propagation





Standard SPH

Enhanced SPH

Additional use of kernel renormalization to get order-1 completeness

The solver SPH-flow

SPH-flow

SPH-flow solver

Model core

- 3D
- specific tools to enhance model accuracy (Riemann solver, kernel renormalisation…)
- variable-size particle algorithm (equivalent to mesh refinment)
- generalized ghost technique for modeling any 3D solid boundary
- 6-dof bodies
- efficient parallel model
- inflow/outflow conditions implemented

Three key points

- accuracy
- HPC
- validation

Extensions of the model

- fluid-structure coupling
- multifluid flows
- Navier-Stokes: viscous flow + turbulence modeling
- wave-structure interaction ; shallow-water SPH model ; …

Mesh-free Lagrangian modelling of fluid dynamics

David LE TOUZÉ, Ecole Centrale Nantes

CFD 2011, keynote lecture

Mesh-free Lagrangian methods in CFD

Smoothed-Particle Hydrodynamics (SPH)

Fast-dynamics free-surface flows

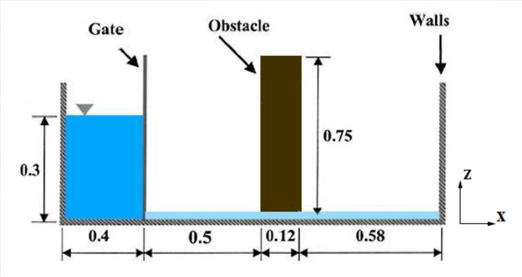
Multi-fluid flows

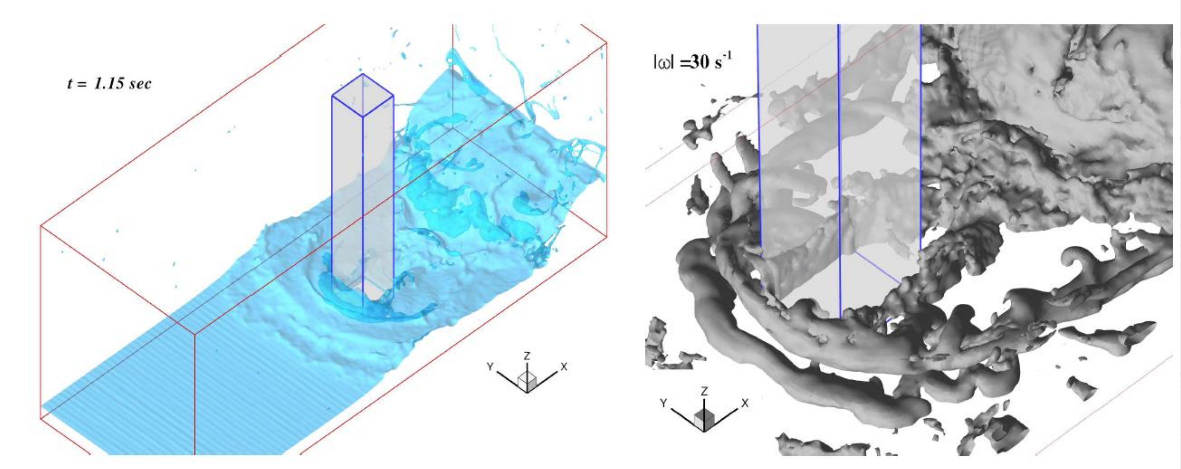
Fluid-Structure Interactions

HPC: massive interactive simulations

Fast dynamics free surface flows: validation

Validation on dam breaking test cases

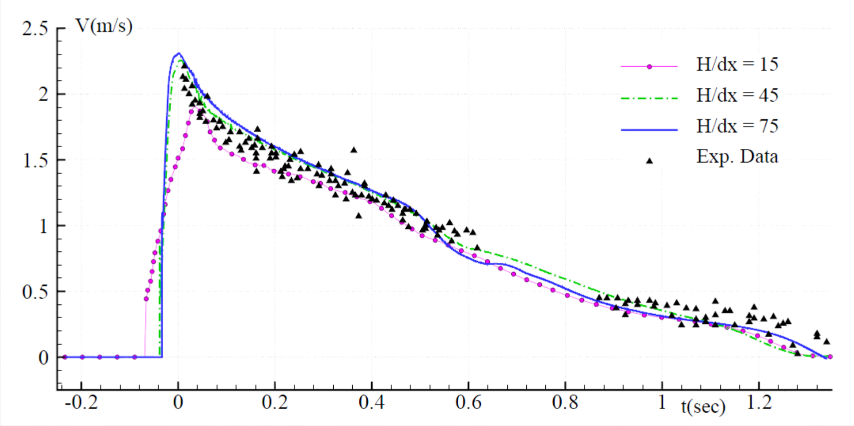


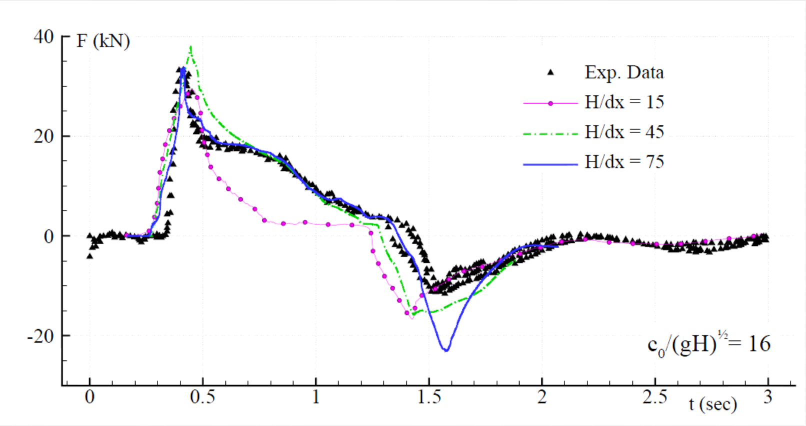


with 80,000 particles only =>

with 1 million particles =>

- Good prediction of velocity and forces :

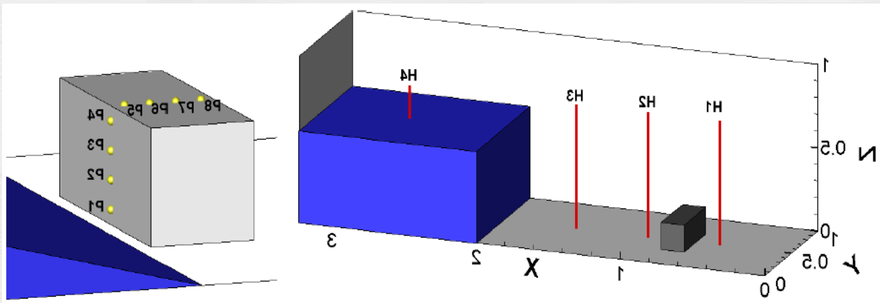


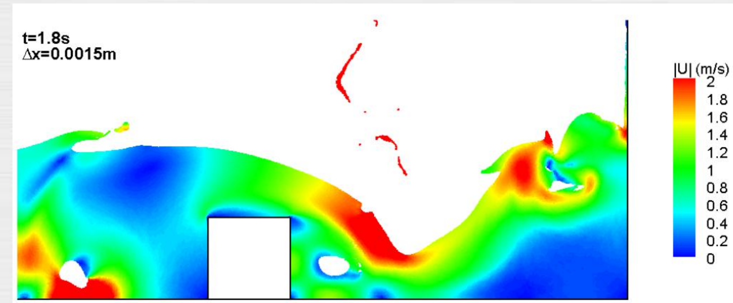


Marrone et al., *Comput. Meth. Appl. Mech. Eng. 200*, 2011

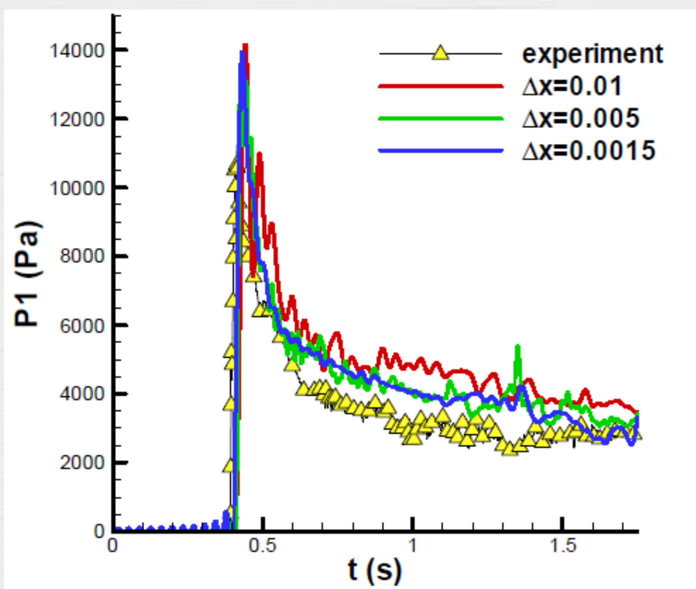
Fast dynamics free surface flows: validation

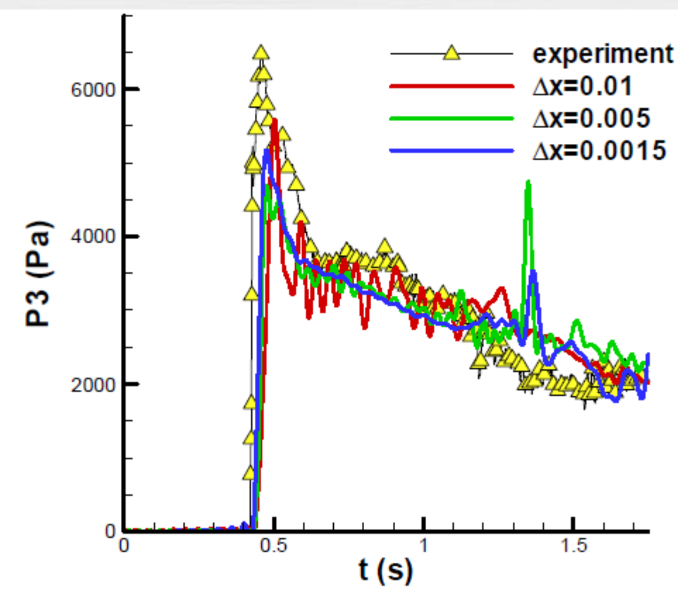
Validation on dam breaking test cases





- Good prediction of local loads:





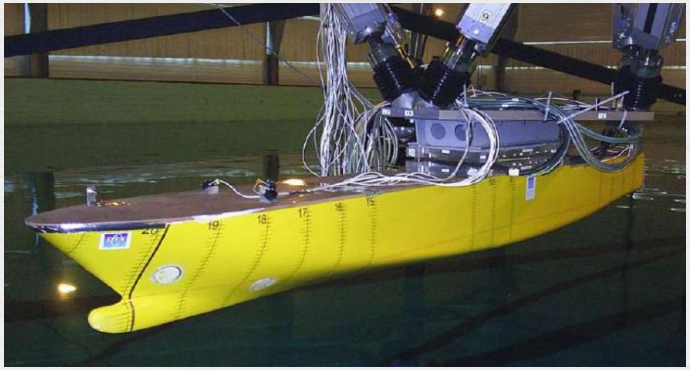
Marrone et al., *Comput. Meth. Appl. Mech. Eng. 200*, 2011

