Universitat Politècnica de Catalunya, Barcelona, 15 – 19 April 2013

Numerical Simulation of Fast Transient Phenomena in Fluid-Structure Systems

A Short Course by F. Casadei
Retired from European Commission, Joint Research Centre
Institute for the Protection and Security of the Citizen
T.P. 480, I-21027 Ispra (VA), Italy.
E-mail: casadeifolco@gmail.com

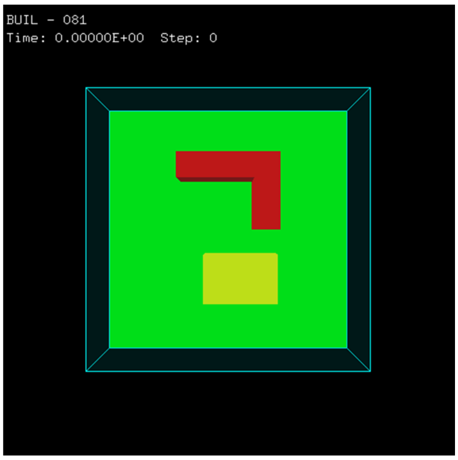
Contents

Detailed Contents of Part II

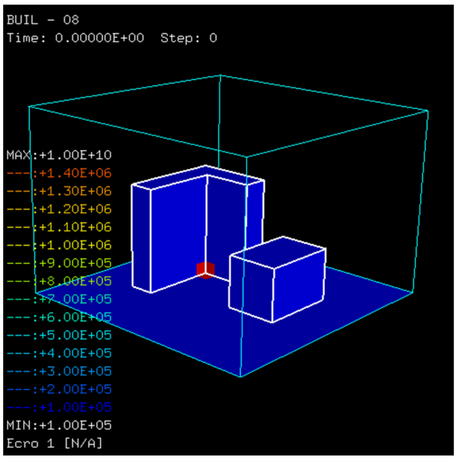
- Modeling the fluid domain
  - ➢ Euler equations
  - ➢ Finite Element discretization
  - ➢ Finite Volume discretization (N-C, C-C)
- Mesh rezoning algorithms
  - ➢ Motivation
  - ➢ Mean-based algorithms
  - ➢ Giuliani's algorithm
- Free surface modeling

3

Purely Fluid Example: Building Vulnerability (Preliminary)



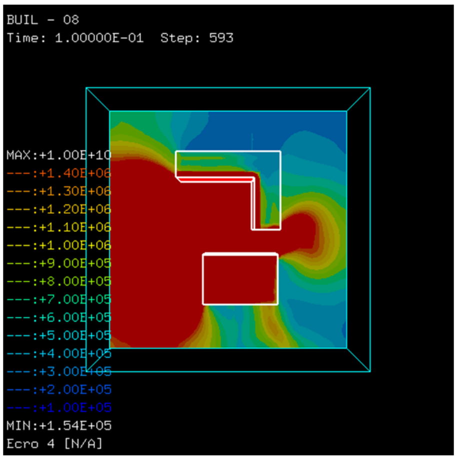
Model



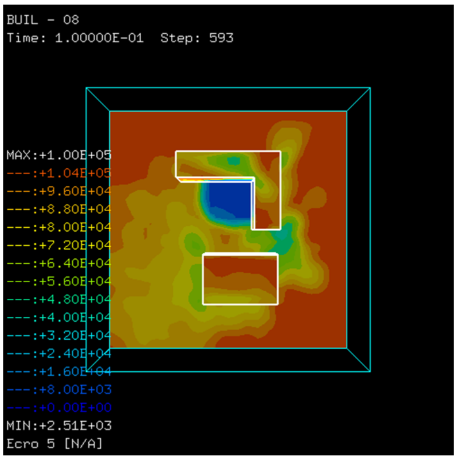
Simulation

4

Building Vulnerability (2)



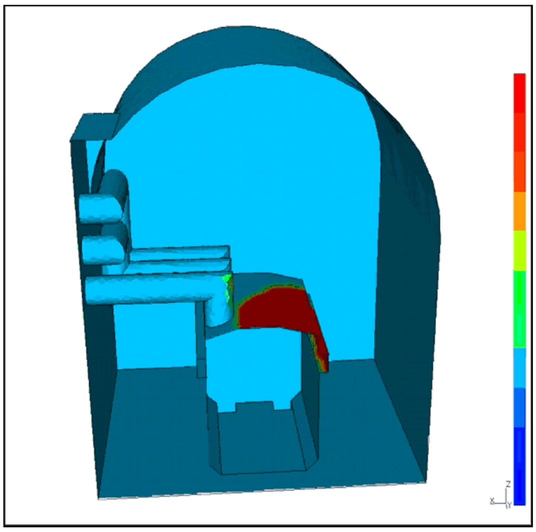
P_{max} contours



P_{min} contours

5

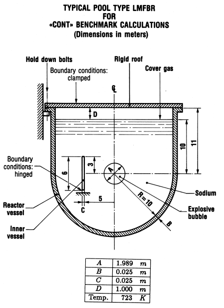
Some Further FSI Examples (Courtesy of ENEL-Hydro)



6

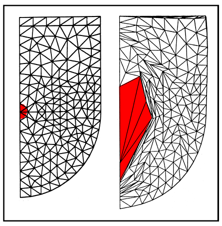
Modeling the fluid domain

- The fluid is assumed to be compressible and inviscid (high pressures and high pressure gradients)
- Viscous forces are negligible compared with pressure and inertia forces
- Governing equations are Euler equations, which express the conservation of mass, momentum and energy
- Plus suitable equation of state for the fluid material



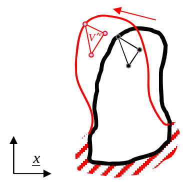
A	1.989	m
B	0.025	m
C	0.025	m
D	1.000	m
Temp.	723	K

Lagrangian Calculation



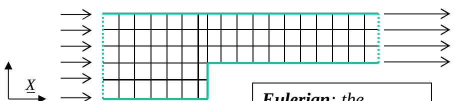
Modeling the fluid domain (2)

- The most “natural” description for the structural domain is the Lagrangian description



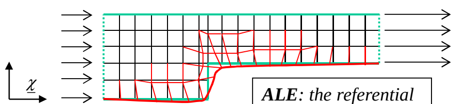
***Lagrangian**: the referential domain (mesh) is attached to material particles*

- In purely fluid problems, an Eulerian description is often more intuitive



***Eulerian**: the referential domain (mesh) is fixed*

- In problems involving both structures and fluids, a mixed description (ALE) can bring substantial benefits



***ALE**: the referential domain (mesh) is arbitrarily moving*

Euler equations

- These equations are conveniently expressed in <u>integral form</u>:

$\Phi =$ fluid domain

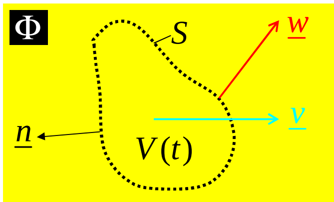
$V(t) =$ control volume

$S(t) =$ control surface

$\underline{n} =$ unit normal

$\underline{v}(\underline{x},t) =$ fluid velocity (particles)

$\underline{w}(\underline{x},t) =$ arbitrary velocity (mesh)



Euler equations (2)

$$\frac{dM}{dt} \equiv \frac{d}{dt}\int_{V(t)} \rho \, dV = \underbrace{\oint_{S(t)} \rho(\underline{w}-\underline{v})\bullet\underline{n} \, dS}_{\text{transport}} \qquad \text{(Mass)}$$

$$\frac{d\underline{Q}}{dt} \equiv \frac{d}{dt}\int_{V(t)} \rho\underline{v} \, dV = \underbrace{\oint_{S(t)} \rho\underline{v}(\underline{w}-\underline{v})\bullet\underline{n} \, dS}_{\text{transport}} - \underbrace{\int_{V(t)} \underline{\nabla}p \, dV}_{\text{pressure}} + \underbrace{\int_{V(t)} \rho\underline{g} \, dV}_{\text{body force}} \qquad \text{(Momentum)}$$

$$\frac{dE}{dt} \equiv \frac{d}{dt}\int_{V(t)} \rho e \, dV = \underbrace{\oint_{S(t)} \rho e(\underline{w}-\underline{v})\bullet\underline{n} \, dS}_{\text{transport}} - \underbrace{\oint_{S(t)} p\underline{v}\bullet\underline{n} \, dS}_{\text{pressure}} + \underbrace{\int_{V(t)} \rho\underline{g}\bullet\underline{v} \, dV}_{\text{body force}} \qquad \text{(Energy)}$$

$M =$ mass of control volume

$\underline{Q} =$ momentum of control vol.

$E =$ energy of control vol.

$\rho =$ fluid density

$p =$ pressure

$\underline{g} =$ gravity

$e =$ total specific energy

$\underline{\nabla} =$ gradient operator

$\bullet$ = scalar product

Plus suitable <u>equation of state</u>: $p = p(\rho, i)$

Euler equations (3)

- For a compressible fluid, the total specific energy is:

$$e = i + \frac{1}{2}v^2 \qquad (i = \text{ internal specific energy})$$

- Instead of the total energy equation, one can use the internal energy form:

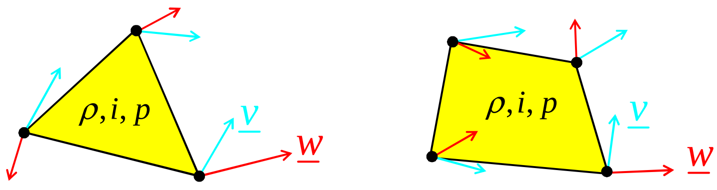
$$\frac{dI}{dt} \equiv \frac{d}{dt}\int_{V(t)} \rho i \; dV = \underbrace{\oint_{S(t)} \rho i(\underline{w} - \underline{v}) \bullet \underline{n} \; dS}_{\text{transport}} - \underbrace{\int_{V(t)} p\nabla \bullet \underline{v} \; dV}_{\text{pressure}} \quad \text{(Internal energy)}$$

$I =$ internal energy of control vol.

11

Finite Element formulation

- Discretize via <u>linear</u> elements with velocities at nodes
- ρ and i (hence p) are <u>uniform</u> over each element



- Integral forms of mass and energy conservation can be used directly, provided $V(t)$ represents the current element volume
- Conservation of momentum is more complex

12

Treatment of momentum equation

- Previous integral statement is unable to furnish enough equations, since the velocity field in each element depends upon *Nd* parameters (*N* being the number of nodes and *d* the space dimension).
- One formulates a variational statement associated with the differential form of the momentum equation expressed in mixed coordinates.
- Final result is principle of virtual power (corresponding to principle of virtual displacements in solid mechanics).

13

Treatment of momentum equation (2)

- The principle of virtual power reads:

$$\int_{V(t)} \delta v_i \rho \frac{\partial v_i}{\partial t} dV = \int_{V(t)} \delta v_i \rho (\underline{w} - \underline{v}) \cdot \underline{\nabla} v_i dV + \int_{V(t)} \delta v_i \rho g_i dV + \int_{V(t)} p \frac{\partial(\delta v_i)}{\partial x_i} dV + \oint_{S(t)} \delta v_i T_i dS$$

v_i are the components of fluid velocity $\underline{v}$

δv_i are arbitrary admissible variations of the fluid velocity

g_i are the components of the acceleration of gravity

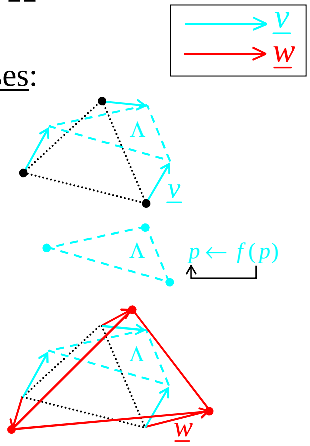
T_i are the component of prescribed boundary loads per unit area

x_i are spatial coordinates (current position of particles in fixed frame)

14

Time integration

Each time increment is split into three phases:



1. Explicit Lagrangian phase: by posing $\underline{w} = \underline{v}$ all transport terms vanish
2. Implicit Lagrangian phase (see details below) whereby the pressure is iteratively refined (this stabilizes the solution)
3. Convective flux phase whereby the transport term contributions are added

- This scheme is not as limpid as the purely Lagrangian one. Second-order accuracy guaranteed only for phase 1.

