Efficient fluid-structure interaction simulation of plates subjected to shock loading under water and in air

Ralf Deiterding

Oak Ridge National Laboratory

Int. Workshop on Fluid-Structure Interaction, Herrsching

Sep 30, 2008

Collaboration with

- Sean Mauch and Daniel Meiron (Applied and Computational Mathematics Caltech)
- Fehmi Cirak (University of Cambridge)
- David Hill (Graduate Aeronautical Laboratories Caltech)

This work is sponsored by the Office of Advanced Scientific Computing Research; U.S. Department of Energy (DOE) and was performed at the Oak Ridge National Laboratory, which is managed by UT-Battelle, LLC under Contract No. DE-AC05-00OR22725. Part of this work was also performed at the California Institute of Technology and was supported by the ASC program of the Department of Energy under subcontract No. B341492 of DOE contract W-7405-ENG-48.

Outline of the talk

- Eulerian fluid solver
  - *Governing equations*
  - *Level-set-based ghost fluid approach*
  - *Structured adaptive mesh refinement*
  - *Verification and validation*
- Fluid-structure coupling
  - *Algorithmic approach*
  - *Level set evaluation*
- Examples
  - *Verification and validation: deformation of thin plates in air and water*
  - *Large-scale validation: detonation-driven deformation and fracture of thin tubes*
- Software, conclusions

Hydrodynamic equations

Euler equations

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_k}(\rho u_k) = 0$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_k}(\rho u_i u_k + \delta_{ik} p) = 0$$

$$\frac{\partial E}{\partial t} + \frac{\partial}{\partial x_k}(u_k(E + p)) = 0$$

Stiffened gas equation of state

$$p = (\gamma - 1)(E - \frac{1}{2} u_k u_k) - \gamma p_\infty$$

Hydrodynamic equations

Euler equations with reaction

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_k}(\rho u_k) = 0$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_k}(\rho u_i u_k + \delta_{ik} p) = 0$$

$$\frac{\partial E}{\partial t} + \frac{\partial}{\partial x_k}(u_k(E + p)) = 0$$

$$\frac{\partial}{\partial t}(\rho Y) + \frac{\partial}{\partial x_k}(\rho Y u_k) = W\dot{\omega}$$

Equation of state with heat release

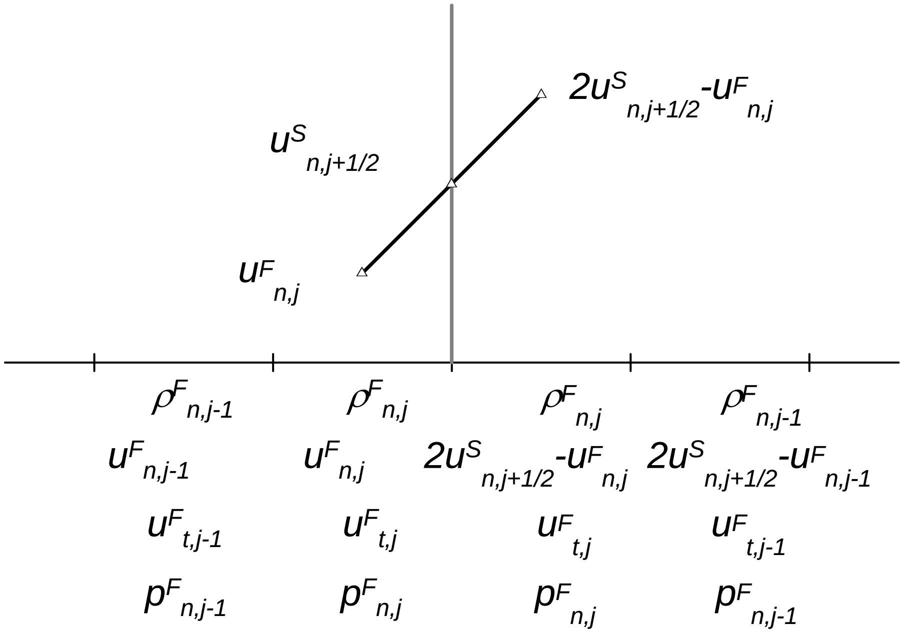
$$p = (\gamma - 1)(E - \frac{1}{2}\rho u_k u_k - \rho Y q)$$