Fast dynamics free surface flows: naval applications

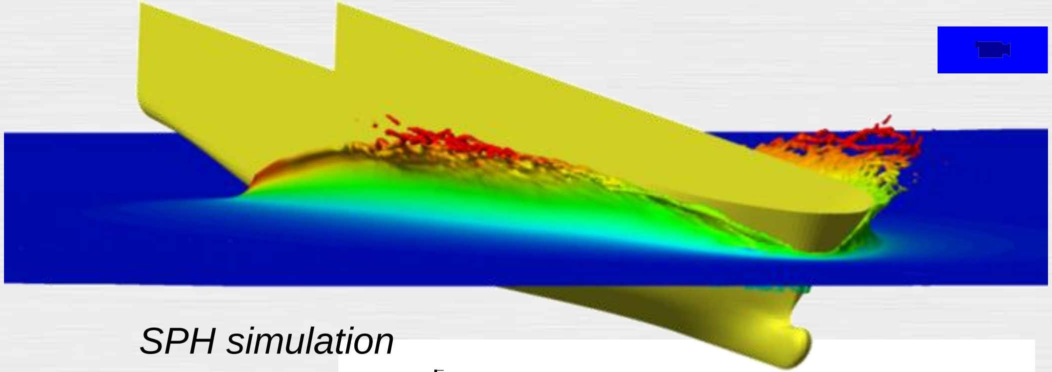
SPH-flow

Slamming impact

Maruzewski et al., *J. Hydraul. Res. 48*, 2010

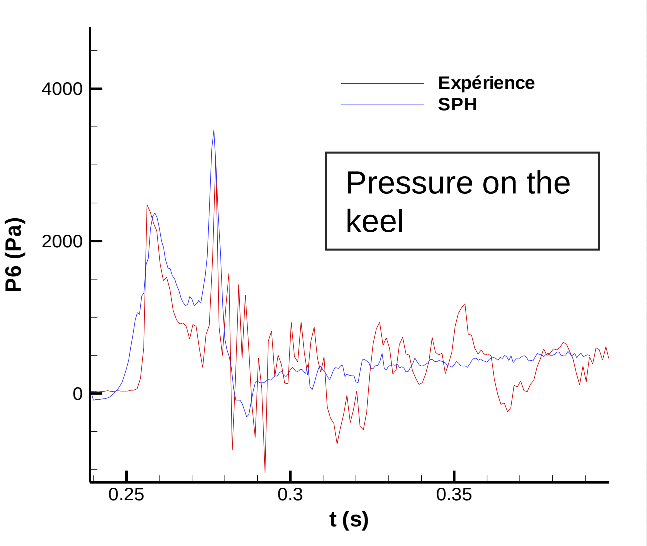


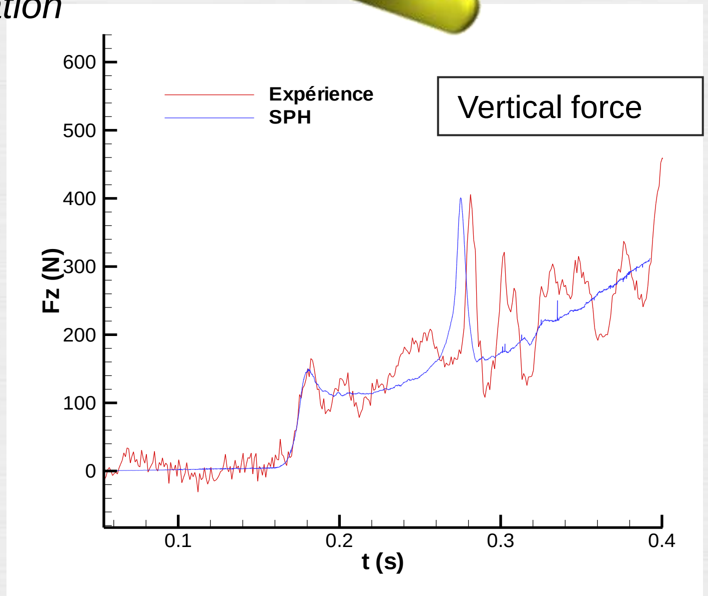
Experiment (ECN wave tank)



SPH simulation

- 3 million particles
- 6m/s impact
- 250m long ship at real scale





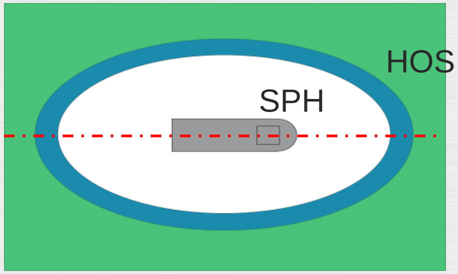
Fast dynamics free surface flows: naval applications

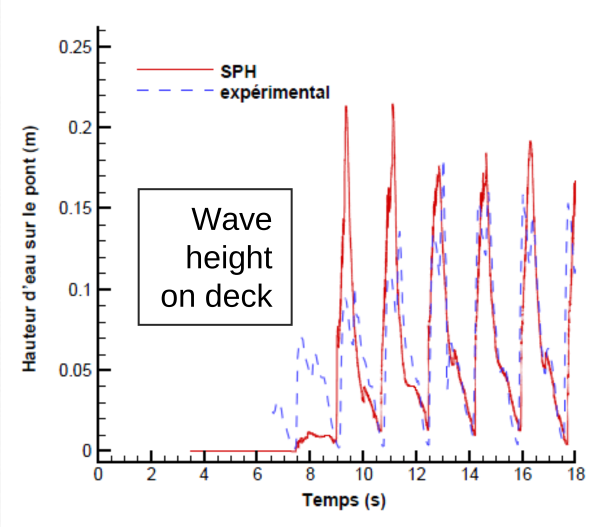
SPH-flow

Violent wave-structure interactions

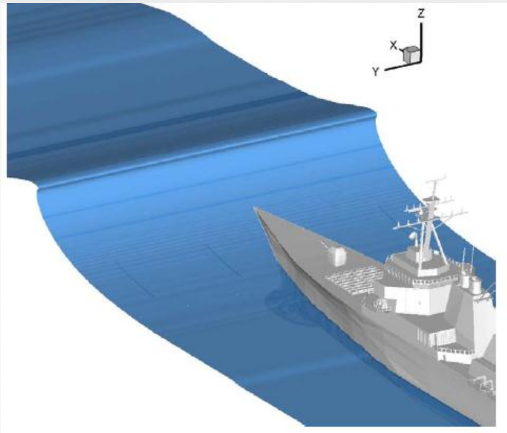
- FPSO in severe sea state

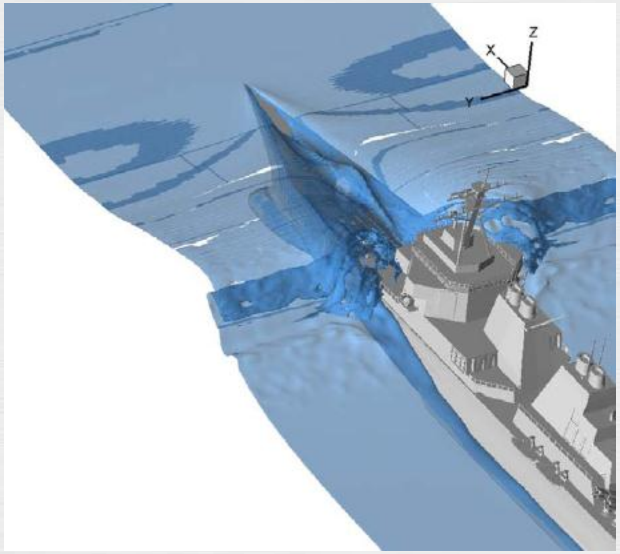
coupling strategy:





- Fregate facing a dimensioning wave





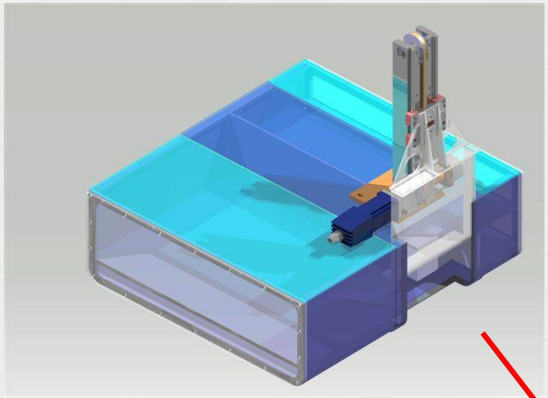
Fast dynamics free surface flows: naval applications