15

Time integration (2)

0. First compute $\underline{v}^{n+1/2} = \underline{v}^{n-1/2} + \Delta t \underline{a}_n$, where: $\underline{a}_n = \dfrac{\underline{F}_t^{\,old} + \underline{F}_p^{\,n} + \underline{F}_b^{\,n} + \underline{F}_s^{\,n}}{M}$

   $\underline{F}_t^{\,old}$ momentum transport forces at the end of previous step

   $\underline{F}_p^{\,n}$ pressure forces $\quad \underline{F}_b^{\,n}$ body forces $\quad \underline{F}_s^{\,n}$ surface forces

1. Obtain new "Lagrangian" configuration: $\boxed{\underline{x}^L = \underline{x}^n + \Delta t \cdot \underline{v}^{n+1/2}}$

2. Evaluate *L*-volume and *L*-density (mass *M* is constant!):

$$V^L = V^L(\underline{x}^L) \qquad \rho^L = M^n / V^L$$

3. From the internal energy eqn. without the transport term, using the divergence theorem:

$$\frac{d}{dt}(\rho i V) = -(p+q)\oint_S \underline{v} \bullet \underline{n}\, dS$$

pseudo-viscous pressure

16

Time integration (3)

Some pseudo-viscosity is needed to stabilize solution at shock fronts:

$$q' = \begin{cases} \rho[C_Q l^2 (\underline{\nabla} \cdot \underline{v})^2 - C_L la(\underline{\nabla} \cdot \underline{v})] & \text{for } (\underline{\nabla} \cdot \underline{v}) < 0 \\ 0 & \text{for } (\underline{\nabla} \cdot \underline{v}) \geq 0 \end{cases} \qquad q = \min\left(q', \frac{|p|}{2}\right)$$

C_Q, C_L quadratic and linear coefficient a dilatational wave speed

l characteristic length of the element

4. Former expression can be approximated to the first order by:

$$\frac{(\rho i V)^L - (\rho i V)^n}{\Delta t} = -(p+q)^n \frac{V^L - V^n}{\Delta t}$$

5. By noting that $(\rho V)^L = (\rho V)^n = M^n$ we obtain for the internal energy:

$$\boxed{i^L = i^n - (p+q)^n \frac{V^L - V^n}{M^n}}$$

17

Time integration (4)

6. The obtained i^L is just a first guess since <u>the pressure changes over the step and must satisfy the state equation</u>!

7. Obtain *L*-value <u>implicitly</u> by iterating the following expressions (just one or two iterations are usually sufficient):

$$p^L = f(\rho^L, i^L) \qquad f \text{ suitable state equation}$$

$$i^L = i^n - \left(\frac{p^n + p^L}{2} + q^n\right) \frac{V^L - V^n}{M^n}$$

8. Compute true end-of-step configuration: $\boxed{\underline{x}^{n+1} = \underline{x}^n + \Delta t \cdot \underline{w}^{n+1/2}}$

9. New volume: V^{n+1}

18

Time integration (5)

10. Account for mass transport across element boundaries (face by face!) to compute new element mass:

$$M^{n+1} = M^n + \Delta t \cdot \oint_{S^n} \rho^L (\underline{w} - \underline{v})^{n+1/2} \bullet \underline{n}\, dS$$

11. For stabilization reasons, the density ρ_J (depending on face J) is computed as a weighted average of the neighbor elements' densities:

$$\boxed{\rho_J = \frac{1}{2}[(1-\alpha_J)\rho^e + (1+\alpha_J)\rho^{e'}]}$$

$$\alpha_J = \alpha_0 \cdot \text{ sign } (F) \qquad F = \Delta t \cdot \int_{S_J} (\underline{w} - \underline{v}) \bullet \underline{n}\, dS$$

$$0 < \alpha_0 < 1$$

$\alpha_0 = 0$: centered approximation (oscillations!)
$\alpha_0 = 1$: full donor (too diffusive!)

19

Time integration (6)

12. The new element density is obtained as: $\rho_e^{n+1} = \dfrac{M_e^{n+1}}{V_e^{n+1}}$

13. Similar treatment then applied also to energy transport:

$$\frac{(Mi)^{n+1} - (Mi)^n}{\Delta t} = [\sum_J (\rho^L i^L)_J \int_{S_J} (\underline{w} - \underline{v})^{n+1/2} \cdot \underline{n} dS] + \frac{(Mi)^L - (Mi)^n}{\Delta t}$$

$$i^{n+1} = \frac{M^n}{M^{n+1}} i^L + \frac{\Delta t}{M^{n+1}} [\sum_J (\rho^L i^L)_J \int_{S_J} (\underline{w} - \underline{v})^{n+1/2} \cdot \underline{n} dS]$$

14. The final pressure is: $p^{n+1} = f(\rho^{n+1}, i^{n+1})$

15. The nodal forces due to momentum transport (for the next step) are:

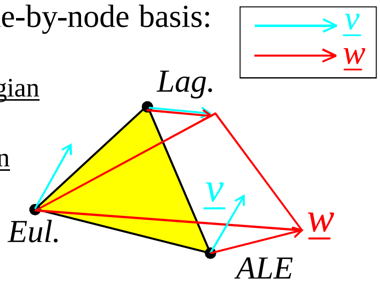
$$F_{tiI}^e = \int_{V^e} N_I \rho^L (\underline{w} - \underline{v})^{n+1/2} \cdot \underline{\nabla} v_i^{n+1/2} dV$$

20

ALE description

The ALE description is a generalization of the Lagrangian and Eulerian descriptions. On a node-by-node basis:

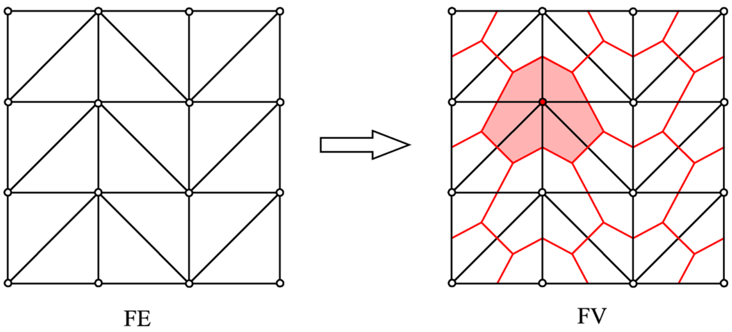
- When $\underline{w} \equiv \underline{v}$ the description is Lagrangian
- When $\underline{w} \equiv \underline{0}$ the description is Eulerian
- Else, the description is ALE



- Since $\underline{w}$ is arbitrary, it must be provided either by the user or via suitable automatic rezoning algorithms.

N-C Finite Volume Formulation

- Node-centered FV fluid model:



- Fully unstructured FV grid is automatically built up by the code from a FE grid of the fluid domain

Euler Equations (NCFV)

- They are written in <u>conservative</u> form : $\frac{\partial \underline{U}}{\partial t} + \underline{\nabla} \cdot \underline{F} = \underline{0}$

  $\underline{U} \doteq \{\rho, \rho\underline{u}, \rho E\}$ (conserved variables)

  $\underline{F}(\underline{U}) \doteq \{\rho\underline{u}, \rho\underline{u}\underline{u} + p\underline{I}, (\rho E + p)\underline{u}\}$ (flux matrix)

- Here $\underline{u}$ is the fluid (particles) velocity
- A suitable <u>equation of state</u> relates the pressure p to the conserved variables
- The <u>weak form</u> of the conservation equations is obtained by <u>integrating</u> them over a generic <u>control volume</u> V (fixed in space) and by applying Green's (divergence) theorem

$$\frac{d}{dt}\int_V \underline{U}\, dV = -\oint_S \underline{F} \cdot \underline{n}\, dS$$

- If the control volume <u>moves</u> in time with an arbitrary velocity $\underline{w}$, then by Reynold's transport theorem

$$\frac{d}{dt}\int_{V(t)} \underline{U}\, dV = \int_{V(t)} \left[\frac{\partial \underline{U}}{\partial t} + \underline{\nabla} \cdot (\underline{U}\underline{w})\right] dV$$

23

Euler Equations (NCFV) - 2

- By Green's theorem in the RHS of the previous equation

$$\frac{d}{dt}\int_{V(t)} \underline{U}\, dV = \int_{V(t)} \frac{\partial \underline{U}}{\partial t} dV + \oint_{S(t)} \underline{U}(\underline{w} \cdot \underline{n})\, dS$$

- The <u>discrete</u> conserved variables are integral means over the generic control volume V_I

$$\underline{U}_I^n \doteq \frac{1}{V_I^n}\int_{V_I^n} \underline{U}^n(\underline{x}, t^n)\, dV$$

- The above equations are integrated in time over an interval $\Delta t^n \doteq t^{n+1} - t^n$ and, after some further manipulations (omitted here for brevity), one obtains the following scheme
- The solution is advanced in time by the <u>first-order</u> explicit scheme (formally: forward Euler)

$$V_I^{n+1}\underline{U}_I^{n+1} = V_I^n \underline{U}_I^n + \Delta t^n \underline{\Phi}_I^*$$

where : $\underline{\Phi}_I^* \doteq \sum_{J \in \Psi(I)} N_{IJ}(\underline{U}_{IJ}^* w_{IJn} - \underline{\varphi}_{IJ}^*)$ (assembled numerical fluxes)

$(\varphi_k^*)_{IJ} \doteq (\underline{F}_k^*)_{IJ} \cdot \underline{n}_{IJ}^*$ (flux matrix projection)

24

$V_I^{n+1}\underline{U}_I^{n+1} = V_I^n\underline{U}_I^n + \Delta t^n \underline{\Phi}_I^*$

Euler Equations (NCFV) - 3

- The calculation of numerical fluxes is done (e.g.) by using Roe's approximate Riemann solver for the Riemann problem defined by each couple of neighboring volumes' physical states.
- The solution is advanced in time by the explicit "first-order" scheme presented above, irrespective of the desired accuracy.
- The obtained accuracy in space and time of the update depends only on the procedure used to compute the numerical fluxes $\underline{\Phi}_I^*$:
  - Choosing $t^* = t^n$ and node-centred values in space gives first-order accuracy in time and in space
  - Choosing $t^* = t^{n+1/2}$ and spatially extrapolated intra-cell boundary values gives second-order accuracy in time and in space

25

Euler Equations (NCFV) - 4

- To obtain second-order accuracy in time and in space, the following predictor-corrector scheme in time is adopted, combined with spatial extrapolation of the conserved variables to intra-cell boundaries (van Leer's MUSCL-like technique)
  1. Compute spatial gradients of conserved variables at intra-cell boundaries, then at cell centres (by "min mod" limitation)
  2. Perform predictor step in time : advance provisionally the conserved variables by half a step by using the nodal gradients
  3. Use again the nodal gradients to extrapolate the variables in space onto the intra-cell boundaries (MUSCL)
  4. Corrector step in time : compute the assembled numerical fluxes according to Roe's scheme, then update the conserved variables

26

NCFV / FE Synchronization

- To make the FE and NCFV formulations compatible, in view of performing FSI simulations by Lagrange multipliers, the two schemes must be properly synchronized:
  1. Shift the FV scheme forward in time by half step, so that conserved variables (and fluid velocity $\underline{u}$) are discretized at the same time instants as the structure velocity $\underline{v}$ (i.e. at half steps)

$$V_I^{n+3/2}\underline{U}_I^{n+3/2} = V_I^{n+1/2}\underline{U}_I^{n+1/2} + \frac{\Delta t^n + \Delta t^{n+1}}{2}\underline{\Phi}_I^*$$

  2. Use formally the same (CD) time integration scheme for both sub-domains, by identifying a suitable additional force term in the fluid sub-domain ("synchronizing" force term due to mass changes)

$$F_{\text{mass}}^{n+1} \doteq \frac{2}{\Delta t^n + \Delta t^{n+1}}(M^{n+3/2} - M^{n+1/2})u^{n+1/2}$$

27

NCFV / FE Synchronization (2)

- The same (CD) time integration scheme can be used in both sub-domains :

$$\text{Equilibrium :} \quad a^{n+1} = \frac{(F_{\text{ext}} - F_{\text{int}} - F_{\text{mass}})^{n+1}}{M}$$

$$\text{Velocity update :} \quad s^{n+3/2} = s^{n+1/2} + \frac{\Delta t^n + \Delta t^{n+1}}{2}a^{n+1}$$

provided one poses :

$$\text{For structure dofs :} \quad M = M_S^{n+1} \qquad s = v \qquad F_{\text{mass}} = 0$$

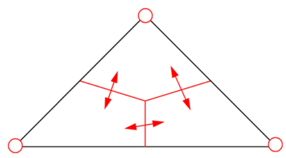
$$\text{For fluid dofs :} \quad M = M_F^{n+3/2} \qquad s = u \qquad F_{\text{mass}} = \frac{2}{\Delta t^n + \Delta t^{n+1}}\left(M^{n+3/2} - M^{n+1/2}\right)u^{n+1/2}$$