Single exothermic chemical reaction

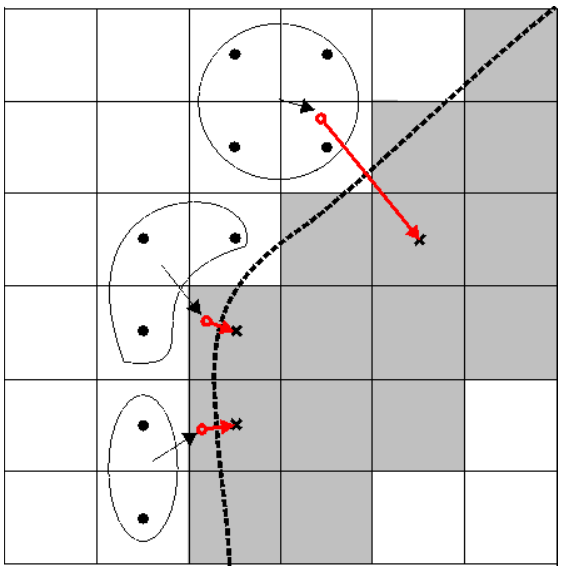
$$W\dot{\omega} = -A\rho \exp(-E/\mathcal{R}T)$$

Embedded boundary method

- Incorporate complex moving boundary/ interfaces into a Cartesian solver (extension of work by R.Fedkiw and T.Aslam)
- Implicit boundary representation via distance function φ, normal $n = \nabla\varphi / |\nabla\varphi|$
- Treat an interface as a moving rigid wall
- Method diffuses boundary and is therefore not conservative
- Construction of values in embedded boundary cells by interpolation / extrapolation



Velocity: $\mathbf{u}^F_{Gh} = 2((\mathbf{u}^S - \mathbf{u}^F_M)\cdot\mathbf{n})\,\mathbf{n} + \mathbf{u}^F_M$



- Higher resolution at embedded boundary required than with first-order unstructured scheme
- Appropriate level-set-based refinement criteria are available to cure deficiencies

Structured AMR for hyperbolic problems

- For simplicity

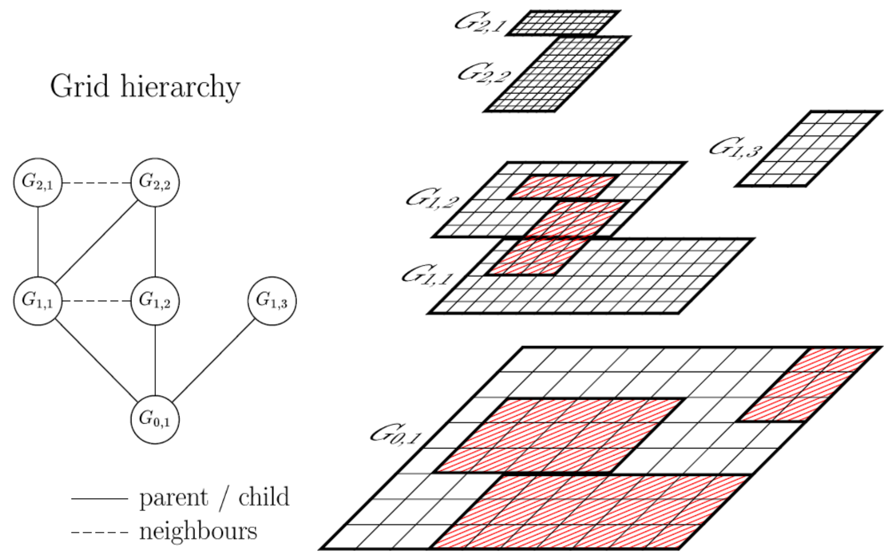
$$\partial_t \mathbf{q} + \nabla \cdot \mathbf{f}(\mathbf{q}) = 0$$