Other naval applications

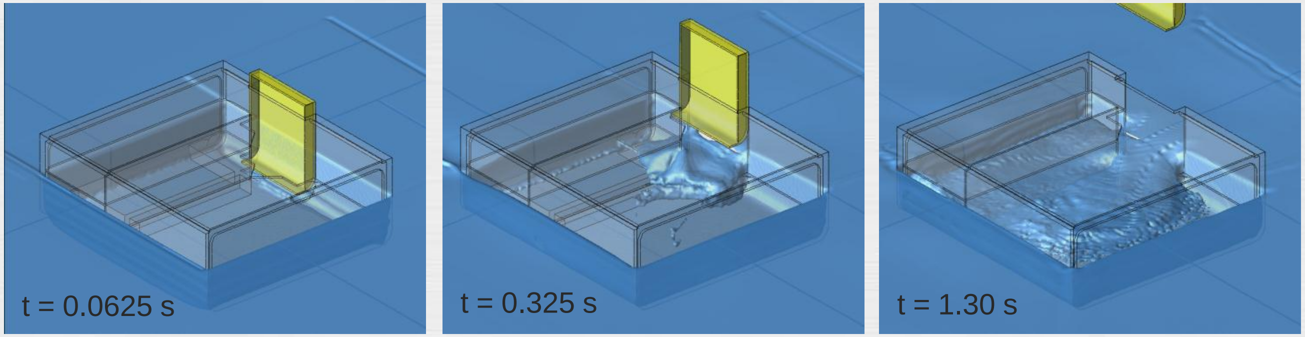
SPH-flow

- Flooding

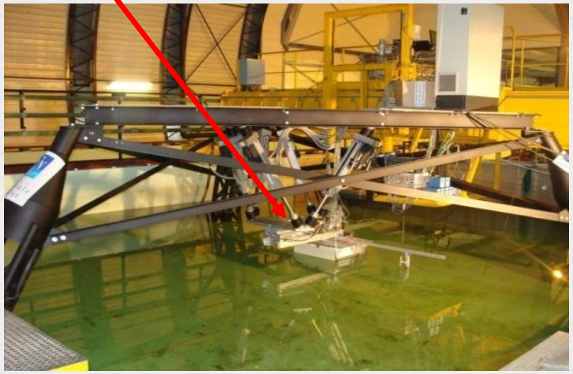
SPH calculation

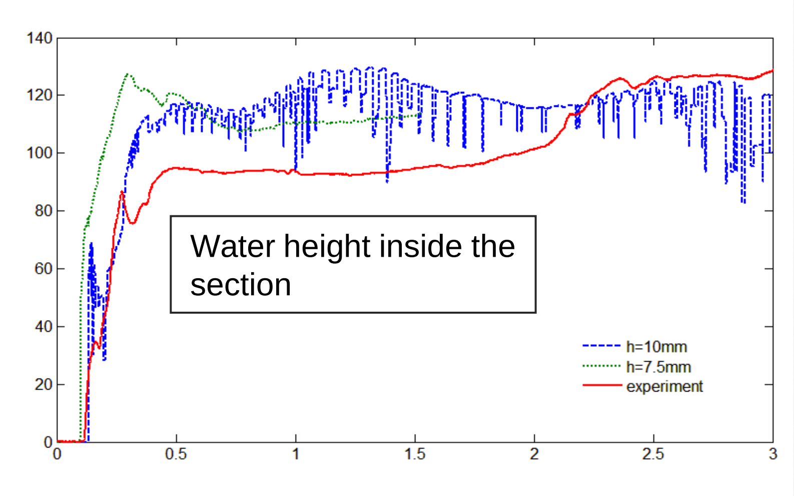


Experiment in the ECN tank



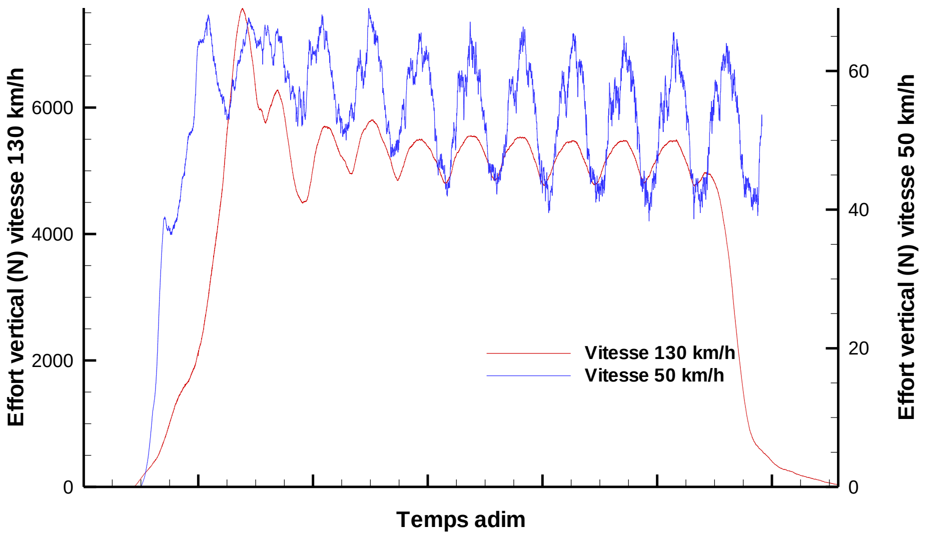
Miship Ro-Ro ferry section flooding

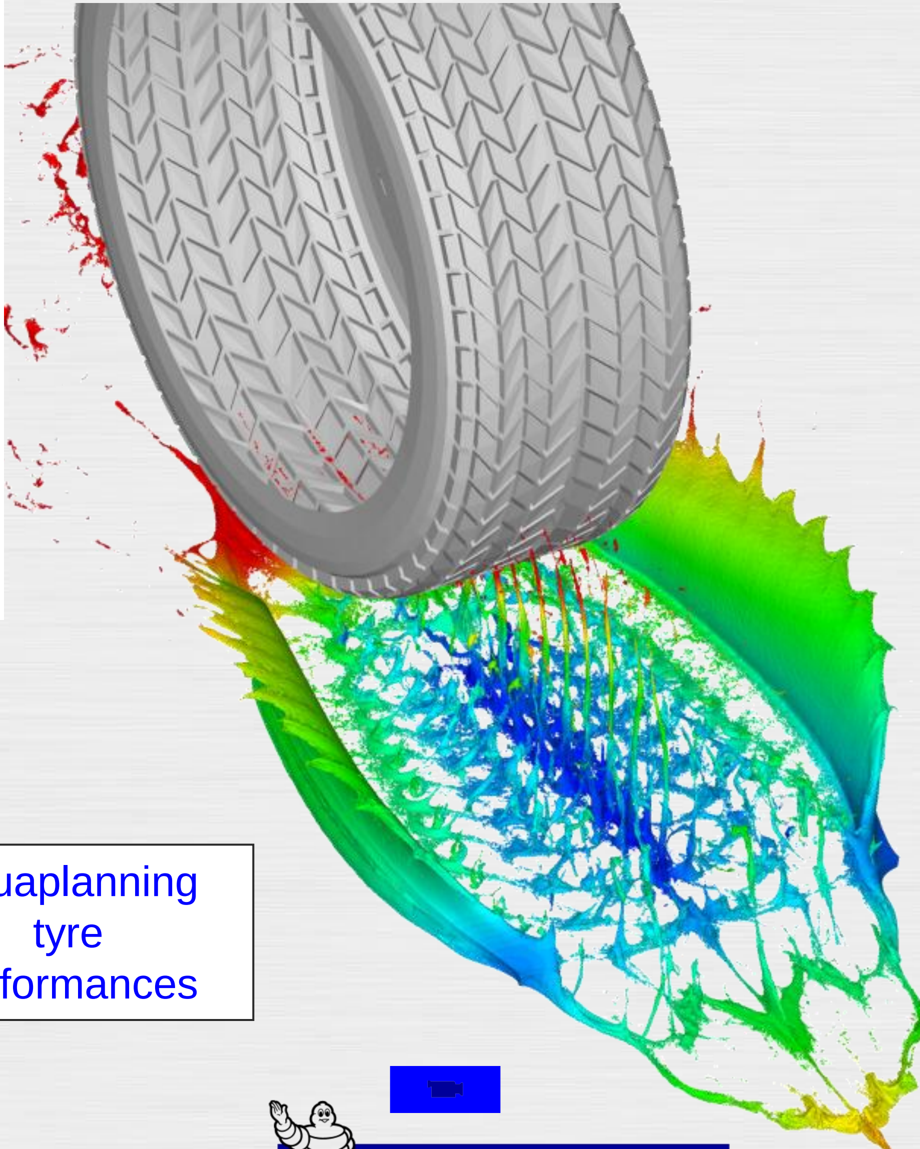




- Fast ship

Fast dynamics free surface flows: tyre aquaplanning





- very complex geometry
- 6 million particles
- 130 km/h and 50 km/h velocity
- 1cm-thick layer of water
- tyre deformations imposed

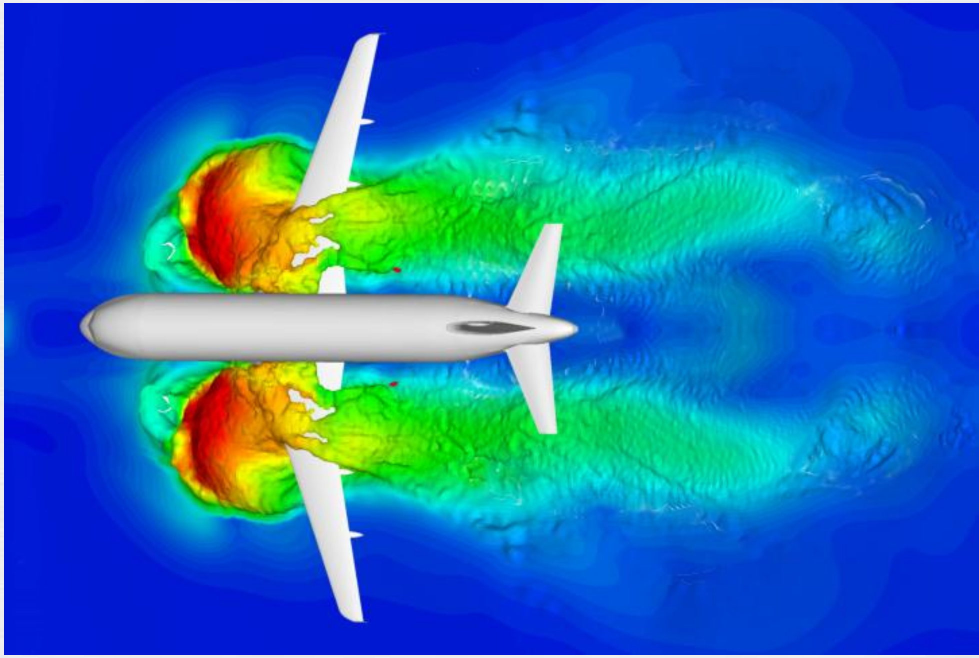
Aquaplanning tyre performances

Fast dynamics free surface flows: other demonstrative applications

SPH-flow

Aeronautics: aircraft ditching

- 11° initial trim angle
- 65 m/s initial velocity
- Air effects neglected
- 3 DOF (X, Z and θ)
- 6,000,000 particles



Energy: offshore wind turbines

Mesh-free Lagrangian modelling of fluid dynamics

David LE TOUZÉ, Ecole Centrale Nantes

CFD 2011, keynote lecture

Mesh-free Lagrangian methods in CFD

Smoothed-Particle Hydrodynamics (SPH)

Fast-dynamics free-surface flows

Multi-fluid flows

Fluid-Structure Interactions

HPC: massive interactive simulations

Multiphase flow modelling: method 1

- Modification of the standard formulation to preserve density of each phase

$$\langle\rho_i\rangle = \frac{\sum_{j\in\mathcal{X}} dm_j W_j(\mathbf{x}_i)}{\Gamma_i^{\mathcal{X}}};\ \Gamma_i^{\mathcal{X}} = \sum_{k\in\mathcal{X}} W_k(\mathbf{x}_i)dV_k;\ p_i = \frac{c_{0\mathcal{X}}^2\rho_{0\mathcal{X}}}{\gamma_{\mathcal{X}}}\left[\left(\frac{\langle\rho_i\rangle}{\rho_{0\mathcal{X}}}\right)^{\gamma_{\mathcal{X}}} - 1\right];\forall \mathbf{x}_i\in\mathcal{X}$$

$$\begin{cases} \dfrac{\mathrm{D}\log\mathcal{J}_i}{\mathrm{D}t} = \sum_j(\mathbf{u}_j - \mathbf{u}_i)\cdot\dfrac{\nabla W_j(\mathbf{x}_i)}{\Gamma_i}dV_j; \quad \Gamma_i = \sum_k W_k(\mathbf{x}_i)dV_k \\ \dfrac{\mathrm{D}\mathbf{u}_i}{\mathrm{D}t} = -\dfrac{1}{\rho_i}\sum_j\left(\dfrac{p_i}{\Gamma_i}+\dfrac{p_j}{\Gamma_j}\right)\nabla W_j(\mathbf{x}_i)dV_j + \mathbf{F}_V + \mathbf{F}_S + \mathbf{F}_B \quad ; \quad \dfrac{\mathrm{D}\mathbf{x}_i}{\mathrm{D}t} = \mathbf{u}_i \end{cases}$$

  - Continuous Surface Stress surface tension modelling

$$\mathbf{F}_{Si}^{\chi y} = \sigma\kappa\delta_I\mathbf{n}_I$$

$$\begin{cases} \mathbf{T}_{Si}^{\chi t} = \sigma^{\chi y}\dfrac{1}{|\nabla C_i|}\left(\dfrac{1}{d}|\nabla C_i|^2 - \nabla C_i\otimes\nabla C_i\right) \\ \mathbf{F}_{Si}^{\chi y} = div(\mathbf{T}_{Si}^{\chi y}) \qquad \forall i\in\chi \end{cases}$$

Grenier et al., *J. Comput. Phys.*, 2009

Multiphase flow modelling: method 2

- In the Riemann-solver ALE formulation, prevent any mass flux through the interface

$$\frac{d(\mathrm{x}_{\mathrm{i}})}{dt} = \mathrm{v}_0(\mathrm{x}_{\mathrm{i}},\ \mathrm{t})$$

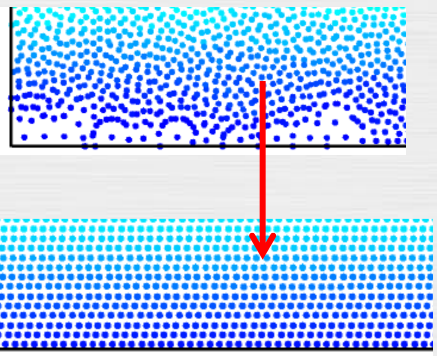
$$\frac{d(\omega_{\mathrm{i}}\rho_{\mathrm{i}})}{dt} + \omega_{\mathrm{i}} \sum_{j\in D_i} \omega_{\mathrm{j}} 2\rho_{E,ij} \left(\cancel{\mathrm{v}_{E,ij} - \mathrm{v}_0}(\mathrm{x}_{\mathrm{ij}},\ \mathrm{t}) \right) . \nabla_i W_{ij} = 0$$

$$\frac{d(\omega_{\mathrm{i}}\rho_{\mathrm{i}}\mathrm{v}_{\mathrm{i}})}{dt} + \omega_{\mathrm{i}} \sum_{j\in D_i} \omega_{\mathrm{j}} 2\left[\rho_{E,ij}\mathrm{v}_{E,ij} \otimes \left(\cancel{\mathrm{v}_{E,ij} - \mathrm{v}_0}(\mathrm{x}_{\mathrm{ij}},\ \mathrm{t}) \right) + p_{\mathrm{E,ij}} \right] . \nabla_i W_{ij} = \omega_{\mathrm{i}}\rho_{\mathrm{i}}\mathrm{g}$$

Leduc et al., *Proc. 5th SPHERIC workshop*, 2010

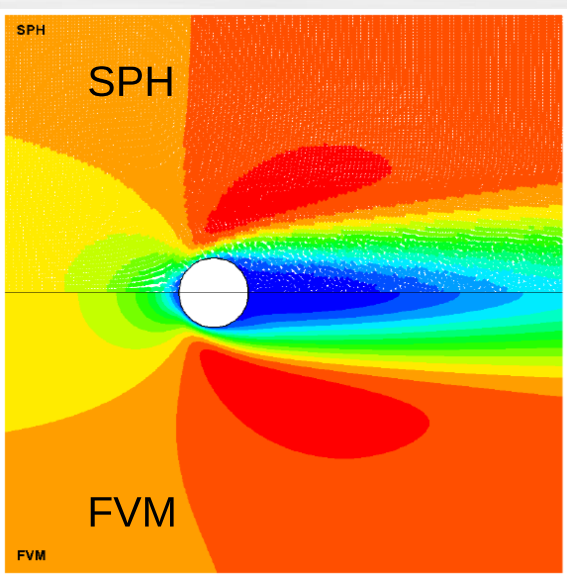
Multiphase flow modelling: laminar flow validation

- Application of ghost viscous condition only on the pressure gradient term to preserve the hyperbolicity of the system inviscid part (De Leffe et al., *Proc. 6th SPHERIC workshop, 2011)*



$$\frac{dE}{dt} = -\sum_{i\in P(\Omega)} \sum_{j\in P(\Omega)\cup P(\partial\Omega)} m_i m_j \left(\frac{p_j \overline{v_i}}{\rho_j^{\,2}} + \frac{p_i \overline{v_j}}{\rho_i^{\,2}} + \frac{1}{2}\Pi_{ij}\left(\overline{v_i} + \overline{v_j}\right) \right).\nabla_i W_{ij}$$
$$= \Delta E + \Delta E^{\Pi}$$