- Then, the same Lagrange multipliers method can be used to impose essential boundary conditions (constraints on velocities) in both sub-domains (and in particular for FSI)

28

N-C Finite Volume Peculiarities

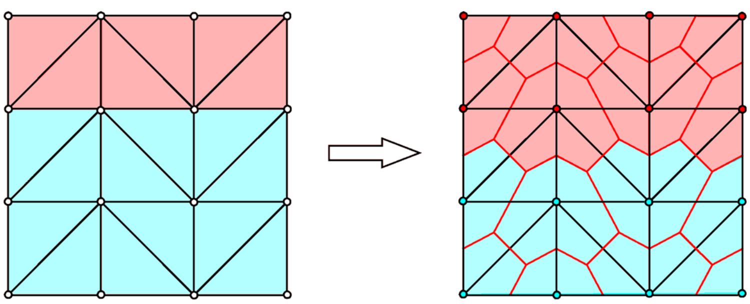
- All quantities (velocities and pressures) are discretized at nodes
- Physical status of FV depends only upon its volume, not on its shape (OK for fluid)
- However, geometry is still used to compute volume and fluxes
- Transport (among FVs, i.e. nodes) is computed internally to each FE : no need to know neighbor FEs



29

Initial Conditions FE / NCFV

- Equivalence between FE and N-C FV initial conditions is not straightforward:

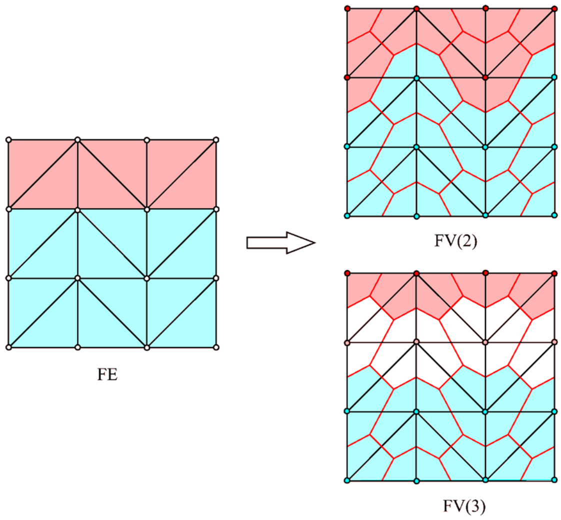


FE FV(1)

30

Initial Conditions FE / NCFV (2)

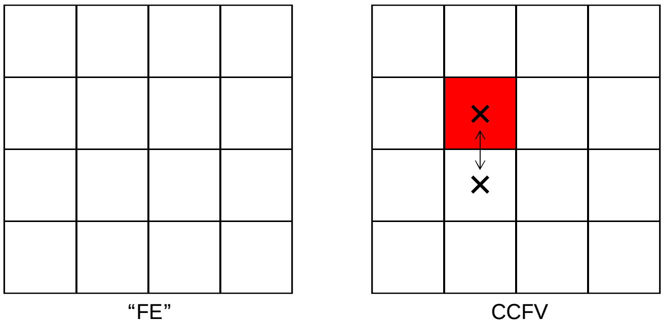
- Some possible remedies:



31

C-C Finite Volume Formulation

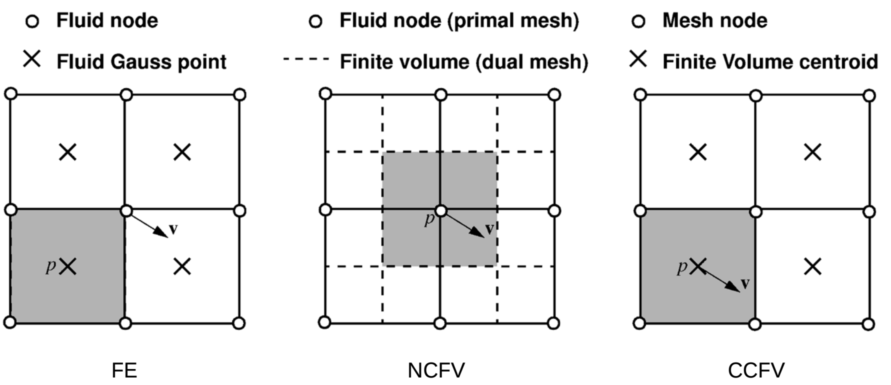
- Cell-centered FV fluid model (CEA : Galon, Beccantini):



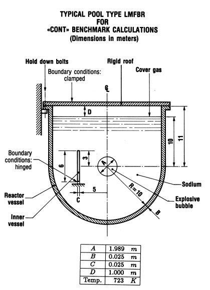
- FV mesh coincides with classical FE mesh
- All quantities are discretized at "cell" (FV) centres
- Initial conditions are set like in FE

32

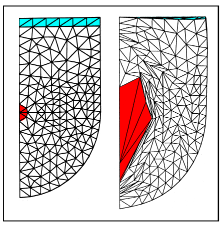
Comparing FE, NCFV and CCFV meshes



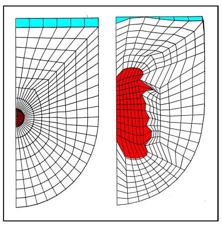
Mesh rezoning - motivations



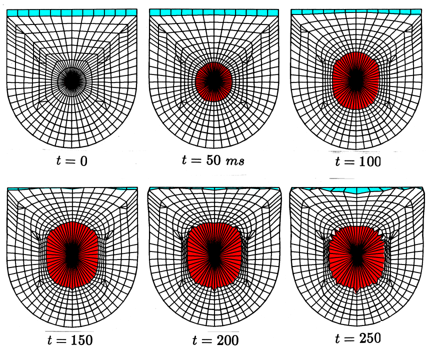
- Rezoning (ALE) is necessary because Lagrangian solutions suffer from excessive distortions and entanglement:



Lagrangian, triangles



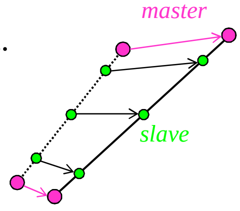
ALE, manual rezoning, quadrangles



ALE, automatic rezoning (Giuliani's)

Mesh rezoning - types

- Rezoning (in this context): motion of nodes belonging to an ALE mesh, at constant mesh topology.
- Do not confuse with remeshing, adaptivity etc.
- Kinematic:
  - Slave node follows master node
  - Nodes along a line are piloted by master end-points
  - Nodes inside a triangle, quadrangle, etc. move homeomorphically



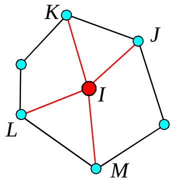
- Geometric:
  - Mean value algorithms
  - Giuliani's method: minimize triangles/tetrahedra distortion
- Mechanic:
  - Solve elasticity eqs. for dual mesh (ALE grid) considered as a solid (**continuum**, bar assembly, etc.). **Explicit** formulation to save CPU!

35

Mean-based rezoning

- Simple: optimal position of a node is the mean of its neighbors:

$$\underline{x}_I^r = \frac{1}{n}\sum_{P=1}^{n}\underline{x}_P$$



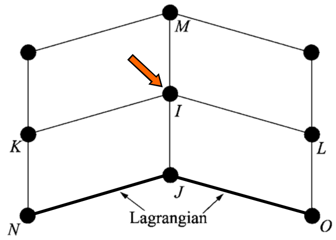
- Alternative: use displacement instead of position
- Drawbacks:
  - Rapidly tends to produce uniform mesh (which can be unwanted in some cases)
  - Fails in some special configurations (local mesh "curvature")

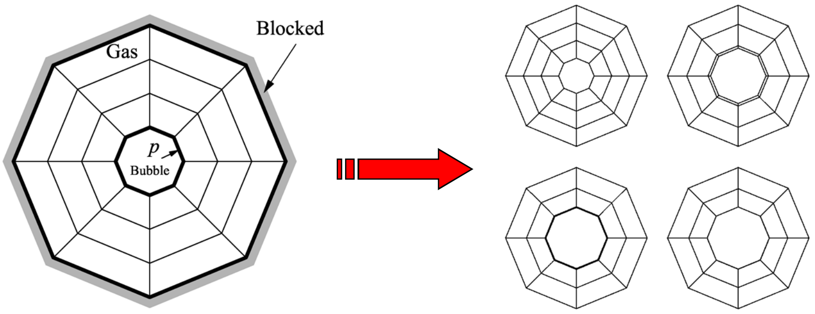
36

Mean-based rezoning (2)

- Pathological example:

$$\underline{x}_I^r = \frac{1}{4}(\underline{x}_J + \underline{x}_K + \underline{x}_L + \underline{x}_M)$$

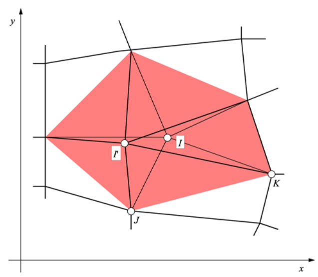




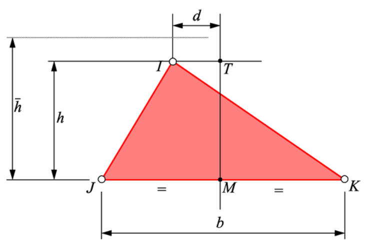
37

Giuliani’s automatic rezoning

- For each ALE node, identify an <u>influence domain</u> made of neighbor triangles (or tetrahedra in 3D)



- Measure “<u>shear</u>” and “<u>stretch</u>” of each triangle in the domain



shear = $f(d)$
stretch = $g(h)$

$\bar{h}$ is the mean height of triangles in the influence domain

38

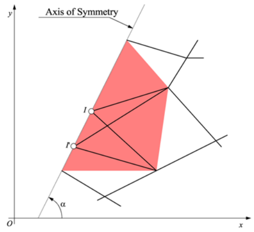
Giuliani's automatic rezoning (2)

- Minimize a function of total shear and stretch over the node's influence domain:

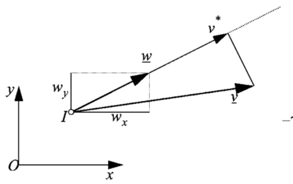
$$E = \sum_{i=1}^{N} \left(\frac{h_i - \bar{h}}{\bar{h}}\right)^2 + \sum_{i=1}^{N} \left(\frac{2d_i}{\bar{b}}\right)^2 = \text{minimum}$$

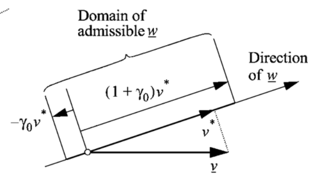
$\bar{h}$ *mean height,* $\bar{b}$ *mean base*

- Special care for nodes subjected to b.c.s:



- It is essential to <u>constrain</u> the resulting grid velocity:

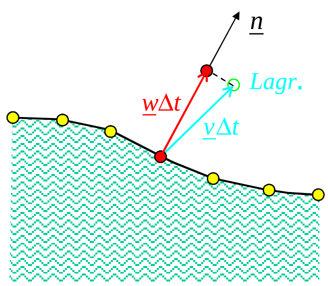




Free surface modeling

- <u>Lagrangian</u> modeling is accurate but can lead to excessive mesh distortions
- Alternative: use <u>Lagrangian</u> description in the <u>normal</u> direction, <u>ALE</u> <u>tangentially</u> (slip)

  - Project along the normal:



  - Alternative: add correction to keep nodes almost equidistant

Numerical representation of interfaces between (unmiscible) fluid components

Two alternative approaches:

1. **Component mixing not allowed**
   - ALE simulation and Lagrangian interface between the components
     - ☑ Exact interface location
     - ☒ Dramatic lack of robustness for complex interface motion
   - Interface reconstruction and cell division (*Volume-Of-Fluid*, *Level-Sets*…)
     - ☑ Accurate interface location
     - ☑ Handling of complex interface motion
     - ☒ Need for specific algorithms and operators to describe the interface
     - ☒ Difficult and time consuming geometric operations within a cell (especially in 3D)
2. **Artificial component mixing allowed**
   - With interface reconstruction (*Front Tracking*)
     - ☑ Accurate interface location
     - ☒ Need for a component mixing model
     - ☒ Need for specific algorithms and operators to describe the interface
   - Without interface reconstruction
     - ☑ No additional component to classical software
     - ☑ Very robust
     - ☒ Need for a **component mixing model**
     - ☒ Risk for **numerical diffusion of concentrations** and lack of accuracy for interface location and motion

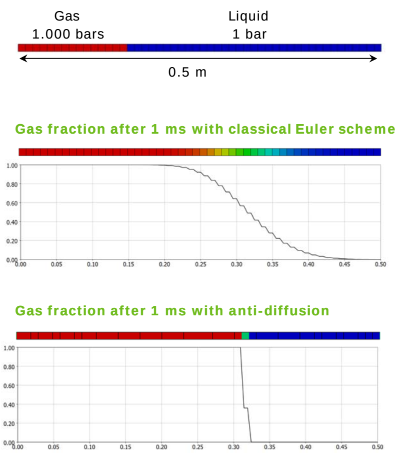
Chosen for **simplicity** and **robustness**
Component mixing model => **isobaric closure**
Numerical diffusion => **VOFIRE scheme**

41

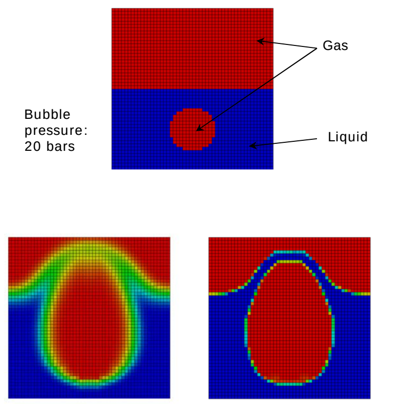
VOFIRE anti-diffusive scheme

[Despres, Lagoutière, Kokh, …]: "***Downwind*** *Scheme with Constraints*"

1D Liquid-Gas shock tube



2D Gas bubble expansion



Gas fraction after 10 ms with classical Euler scheme

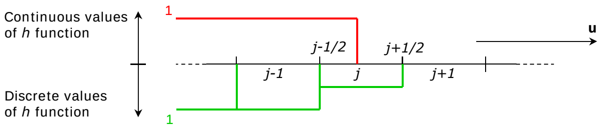
Gas fraction after 10 ms with anti-diffusion

- Introduced in EUROPLEXUS by V. Faucher (CEA)

42

Anti-dissipation – Scheme for structured meshes (1/2)

Model problem: 1D-advection of Heavyside function h in constant velocity field $\mathbf{u}$

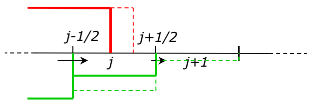


Question: which value $h_{j+1/2}$ to choose for h on cell boundary $j+1/2$ to build function flux ?

Limit cases:

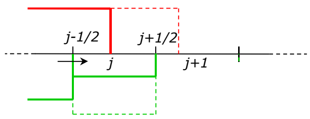
1. **Upwind**: $h_{j+1/2} = h_j$

   Flux is non-zero before the step propagates to cell boundary $j+1/2$
   $\Rightarrow$ **Dissipation**



2. **Downwind**: $h_{j+1/2} = h_{j+1}$

   Flux is always zero $\Rightarrow$ **No dissipation**
   Unstable increase of the discrete value in cell j when the step propagates beyond cell boundary $j+1/2$



Anti-dissipation – Scheme for structured meshes (2/2)

Key Concept [Despres, Lagoutière, Kokh, Dellacherie...]: *Downwind Scheme with Constraints*

Constraints:

- **Consistency** for the flux: $\min\left(h_j, h_{j+1}\right) \le h_{j+1/2} \le \max\left(h_j, h_{j+1}\right)$ [C1]
- **Stability** for the advection scheme: $\min\left(h_{j-1}, h_j\right) \le h_j^* \le \max\left(h_{j-1}, h_j\right)$ [C2]

$$[C1] \Leftrightarrow \; m_{j+1/2} \le h_{j+1/2} \le M_{j+1/2}$$

$$[C2] \Leftrightarrow \; m_{j-1/2} \le h_j + \mathbf{u}\frac{\Delta t}{\Delta x}\left(h_{j-1/2} - h_{j+1/2}\right) \le M_{j-1/2} \;\Leftarrow\; \begin{cases} m_{j-1/2} \le h_j + \mathbf{u}\dfrac{\Delta t}{\Delta x}\left(m_{j-1/2} - h_{j+1/2}\right) \\ h_j + \mathbf{u}\dfrac{\Delta t}{\Delta x}\left(M_{j-1/2} - h_{j+1/2}\right) \le M_{j-1/2} \end{cases}$$

$\downarrow$

Trust intervals for flux $h_{j+1/2}$

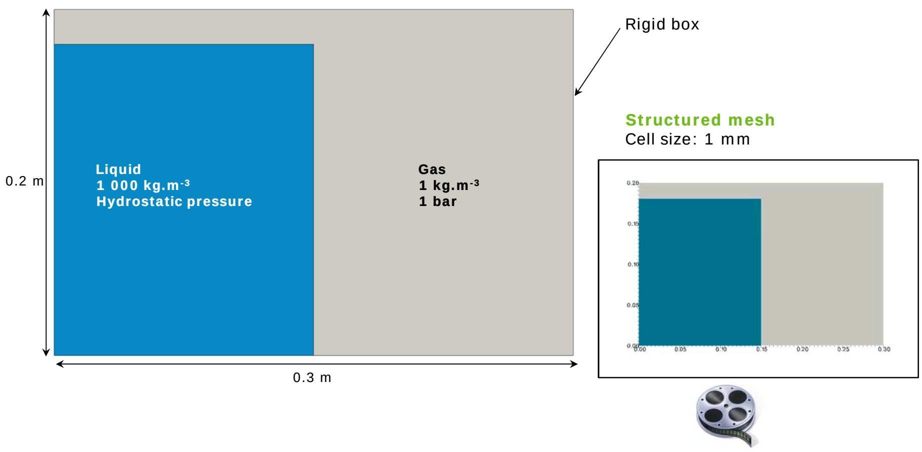
$\downarrow$

Choice of the **value closest to downwind** within **intersection of trust intervals**

Extension to 2D / 3D structured meshes: straightforward using **Direction Splitting**

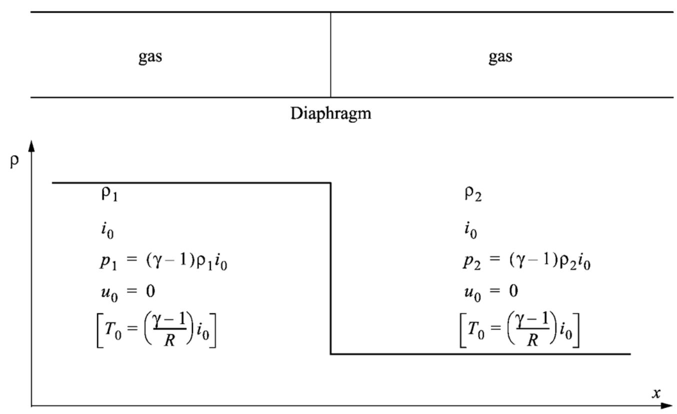
Sloshing under gravity loading

(Courtesy of CEA)



Exercise 1 – Shock tube

- Initial conditions:



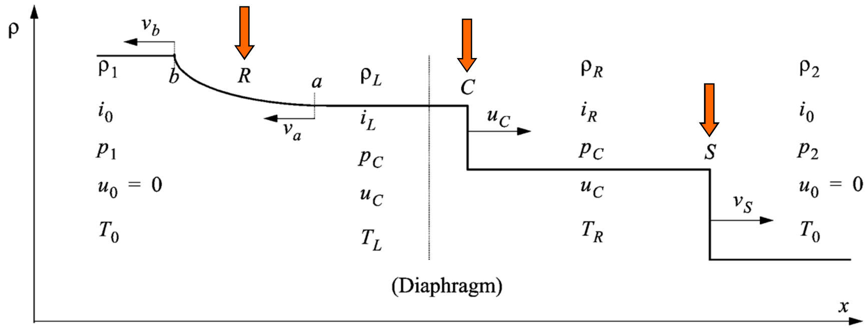
- Equation of state (polytropic gas): $p = (\gamma-1)\rho i$

$\gamma \doteq c_P / c_V$

$c_P \doteq$ specific heat at constant pressure
$c_V \doteq$ specific heat at constant volume

Shock tube (2)

- Analytical solution (self-similar):



- Shock S moves to the right at speed v_S
- Contact discontinuity C (particle initially at diaphragm) moves to the right at speed u_C
- Rarefaction wave $R(a,b)$ moves to the left at speeds v_a, v_b

47

Shock tube (3)

Suggested variations:

- Study influence of pseudo-viscosity and upwinding on Eulerian solution
- Obtain and discuss Lagrangian solution(s). Hint: try different amounts of pseudo-viscosity
- Obtain and discuss ALE solution(s). Compare automatic rezoning and mean-based rezoning
- Solve shock tube problem with Finite Volumes

48

Shock tube (4)

- Assumed parameter values:

ρ_1	1.22
ρ_2	0.1237
γ	1.269
p_1	1.0×10^6
p_2	1.013×10^5
$i_1 = i_2$	3.046×10^6

$= i_0$

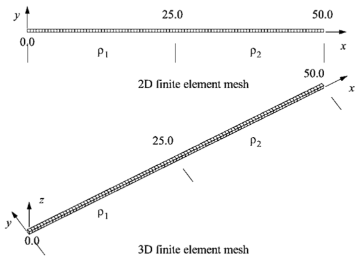
- Analytical solution:

λ	9.863	u_C	925.4
P	2.888	v_S	1672.
p_C	2.927×10^5	v_a	30.12
ρ_R	0.2771	v_b	$-1020.$
ρ_L	0.4635	c_0	1020.
i_R	3.928×10^6	c_a	895.2
i_L	2.348×10^6	c_b	1020.

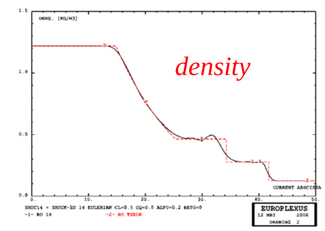
$$\lambda \doteq \rho_1 / \rho_2 = p_1 / p_2$$
$$P \doteq p_C / p_2$$

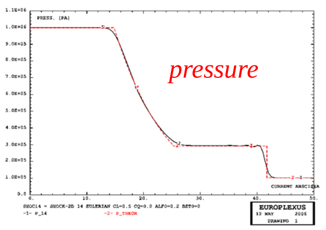
Shock tube (5)

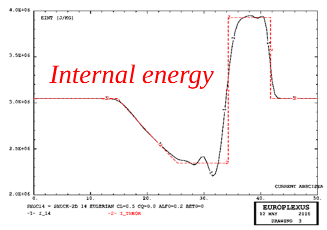
- Discretization:



- “Optimal” numerical solution (Eulerian):
  - ➢ $C_L = 0.5$, $C_Q = 0$
  - ➢ $\alpha_0 = 0.2$, $\beta_0 = 0$







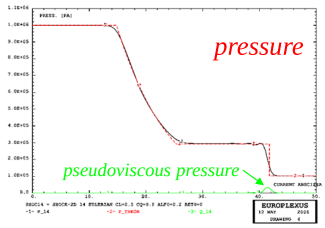
Shock tube (6)

- Influence of pseudoviscosity q' and upwinding:

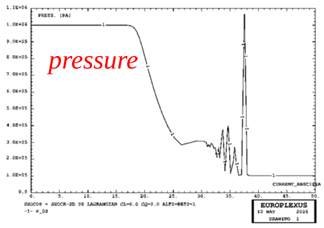
$$q' = \begin{cases} \rho[C_Q l^2(\nabla \cdot \underset{\sim}{v})^2 - C_L l a(\nabla \cdot \underset{\sim}{v})] & \text{for } \nabla \cdot \underset{\sim}{v} < 0 \\ 0 & \text{for } \nabla \cdot \underset{\sim}{v} \geq 0 \end{cases}$$

l: characteristic length, a: wave speed

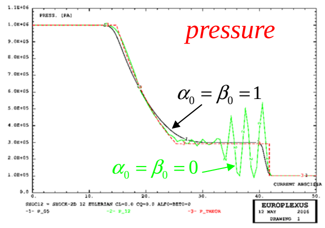
p.v. pressure in optimal Eulerian sol.



Lagrangian solution needs some p.v. to be stable



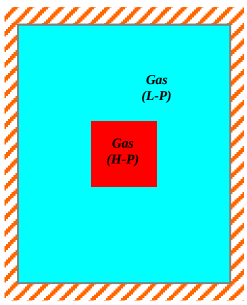
Eulerian solution needs some upwinding to be stable



Exercise 2 – Explosion in air-filled rigid tank

This is a purely fluid problem because the tank is rigid and therefore does not need to be modeled. There is only one type of fluid (single-phase, single-component). This considerably simplifies the modeling, since the bubble surface can be treated as ALE.