- Validation on flow past cylinder at Re=200



	Strouhal	C_d
SPH	2.0	1.47
Experiment	1.9	1.3

experiment by Tritton (1959)

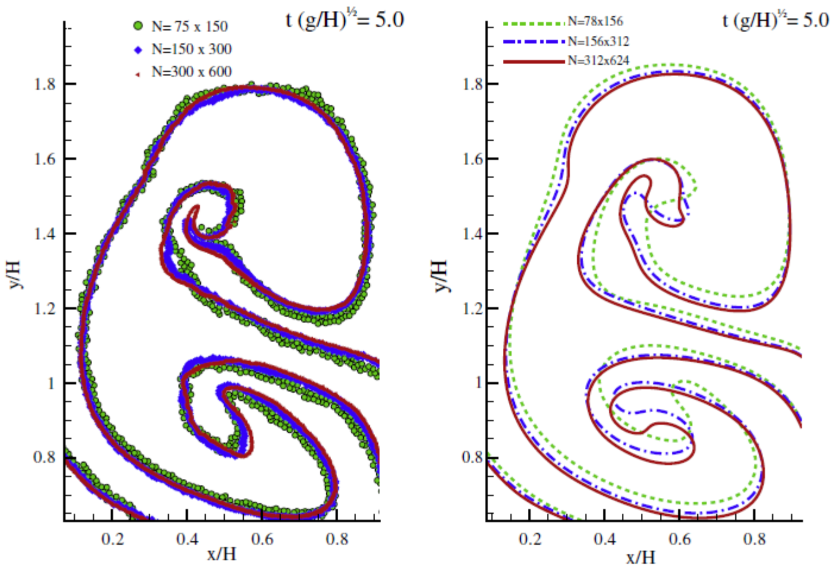
Multiphase flow modelling: validations

Rayleigh-Taylor instabilities

collaboration with CNR-INSEAN (Rome)

Grenier et al., *J. Comput. Phys.*, 2009

Fast convergence on the interface shape thanks to Lagrangian nature of SPH



(a) Present SPH model (b) Level-Set model

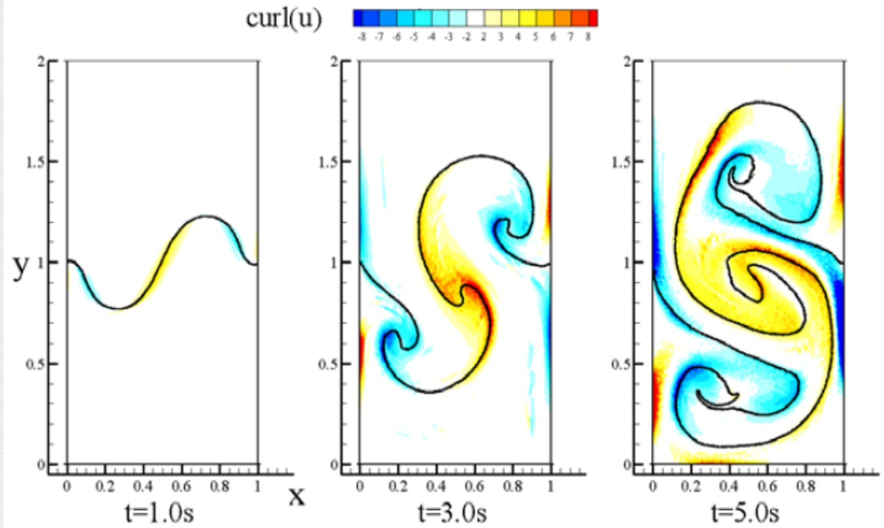


Figure 5: Vorticity of the present SPH formulation.

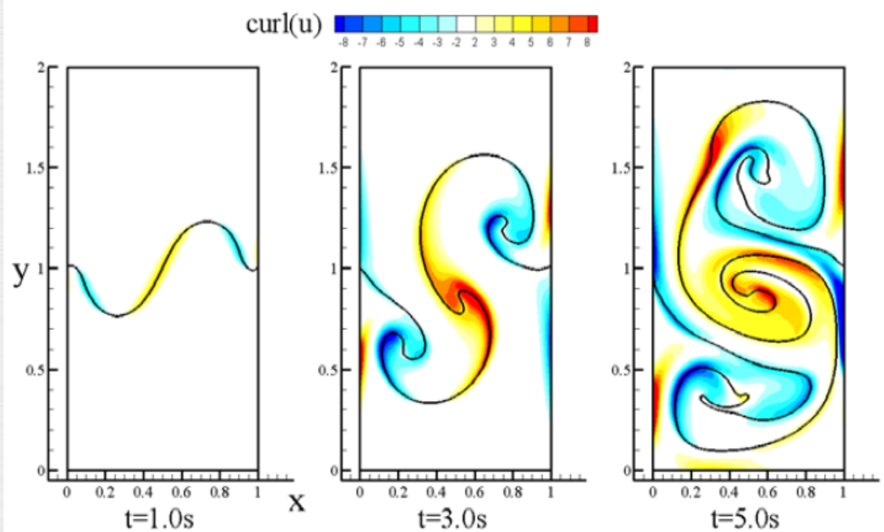
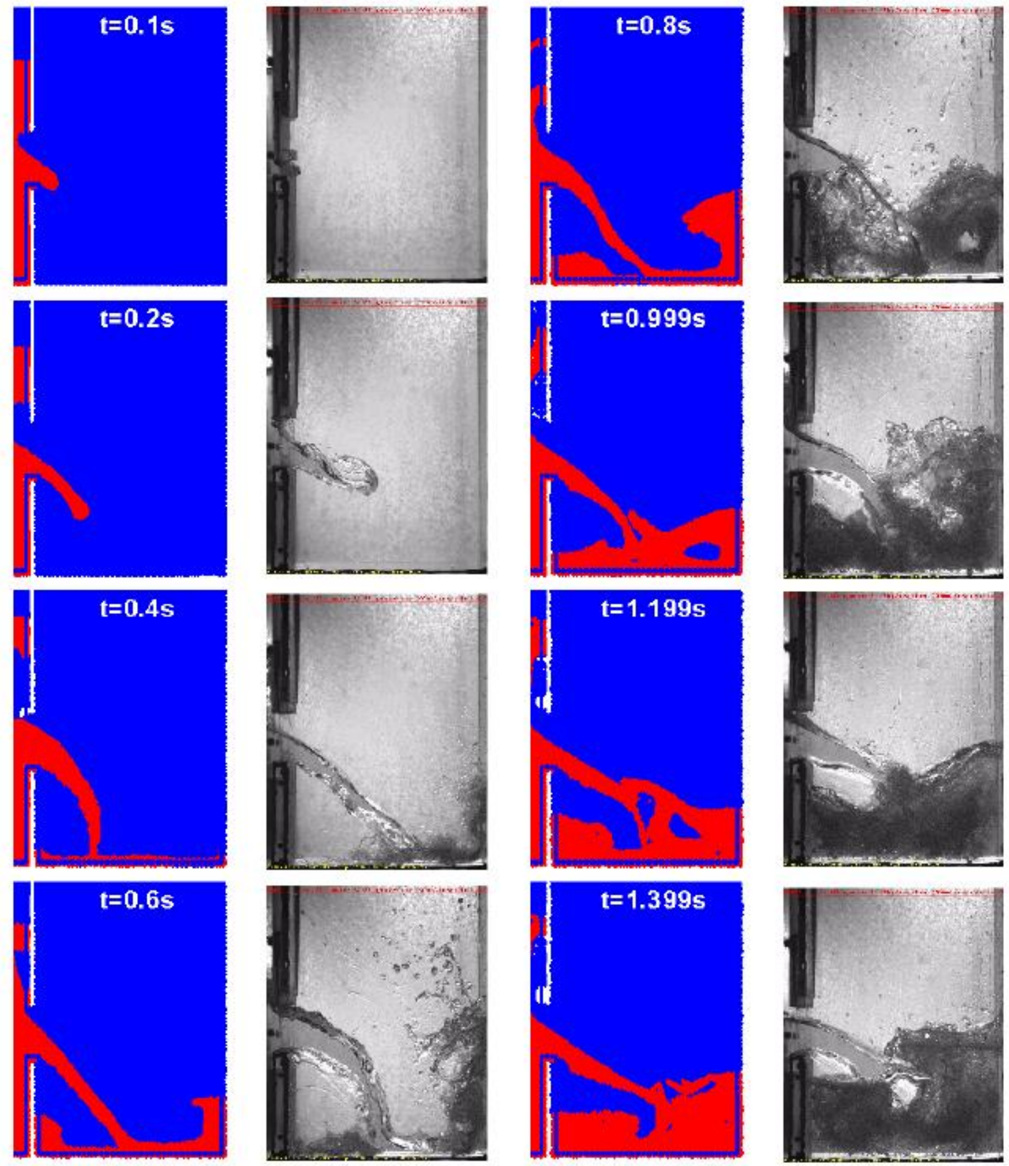


Figure 6: Vorticity of the NS-LS formulation.

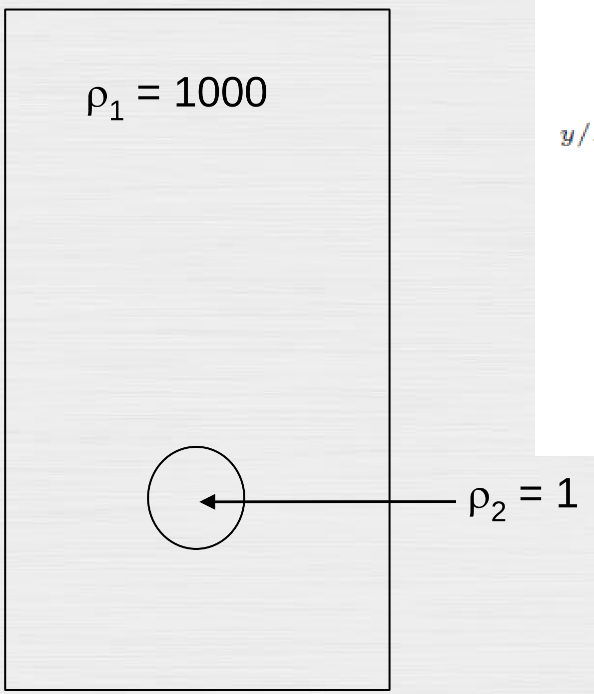
Multiphase flow modelling: validations

Flooding experiment (made in ECN) comparison

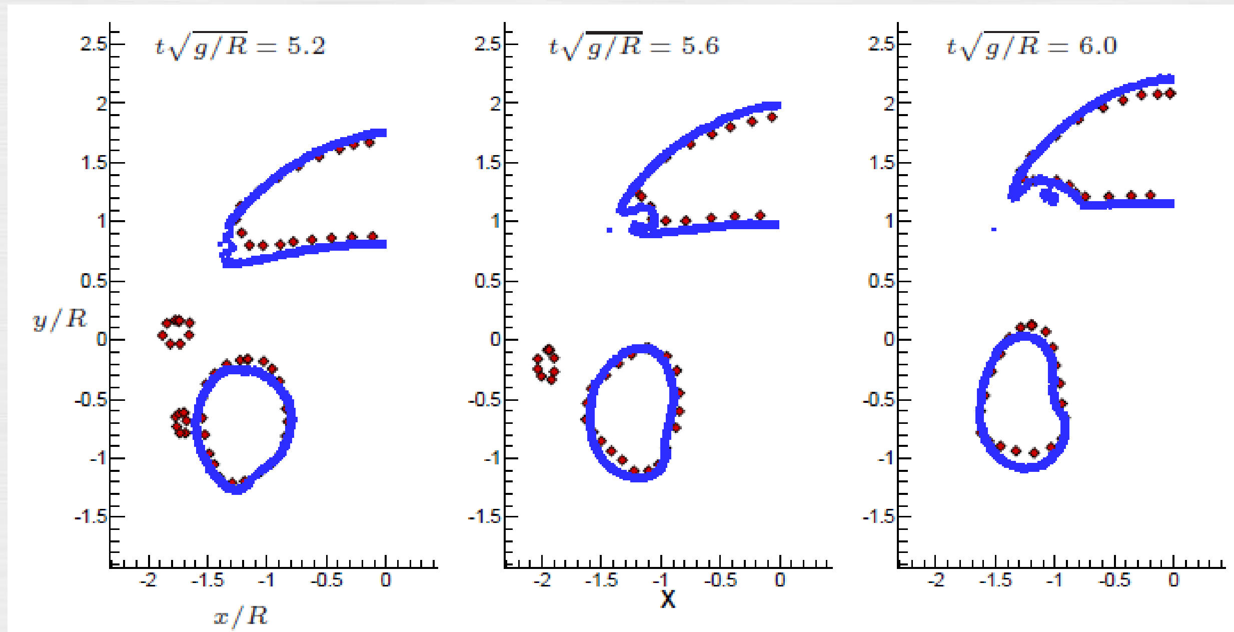


Bubbly flows: isolated bubble

Large Bond number bubble rise



◆ Level Set (Sussman) ● SPH



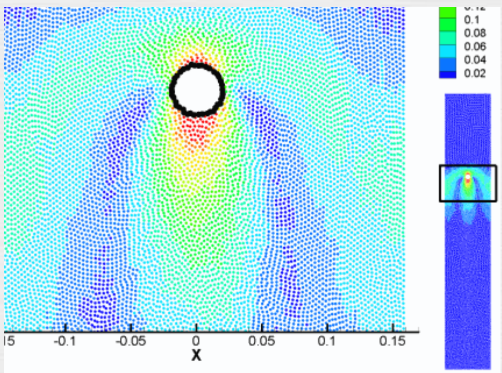
Grenier et al., *J. Comput. Phys.*, 2009