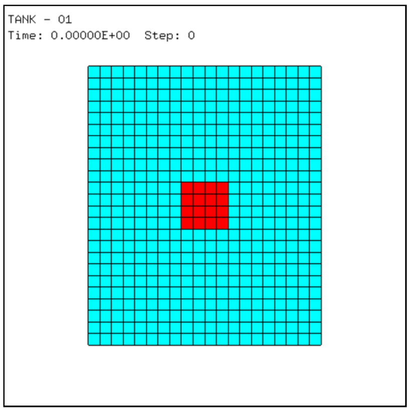
- Try out Lagrangian solution
- Try out Eulerian solution
- Try out ALE solution
- …



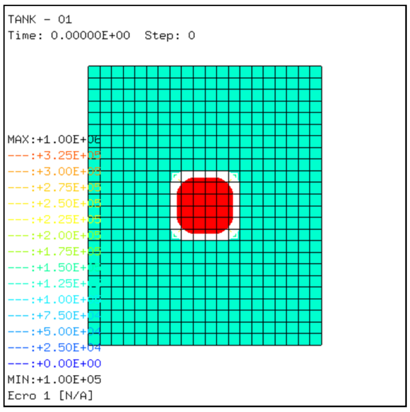
52

Exercise 2 – Explosion in air-filled rigid tank (2)

- Lagrangian solution (TANK01):



Mesh evolution

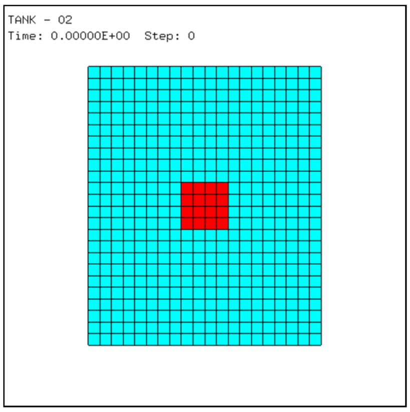


Fluid pressures

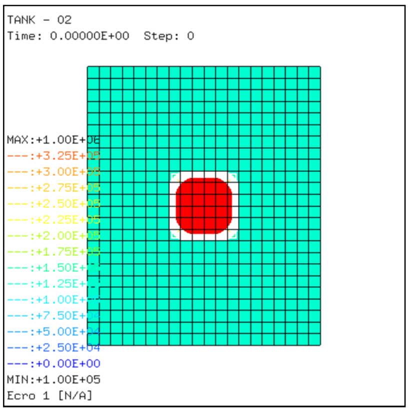
53

Exercise 2 – Explosion in air-filled rigid tank (3)

- Eulerian solution (TANK02):



Mesh evolution

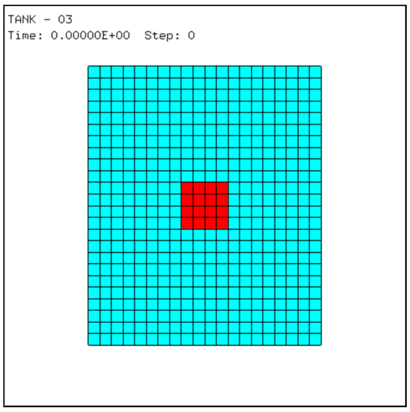


Fluid pressures

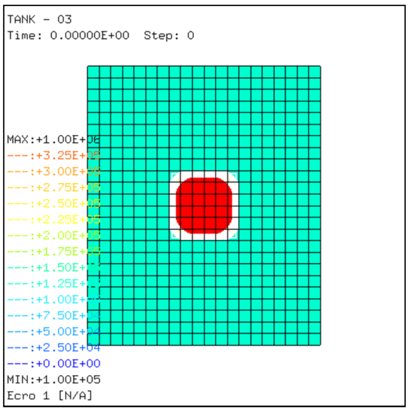
54

Exercise 2 – Explosion in air-filled rigid tank (4)

- ALE solution (with Lagrangian bubble interface) (TANK03):



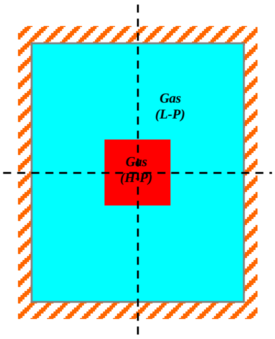
Mesh evolution

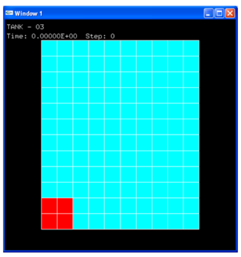


Fluid pressures

Exercise 2 – Explosion in air-filled rigid tank (5)

Input file for ALE Solution (TANK03):



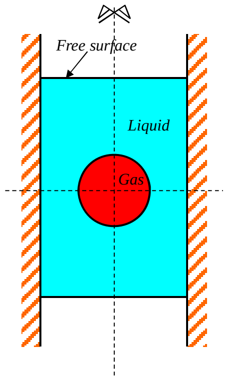


```
TANK - 03
*-----------------------------------------------------------------------
 ECHO
!CONV win
CAST mesh
*----------------------------------------------------------Problem type
DPLA ALE
*----------------------------------------------------------Dimensioning
DIME
 PT2L 143 FL24 145
 NALE 150 NBLE 150
TERM
*--------------------------------------------------------------Geometry
GEOM FL24 flui TERM
*-----------------------------------------------------------Grid motion
GRIL LAGR LECT lag  TERM
     AUTO AUTR
*---------------------------------------------------------Material data
MATE FLUT RO 10  EINT 2.5E5 GAMM 1.4 PB 0 ITER 1 ALF0 1 BET0 1 KINT 1
          AHGF 0 CL 0.5 CQ 2.56 PMIN 0 NUM 1 LECT bull TERM
     FLUT RO 1   EINT 2.5E5 GAMM 1.4 PB 0 ITER 1 ALF0 1 BET0 1 KINT 1
          AHGF 0 CL 0.5 CQ 2.56 PMIN 0 NUM 1 LECT gaz  TERM
*---------------------------------------------------Boundary conditions
LINK COUP
     BLOQ 1 LECT blox TERM
     BLOQ 2 LECT bloy TERM
     CONT SPLA NX 1 NY 0 LECT symx TERM
     CONT SPLA NX 0 NY 1 LECT symy TERM
*---------------------------------------------------------------Outputs
ECRI COOR DEPL VITE CONT ECRO TFRE 1.E-3
     FICH ALIC TEMP FREQ 1
               POIN LECT p4 p6 TERM
               ELEM LECT e1 e2 TERM
*---------------------------------------------------------------Options
OPTI NOTE
     CSTA 0.8
     LOG 1
     REZO GAM0 0.5
*--------------------------------------------------Transient calculation
CALCUL TINI 0 TEND 10.E-3
 . . .
```

Exercise 3 – Bubble expansion in a Tube

This is a also a <u>purely fluid</u> problem because the tank is <u>rigid</u> and therefore does not need to be modeled. However, unlike the previous example, there are <u>two different fluids</u> interacting with each other. The tube wall is vertical so the BC can be trivially imposed as a blockage in the horizontal direction.

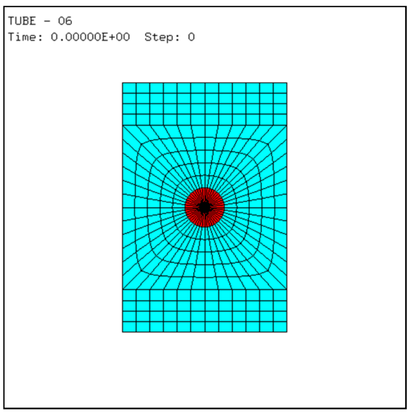
- Try out fully Lagrangian solution
- Try out ALE solution with Lagrangian bubble surface (single-component fluid material)
- Try out fully ALE solution with ALE bubble surface (multi-phase multi-component fluid material)



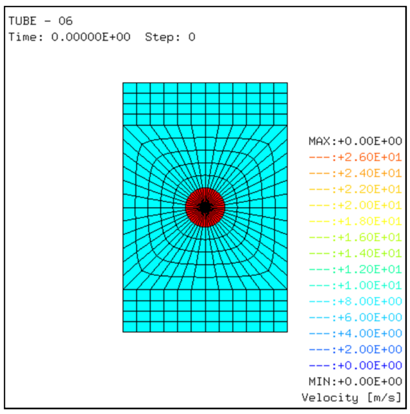
57

Exercise 3 – Bubble expansion in a Tube (2)

- Lagrangian solution (TUBE06):



Mesh

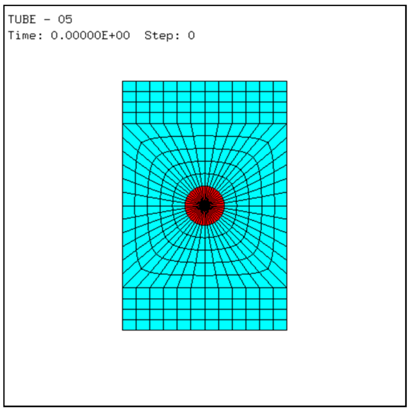


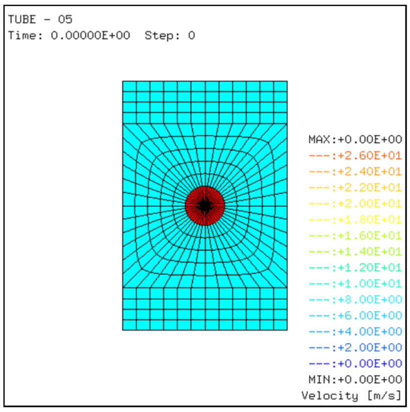
Fluid Velocities

58

Exercise 3 – Bubble expansion in a Tube (3)

- ALE solution with Lagrangian bubble surface (single-component fluid material) (TUBE05):



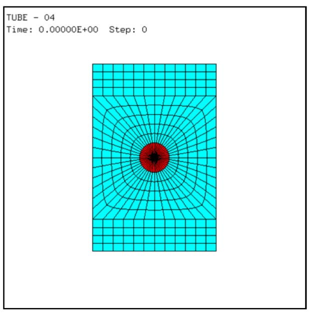


Mesh *Fluid Velocities*

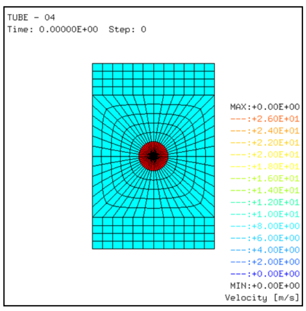
59

Exercise 3 – Bubble expansion in a Tube (4)

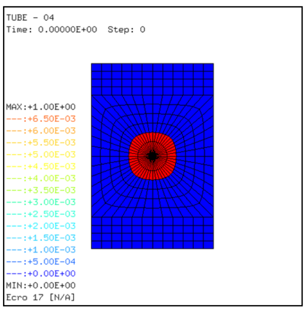
- ALE solution with ALE bubble surface (multi-phase multi-component fluid material) (TUBE04):



Mesh



Fluid Velocities

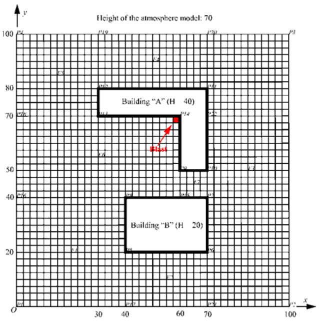


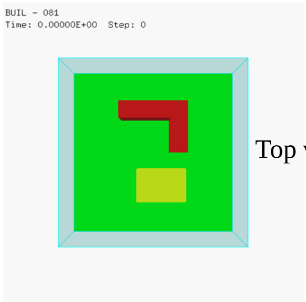
Bubble Mass Fraction

60

Exercise 4 – External blast on two buildings

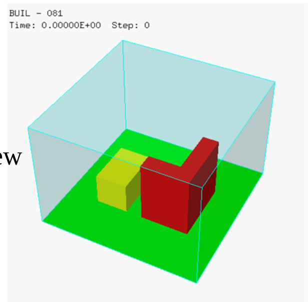
Geometry

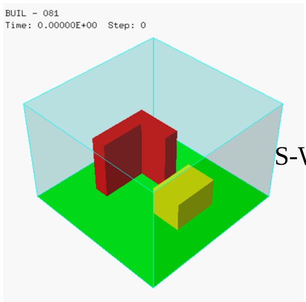




Top view

N-E view





S-W view

61

Exercise 4 – External blast on two buildings (2)

Input file for EULE Solution (BUIL08):

```
BUIL - 08
$
 ECHO
$VERI
!CONV win
CAST MESH
TRID EULE
$
DIME
 PT3L 47029
 CUBE 42496 CL3D 6080 ZONE 2
 BLOQ 10000
TERM
$
GEOM
  CUBE fluid
  CL3D absor
TERM
*
COMP COUL roug LECT explo TERM
          turq LECT air   TERM
          rose LECT absor TERM
*
MATE
$ high-pressure perfect gas (explosive bubble)
     GAZP RO 1000 GAMM 1.4 PINI 100000.E5
          PREF 1.E5 LECT explo TERM
$ air
     GAZP RO 1 GAMM 1.4 PINI 1.E5
          PREF 1.E5 LECT air TERM
$ absorbing
     IMPE ABSO RO 0 C 0
          LECT absor TERM
$
LIAI FREQ 1
     BLOQ 3 LECT bloz TERM
     FSR LECT fsrn TERM
$
ECRI VITE ECRO TFRE 100.E-3
     FICH ALIC TFRE 1.E-3
$
OPTI NOTEST
 csta 0.5e0
 log 1
CALCUL TINI 0 TEND 0.1E0
*=================================================================
FIN
```

62

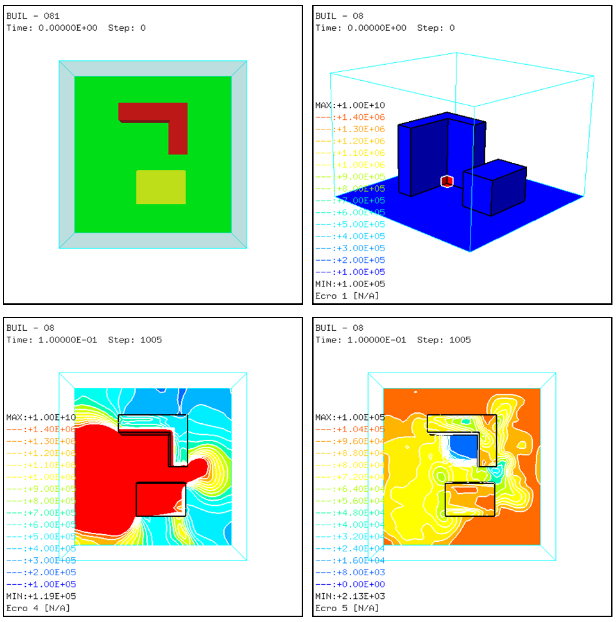
Exercise 4 – External blast on two buildings (3)

Geometry

Pressure

Max pressure

Min pressure



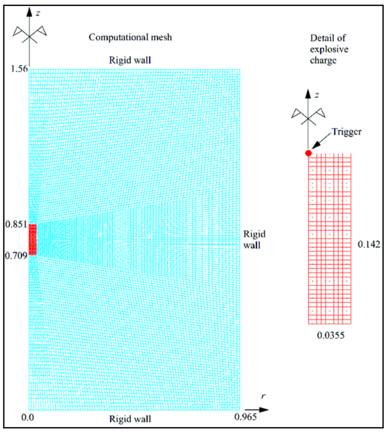
Exercise 5 – Confined detonation of solid TNT charge

Geometry:

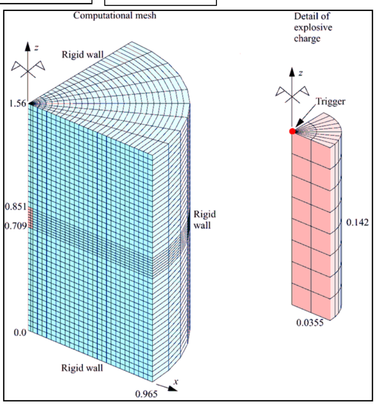
Jones-Wilkins-Lee

$$p(\rho,i)=A\left(1-\frac{\omega}{R_1V}\right)e^{-R_1V}+B\left(1-\frac{\omega}{R_2V}\right)e^{-R_2V}+\omega\rho i$$

$$\boxed{V=\rho_0/\rho} \qquad \boxed{\omega=\gamma-1}$$



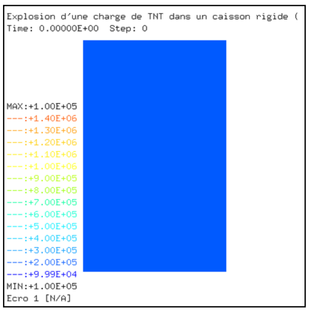
2D axisymmetric model



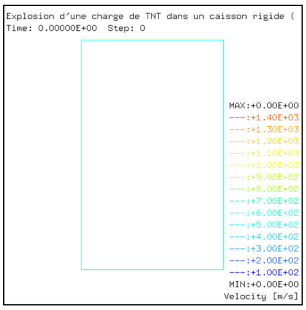
3D axisymmetric model

Exercise 5 – Confined detonation of solid TNT charge (2)

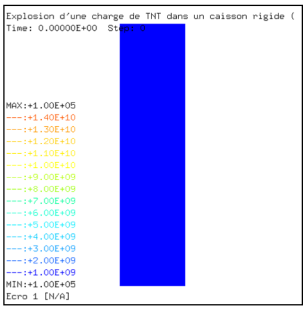
2D axisymmetric solution:



Pressure



Velocity

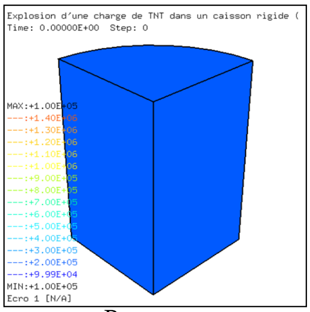


Charge pressure

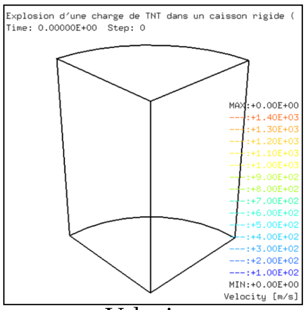
65

Exercise 5 – Confined detonation of solid TNT charge (3)

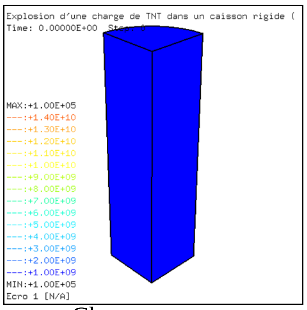
3D axisymmetric solution:



Pressure



Velocity

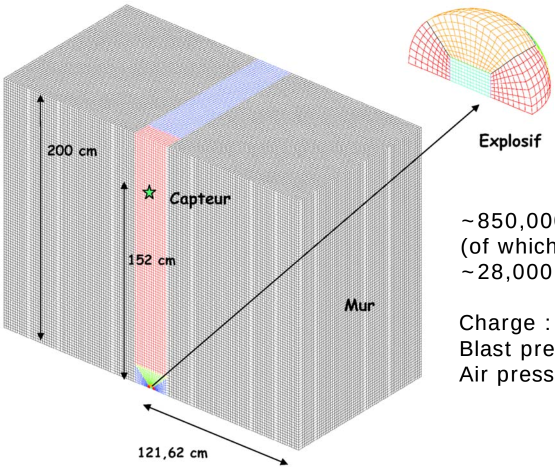


Charge pressure

66

CCFV modeling of spherical wave reflection

(Courtesy of CEA : P. Galon, A. Beccantini)



~850,000 CCFV cubes
(of which 3,000 in the explosive charge)
~28,000 absorbing elements (boundary)

Charge : solid C4
Blast pressure ~104,000 bar
Air pressure 1 bar67

- Strong shock : high-explosive blast wave

67

CCFV results

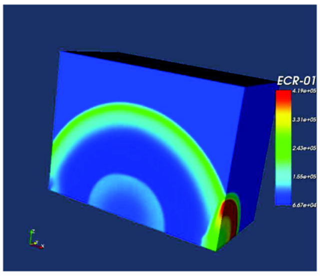


Figure 21 : Champ de pression à T = 1.5 (gaz JWL)

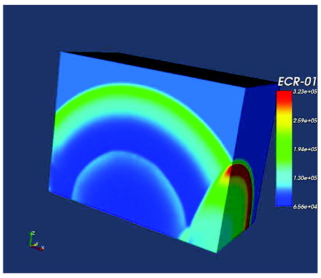


Figure 22 : Champ de pression T = 2 ms (gaz JWL)

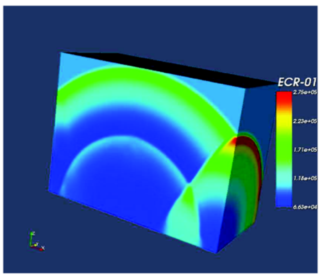


Figure 23 : Champ de pression à T = 3 ms (gaz JWL)

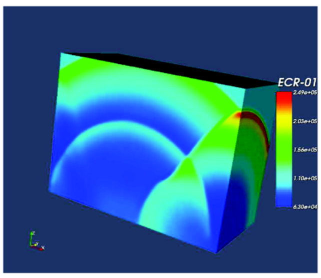
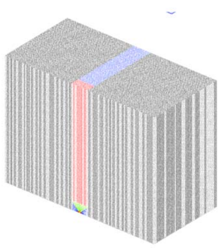
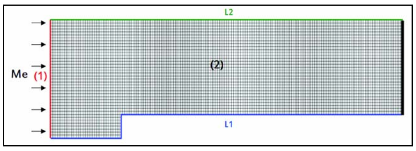


Figure 24 : Champ de Pression T = 3.5 ms (gaz JWL)

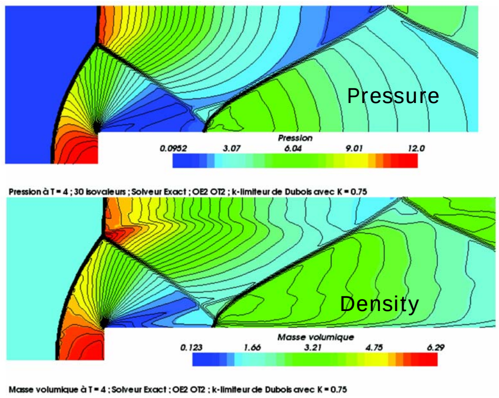


68

CCFV modeling of Mach 3 Flow over Step (Woodward-Colella test) Courtesy of CEA



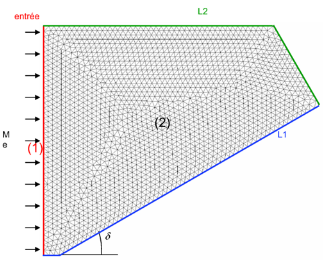
~27,000 CCFV quadrangles



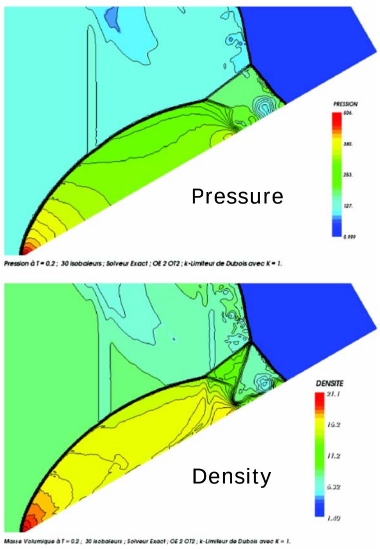
69

CCFV modeling of Shock Wave Reflection

(Courtesy of CEA)

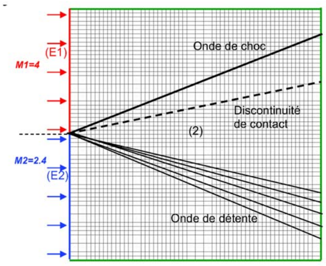


~100,000 CCFV triangles (irregular) for the solutions presented next

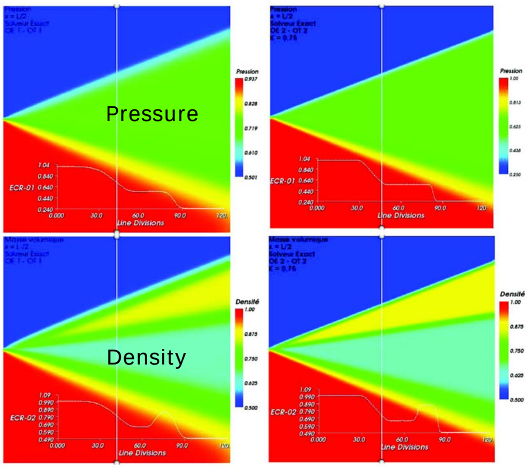


70

CCFV modeling of Interaction between Supersonic Jets (Courtesy of CEA)