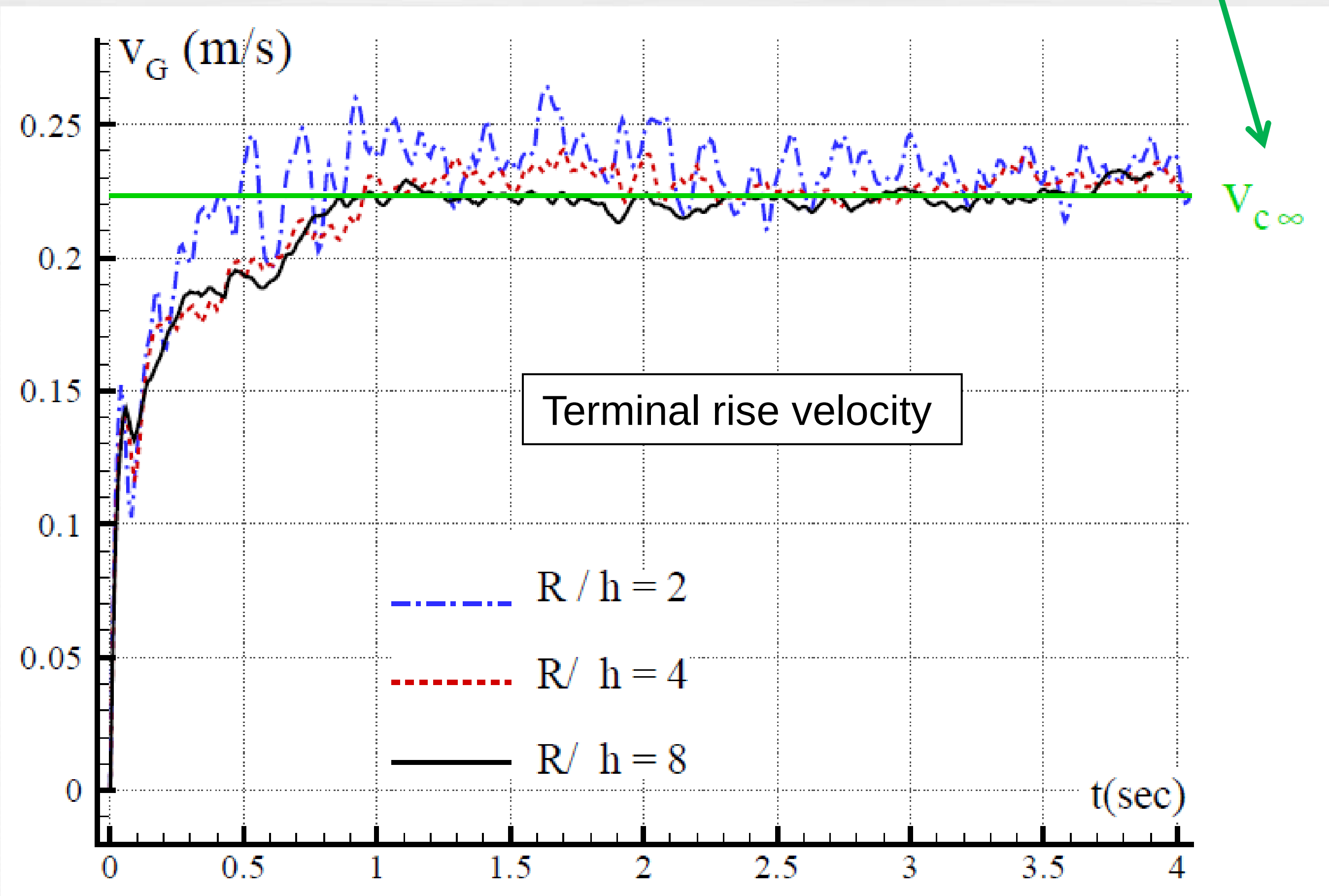
collaboration with CNR-INSEAN (Rome)

Bubbly flows: isolated bubble

Small Bond number
bubble rise

analytic solution

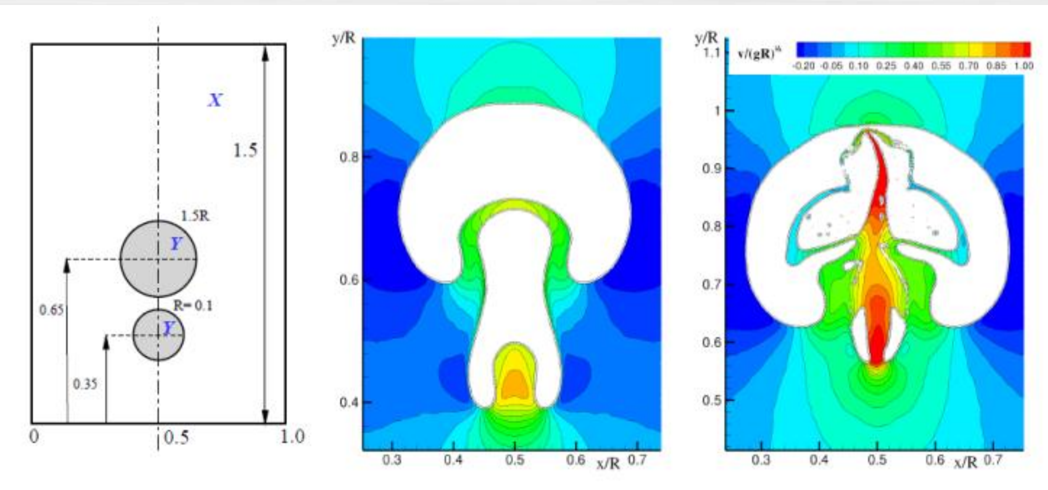


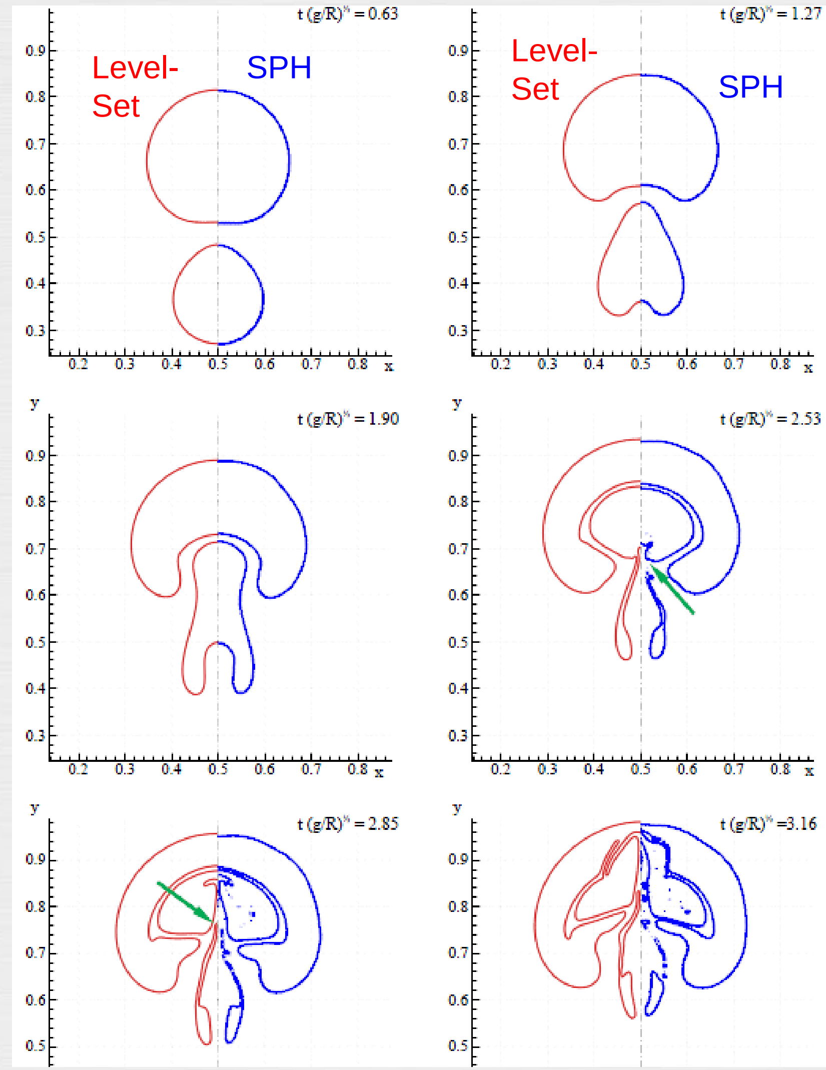


Grenier et al., *submitted to J. Multiphase Flows*

collaboration with CNR-INSEAN (Rome)

Bubbly flows: two-merging bubbles





Infinite Bond number and density ratio = 10

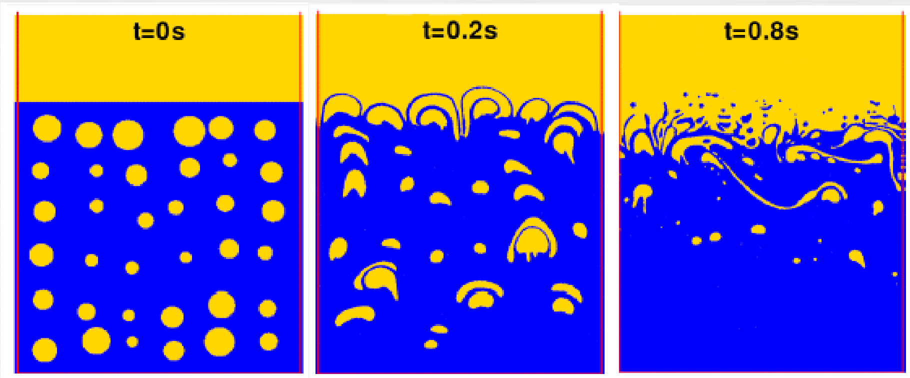
Grenier et al., *submitted to J. Multiphase Flows*

collaboration with CNR-INSEAN (Rome)

Bubbly flows: water-oil separation

collaboration with CNR-INSEAN (Rome)