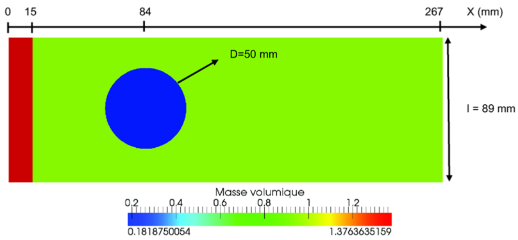
~15,000 CCFV quadrangles



1st order in space and time

2nd order in space and time

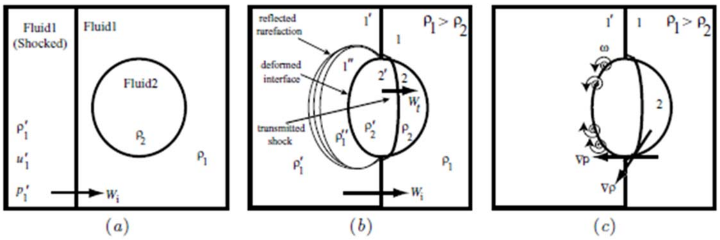
CCFV modeling of Shock-Bubble Interaction (Richtmyer-Meshkov Instability)



Shock wave interacting with a light gas bubble

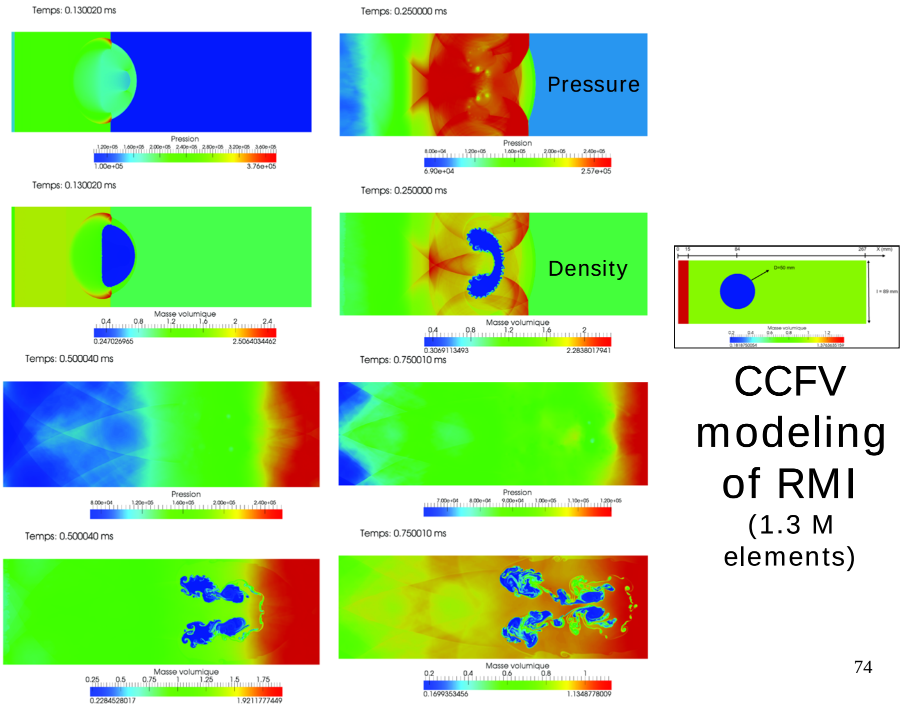
Example of Richtmyer-Meshkov Instability

When the shock wave impacts the bubble interface, some **vorticity** is generated near the interface with a production rate proportional to the vector product between the pressure gradient of the shock wave and the density gradient at the interface: $\sim \nabla p \wedge \nabla \rho / \rho^2$



Courtesy P. Galon, A. Beccantini (CEA)

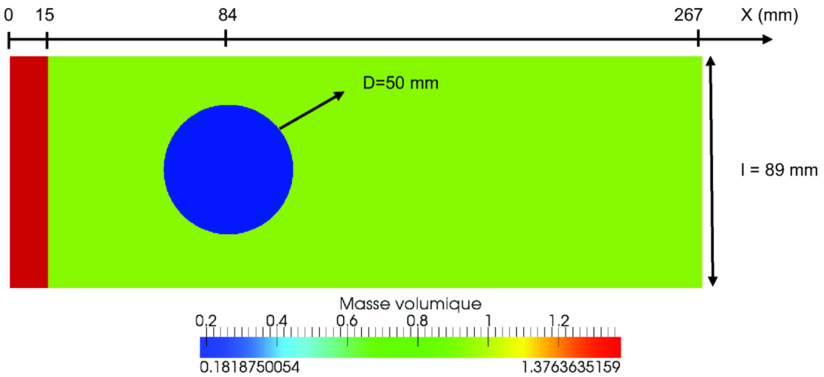
73



CCFV modeling of RMI (1.3 M elements)

74

Exercise 6 – Richtmyer-Meshkov instability

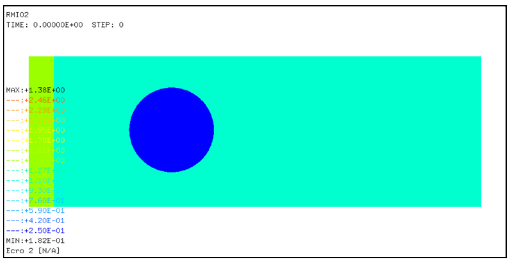


Solution with CCFV (1.3 Million elements)

75

Exercise 6 – Richtmyer-Meshkov instability (2)

Pressure



Density

76

Shock / bubble (He) interaction in air – M = 1.22

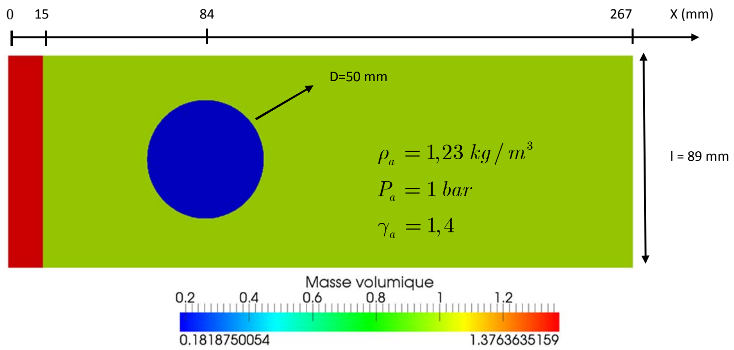
Courtesy P. Galon, A. Beccantini (CEA)

$\rho_a = 1{,}686\ kg/m^3$ $\rho_b = 0{,}2546\ kg/m^3$

$P_a = 1{,}59\ bar$ $P_b = 1\ bar$

$\gamma_a = 1{,}4$ $\gamma_a = 1{,}648$

$u_x = 114{,}5\ m/s$



- J.F Haas and B. Sturtevant. Interaction of weak shock waves with cylindrical and spherical gas inhomogeneities. J. Flui Mech. (1987), vol 181, pp 41-76.
- J.F Haas. Interaction of weak shock waves and discrete gas inhomogeneities, PHD Thesis, California Institute of Technology, 1984.

77

Experiment at t=0.032 ms
Calculation (density) at t=0.030 ms

Expriment at t=0.052 ms
Calculation (density) at t=0.050 ms

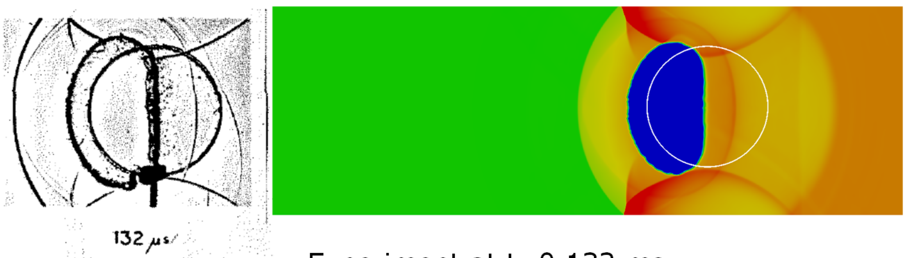
78

Experiment at t=0.062 ms
Calculation (density) at t=0.060 ms

Experiment at t=0.082 ms
Calculation (density) at t=0.080 ms

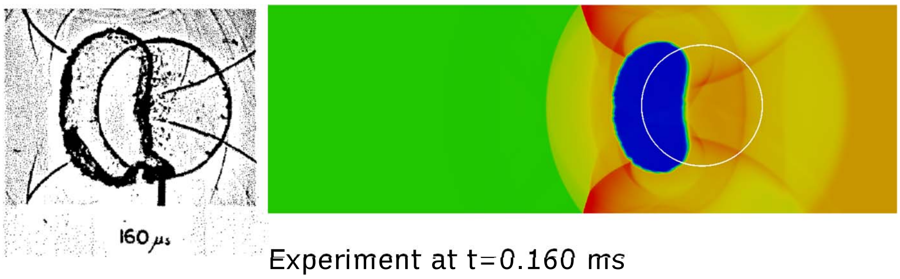
79

Experiment at t=0.102 ms
Calculation (density) at t=0.100 ms

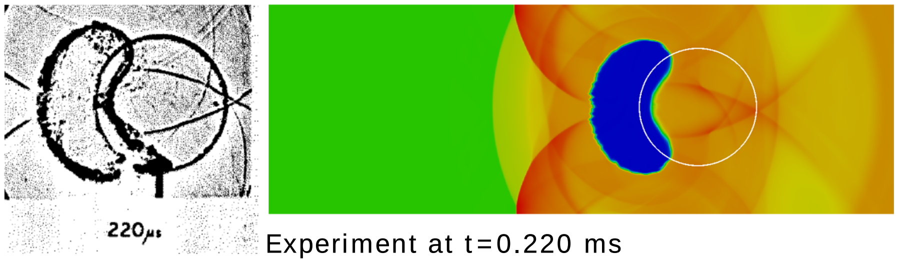


Experiment at t=0.132 ms
Calculation (density) at t=0.140 ms

80



Experiment at t=0.160 ms
Calculation (density) at t=0.170 ms



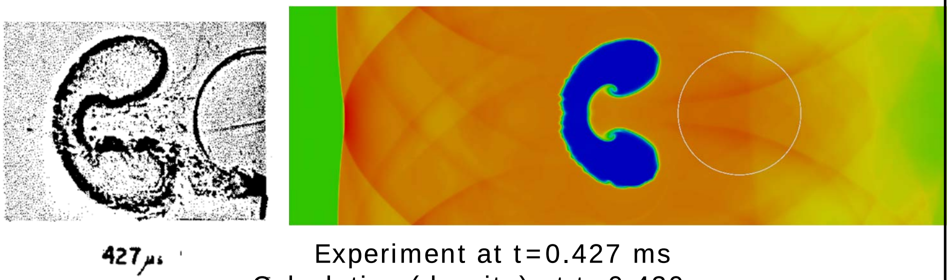
Experiment at t=0.220 ms
Calculation (density) at t=0.230 ms

81

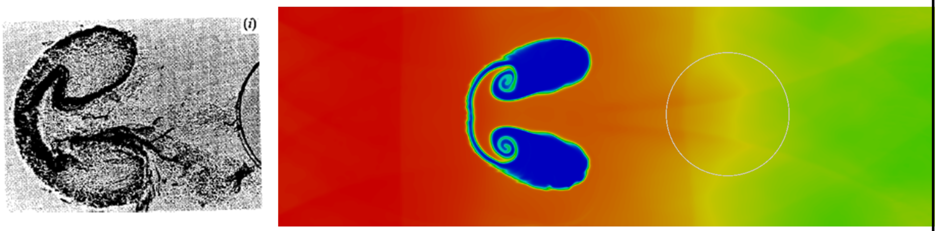
Experiment at t=0.225 ms
Calculation (density) at t=0.240 ms

Experiment at t=0.245 ms
Calculation (density) at t=0.250 ms

82

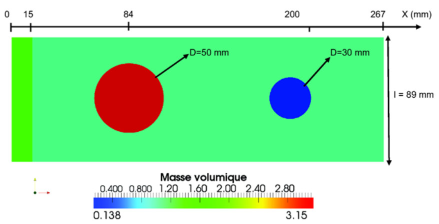


Experiment at t=0.427 ms
Calculation (density) at t=0.430 ms



Experiment at t=0.674 ms
Calculation (density) at t=0.680 ms

CCFV modeling of RMI (2 bubbles)