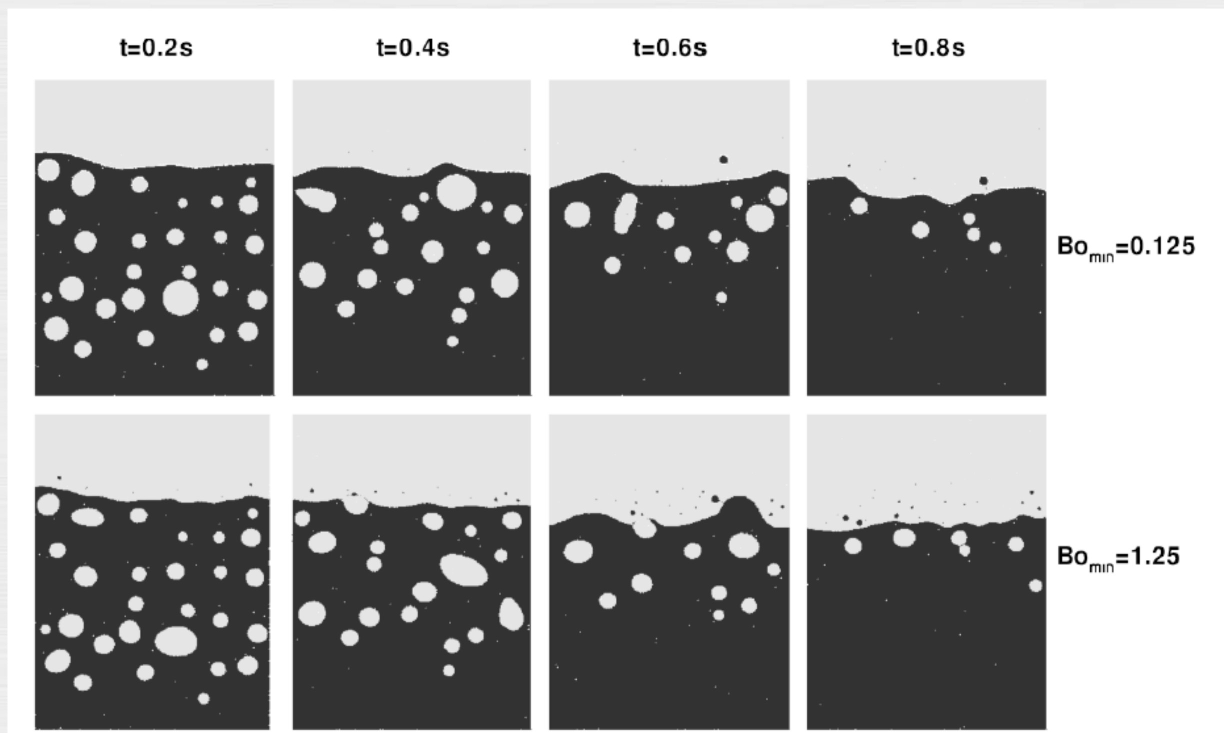
naive gravity separator



Infinite Bond number

$Bo_{min} = 0.1$

$Bo_{min} = 1$



Grenier et al., *submitted to J. Multiphase Flows*

Mesh-free Lagrangian modelling of fluid dynamics

David LE TOUZÉ, Ecole Centrale Nantes

CFD 2011, keynote lecture

Mesh-free Lagrangian methods in CFD

Smoothed-Particle Hydrodynamics (SPH)

Fast-dynamics free-surface flows

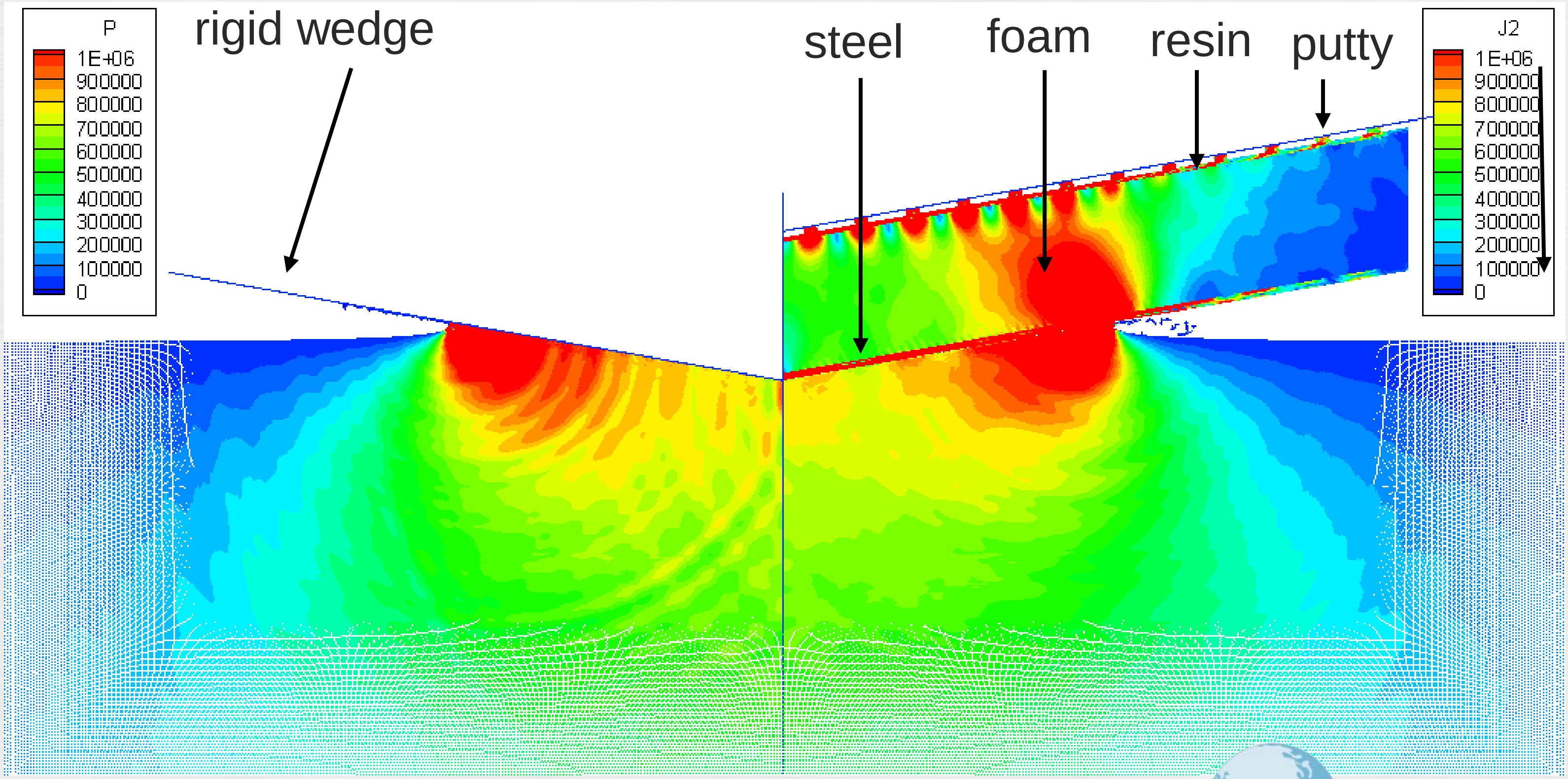
Multi-fluid flows

Fluid-Structure Interactions

HPC: massive interactive simulations

Fluid-Structure coupling: SPH monolithic coupling

- Principle:
  - each particle has its own continuous medium local equations
  - mass fluxes through the interface are prevented, as for multiphase method 2



Fluid-Structure coupling: SPH-FEM strategy

SPH-FEM coupling

- Weak coupling strategy, at each substep of Runge-Kutta 4

Fluid		Structure
• Imposition of boundary displacement • Timestep calculation • Resolution and time advance	Pressure load → ← Boundary displacement	• Imposition of boundary conditions (pressure loading) • Resolution and time advance

- Timestep of FSI simulations is based on SPH timestep (very small)
- In practice, several processors are used to compute flow evolution while only one is used for solid mechanics
- FEM solver is fast and its computation time is masked by the flow solver => scalabity of SPH-Flow code is unaffected

Fluid-Structure coupling: SPH-FEM strategy

Advantages of SPH-FEM coupling

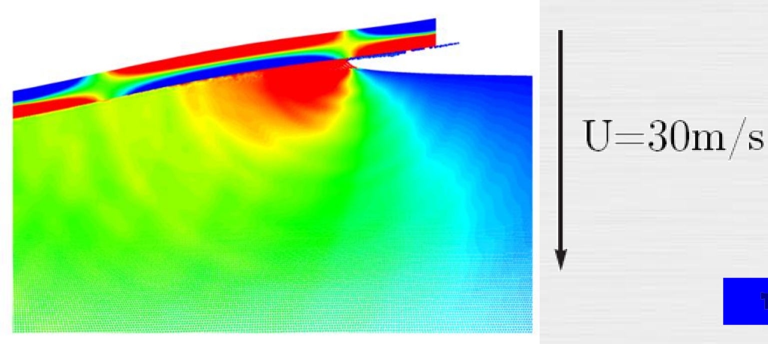
- FEM is an accurate and fast method to simulate structures behaviour under pressure loads
- SPH method easily handles fragmentations and reconnections of free surface
- No free-surface tracking method is needed
- No specific treatment is needed at the fluid-structure interface

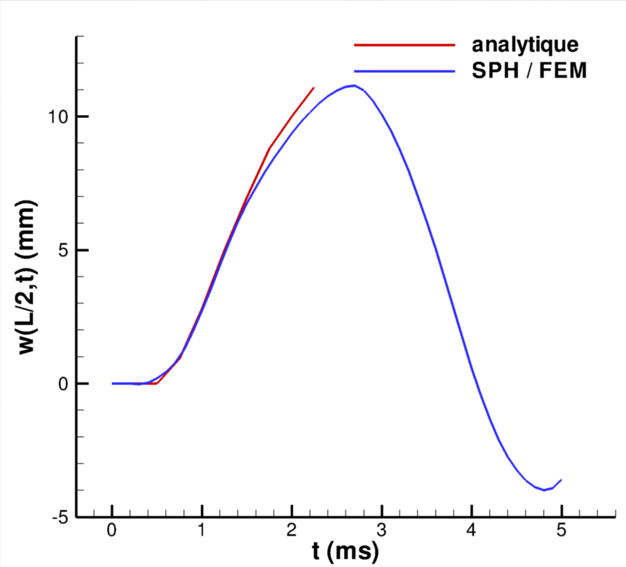
SPH-FEM coupling: validation test cases

Fourey et al., *submitted to Comput. Meth. Appl. Mech. Engng.*

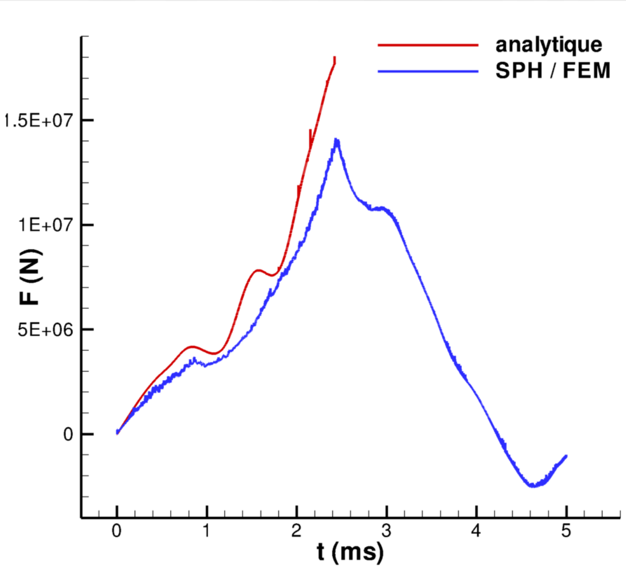
Half-wedge hydroelastic impact

- Aluminium beam
- High velocity impact
- Motion of the beam is prescribed at its two ends