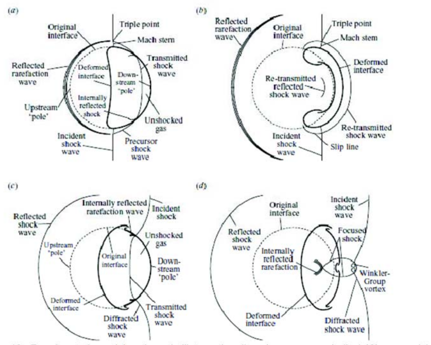
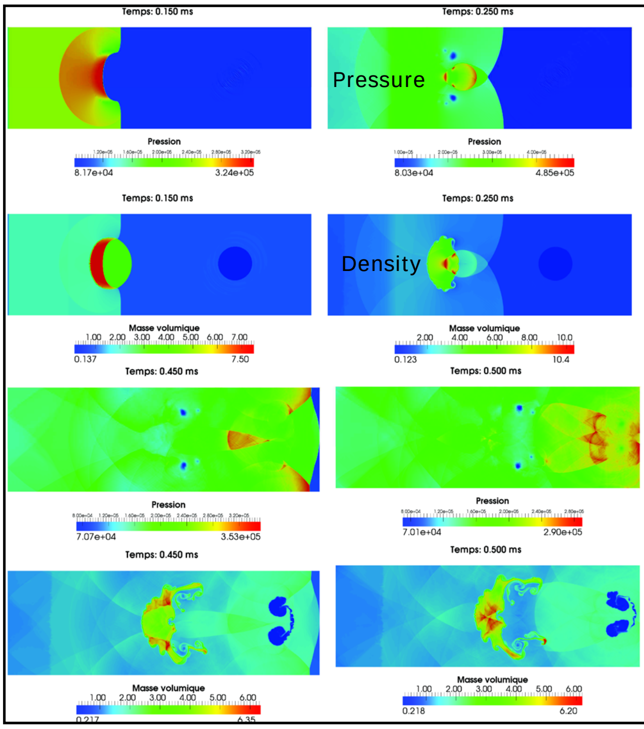
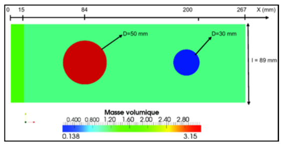


Figure 40 : Représentation schématique de l'interaction d'un choc avec une bulle (référence xxix)

Qualitative physical behavior

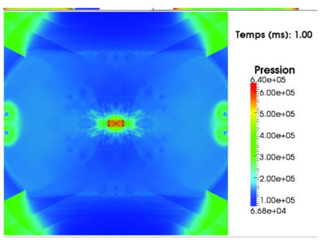




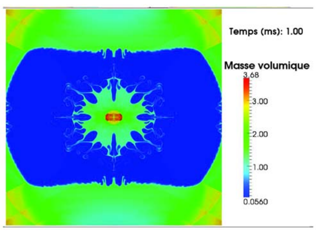
CCFV modeling of RMI (2 bubbles)

(1.3 M elements)

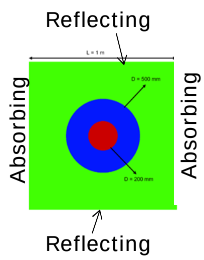
85

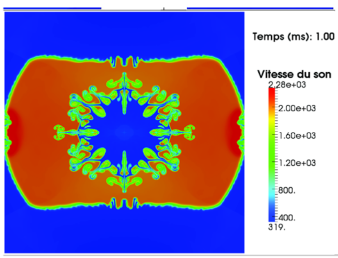


Pressure

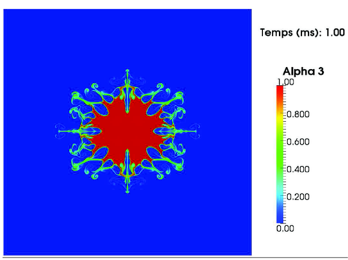


Density





Sound speed



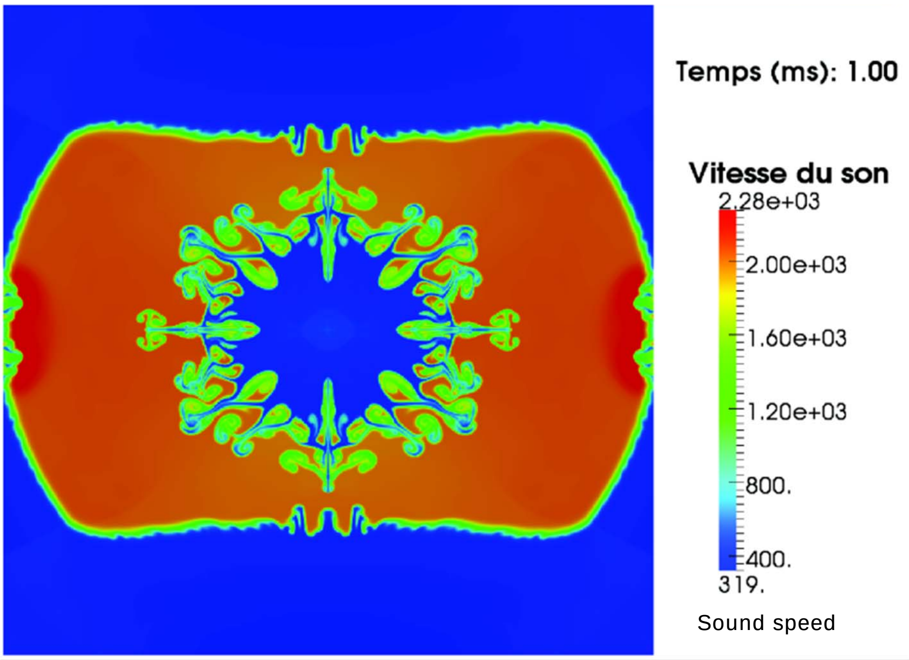
Mass fraction

CCFV modeling of RMI (2 concentric bubbles)

(1.8 M elements)

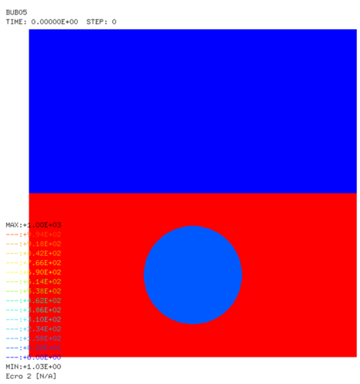
Solution at 1 ms

86



Sound speed

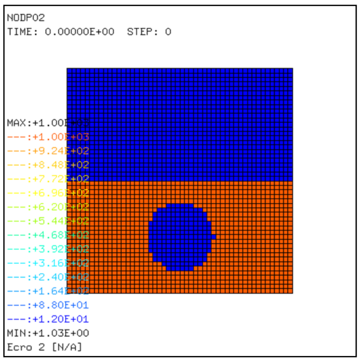
Exercise 7 – Gas bubble expansion in liquid



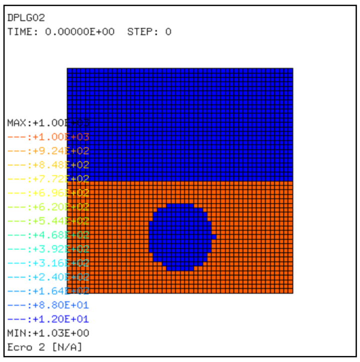
Solution with structured mesh (method of Despres, Lagoutière et al.)

88

Exercise 7 – Gas bubble expansion in liquid (2)



Without anti-diffusion

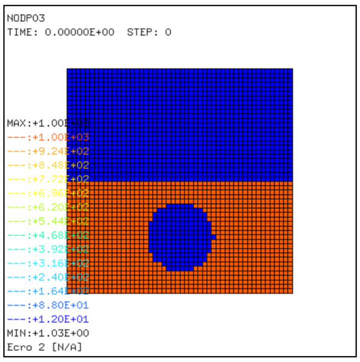


With anti-diffusion (DPLG)

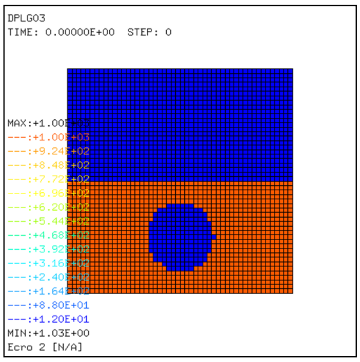
Mesh : 50 x 50 = 2,500 **CAR1** elements

89

Exercise 7 – Gas bubble expansion in liquid (3)



Without anti-diffusion

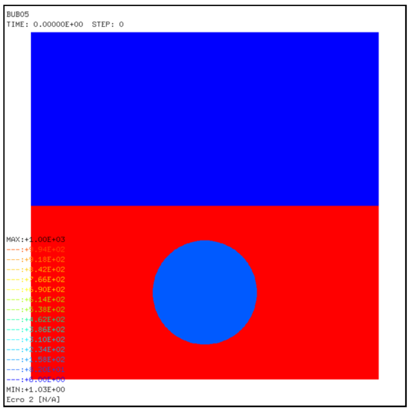


With anti-diffusion (DPLG)

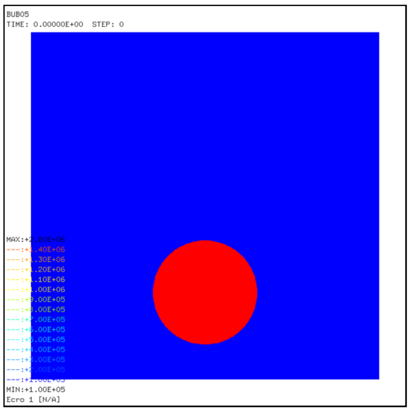
Mesh : 50 x 50 = 2,500 **Q4VF** elements

90

Exercise 7 – Gas bubble expansion in liquid (4)



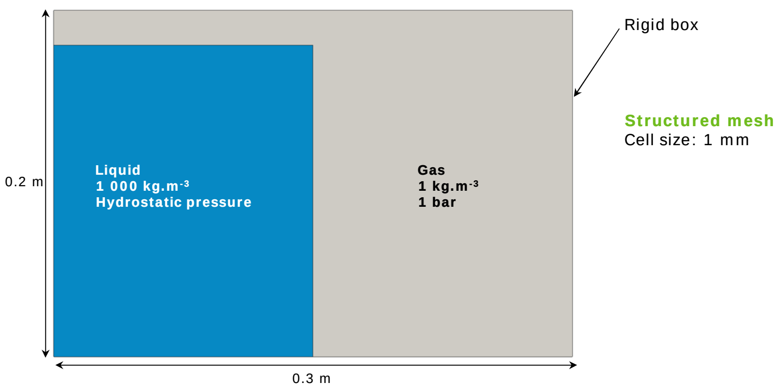
Density



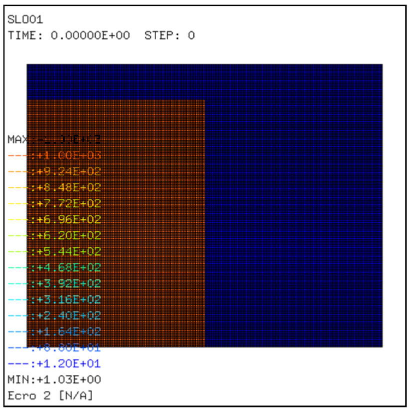
Pressure

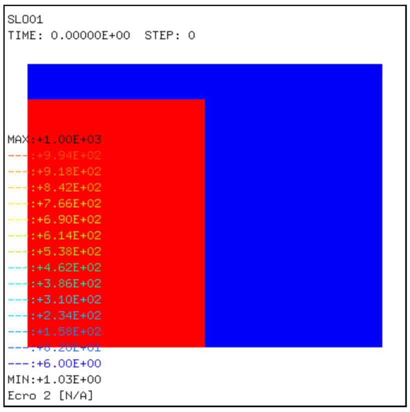
Fine mesh : 800 x 800 = 640,000 CAR1 elements, **DPLG**

Exercise 8 – Sloshing under gravity loading



Exercise 8 – Sloshing under gravity loading (2)

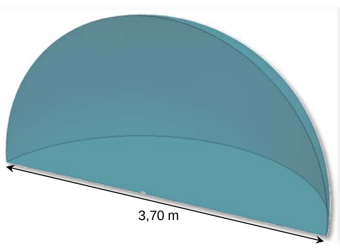




Solution with CAR1 (200 x 160 = 32,000 elements)

Pulsation of Bubble in Rigid Sphere

(Courtesy of CEA)



- Demonstration of capacity of following the life of a non-condensable bubble (cf. BORAX accident)
- *Initial conditions*: bubble of ~ **5 cl** at a pressure of **40 000 bar**
- Simulation of 3 periods
- Instability on the bubble surface at each collapse