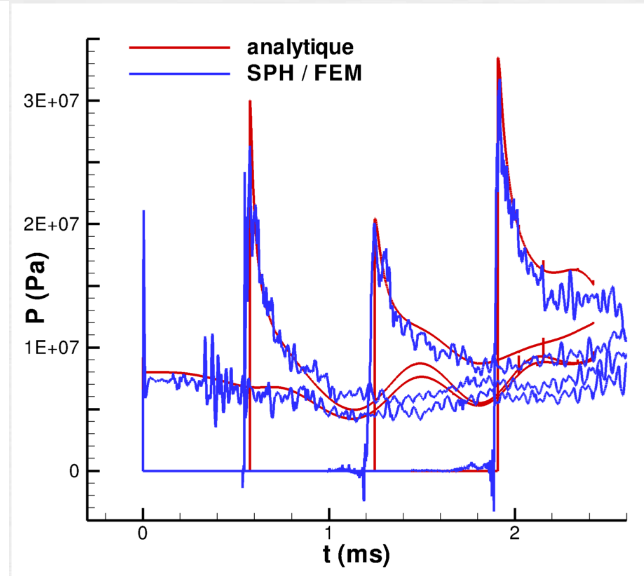




Beam deflection at mid point



Total vertical force

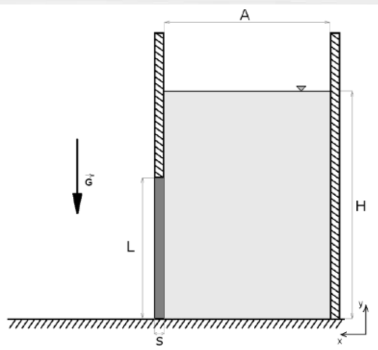


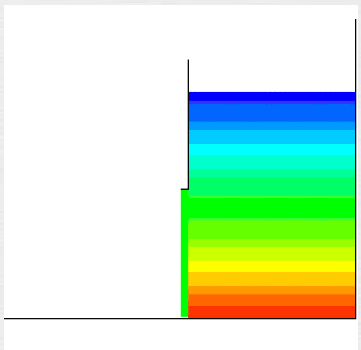
Pressure probe measurements

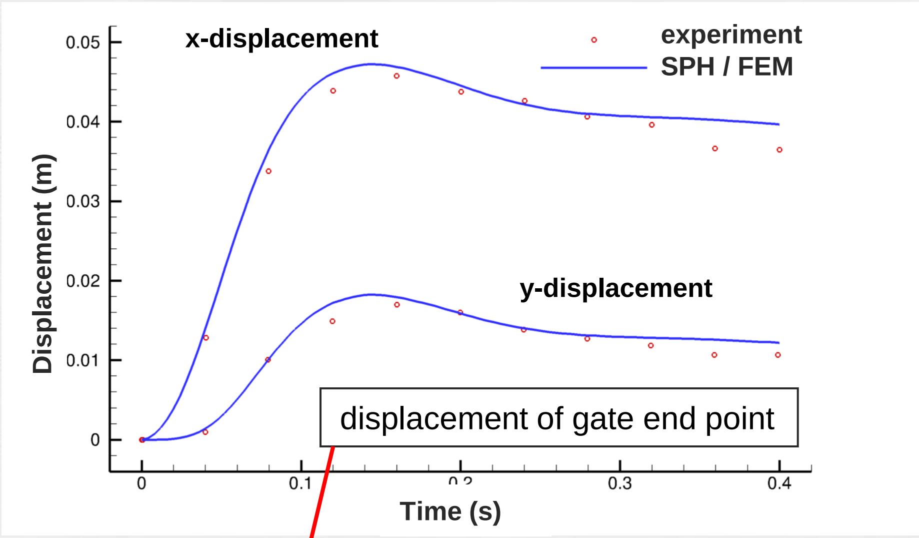
SPH-FEM coupling: validation test cases

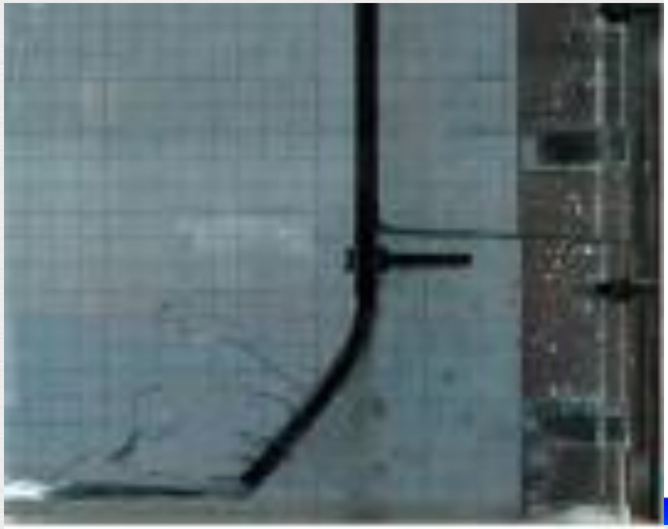
Fourey et al., *submitted to Comput. Meth. Appl. Mech. Engng.*

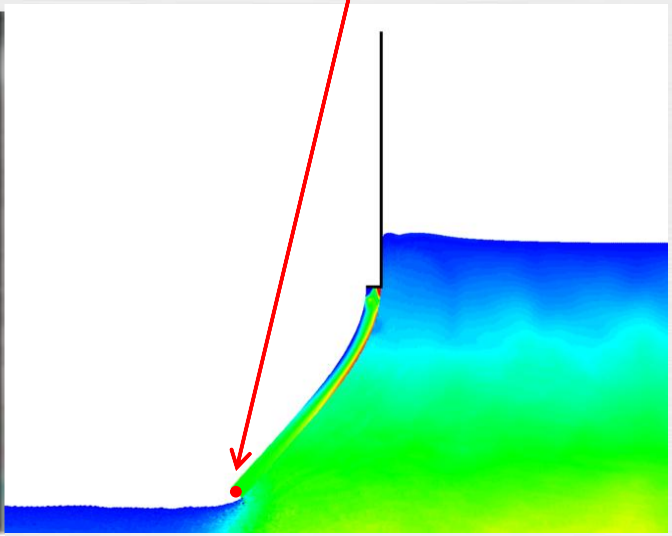
Water exit under an elastic gate







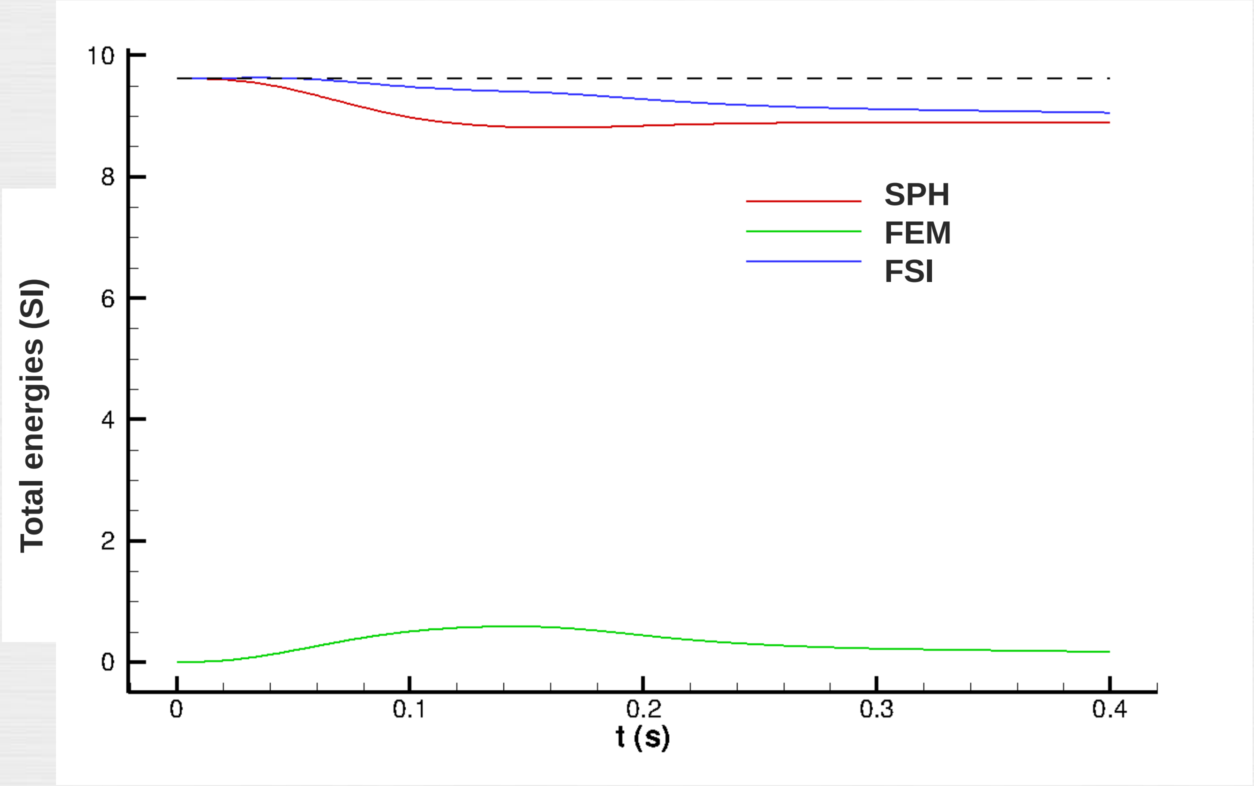




SPH-FEM coupling: validation test cases

Water exit under an elastic gate

- Total energy evolution for the FSI simulation:

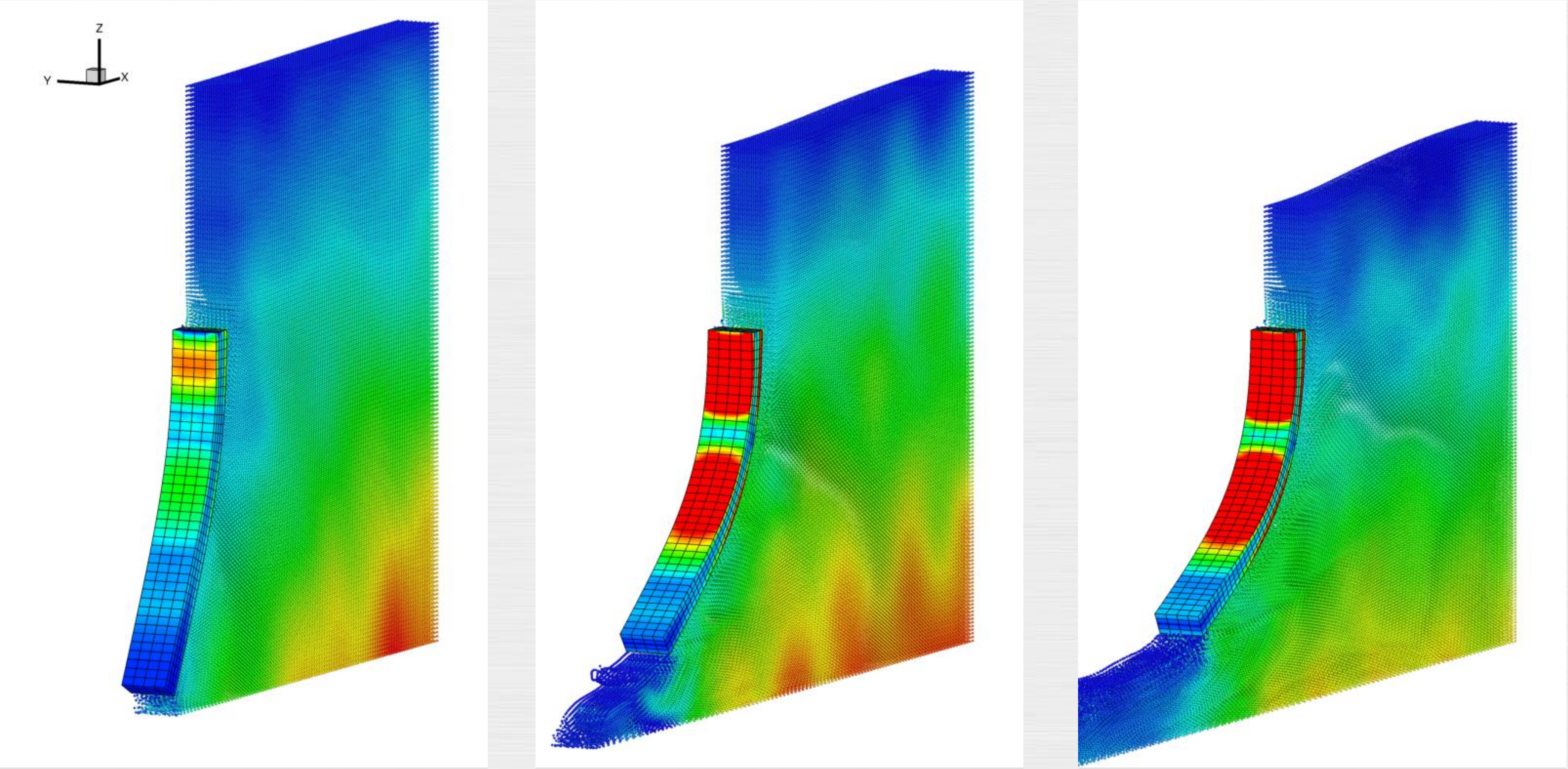


Fourey et al., *submitted to Comput. Meth. Appl. Mech. Engng.*

SPH-FEM coupling: three-dimensional extension

SPH-flow

3D water exit under an elastic gate



Fourey et al., *submitted to Comput. Meth. Appl. Mech. Engng.*

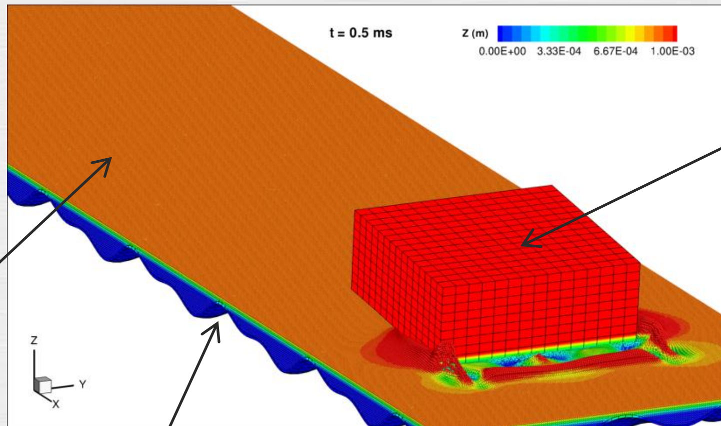
SPH-FEM coupling: Example of industrial application

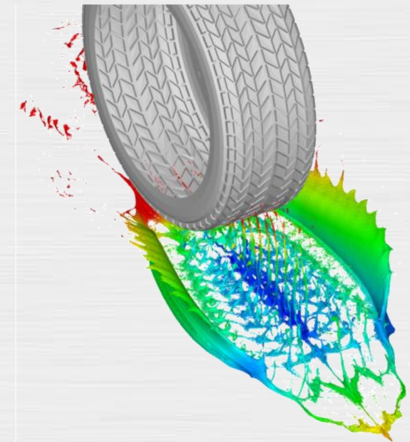
SPH-flow

Tyre deformation when rolling on a wet rough road

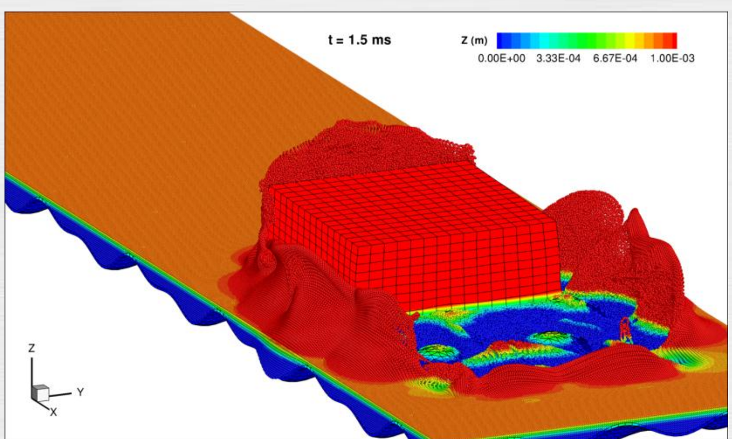
tyre part

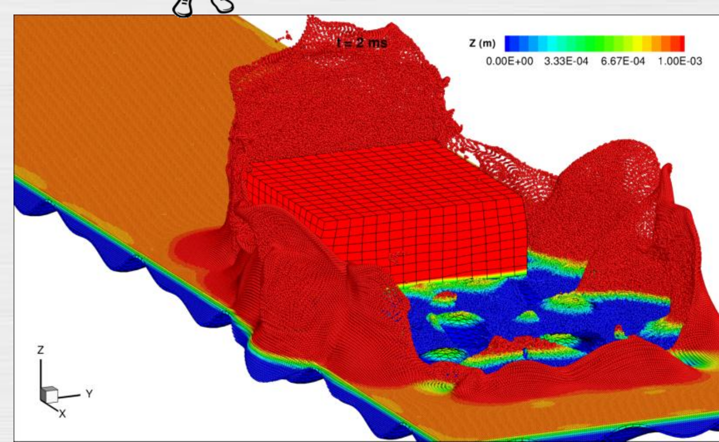
still water





rough road shape





MICHELIN

collaboration with Michelin

Mesh-free Lagrangian modelling of fluid dynamics

David LE TOUZÉ, Ecole Centrale Nantes

CFD 2011, keynote lecture

Mesh-free Lagrangian methods in CFD

Smoothed-Particle Hydrodynamics (SPH)

Fast-dynamics free-surface flows

Multi-fluid flows

Fluid-Structure Interactions

HPC: massive interactive simulations

FP7 European project NextMuSE: SPH & HPC

http://nextmuse.cscs.ch

Initialize

(CAD geometry, no mesh)

Compute

(continuously)

Analyze

(on the fly)

Render

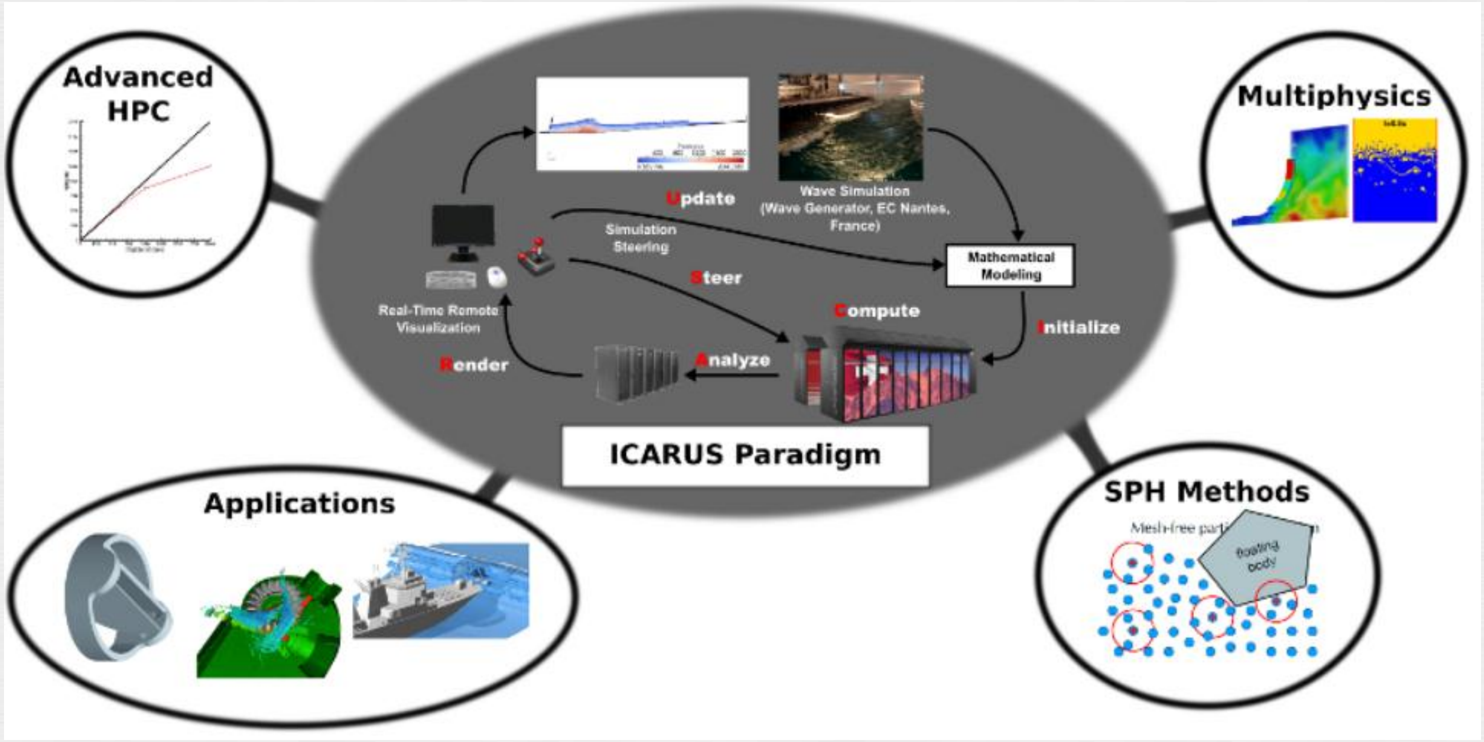
(immersive)

Update

(interactive)

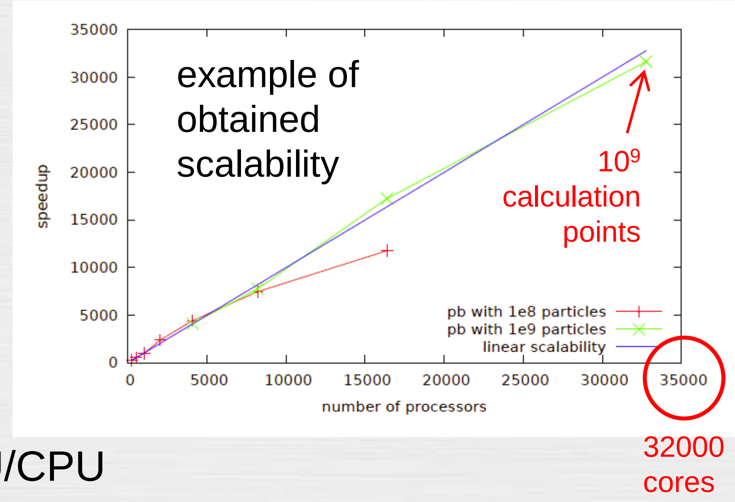
Steer

(interactive)



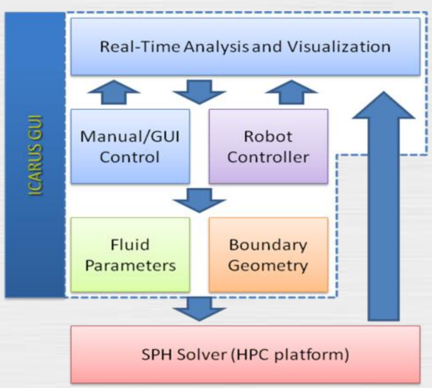
FP7 European project NextMuSE: SPH & HPC

High-performance (HPC) flexible simulation environment

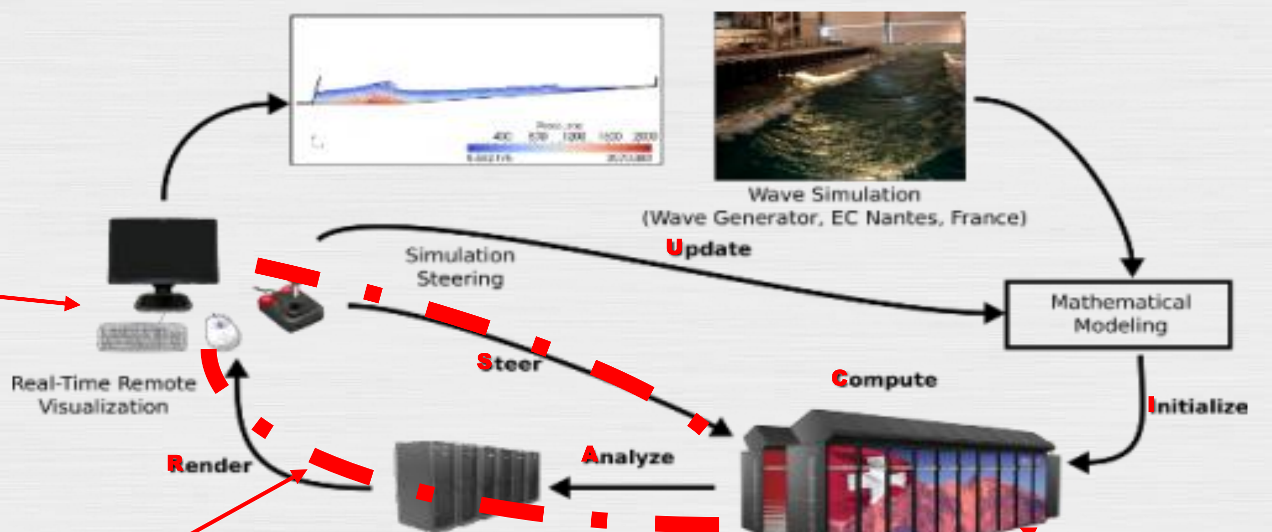


- Immersive environment ICARUS:
  visualization, analysis, steering of geometry/parameters
  simulation runs on a massively-parallel architecture

- Optimization of the method on modern architectures:
  « manycore » CPUs (MPI/OpenMP) and hybrid GPGPU/CPU



Interactive ICARUS robot, accessible remotely



Memory pipe (Distributed Shared Memory buffer)

O(10^4) calculation-core simulation

Conclusion

- Research on mesh-free Lagrangian methods is growing fast and they are reaching industrial maturity on some problems

- These methods can represent a valuable alternative where mesh-based methods find their limits

- They are well suited to:
  - fast dynamics problems, especially with free-surface
  - multi-species/multiphase problems with interfaces: multi-fluid, fluid-structure…

  for which they are simple and accurate

- These methods could probably be used more in the oil & gas industry (presently mainly applied to ocean engineering in this industrial field)