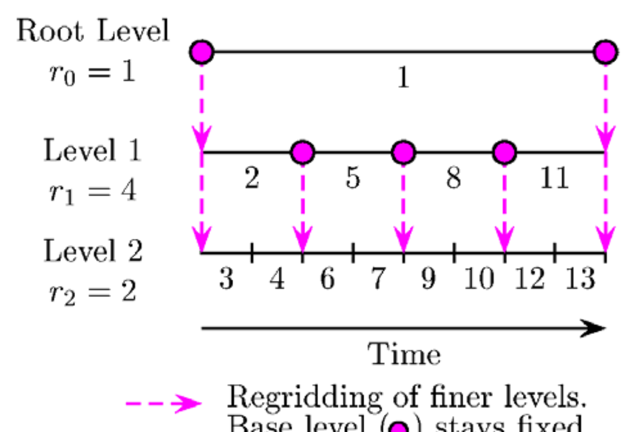
- Refined subgrids overlay coarser ones
- Computational decoupling of subgrids by using ghost cells
- Refinement in space *and* time
- Block-based data structures
- Cells without mark are refined
- Cluster-algorithm necessary
- Efficient cache-reuse / vectorization possible
- Explicit finite volume scheme

$$\mathbf{Q}^{n+1}_{jk} = \mathbf{Q}^{n}_{jk} - \frac{\Delta t}{\Delta x_1}\left[\mathbf{F}^1_{j+\frac{1}{2},k} - \mathbf{F}^1_{j-\frac{1}{2},k}\right] - \frac{\Delta t}{\Delta x_2}\left[\mathbf{F}^2_{j,k+\frac{1}{2}} - \mathbf{F}^2_{j,k-\frac{1}{2}}\right]$$

only for single rectangular grid necessary

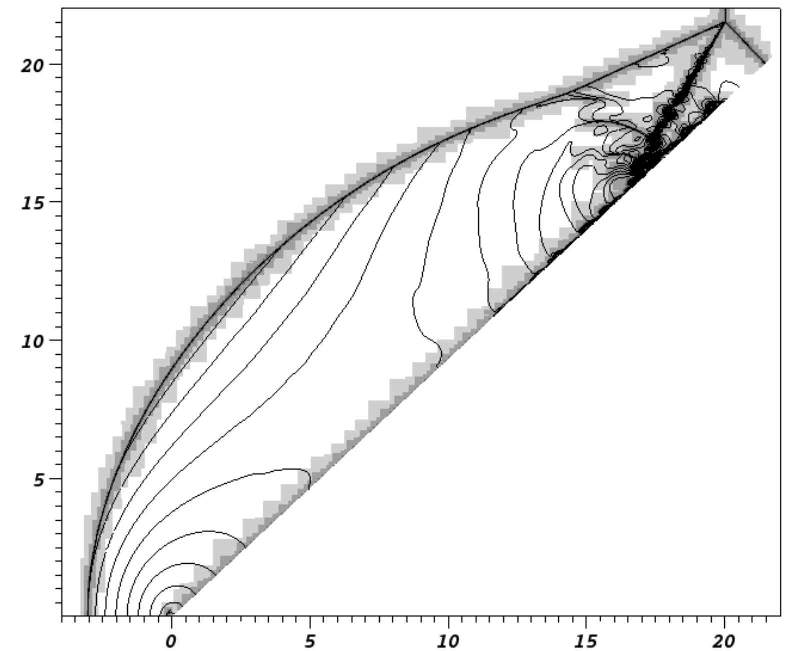
- M. Berger and P. Colella, J. Comput. Phys. 82, 1988.



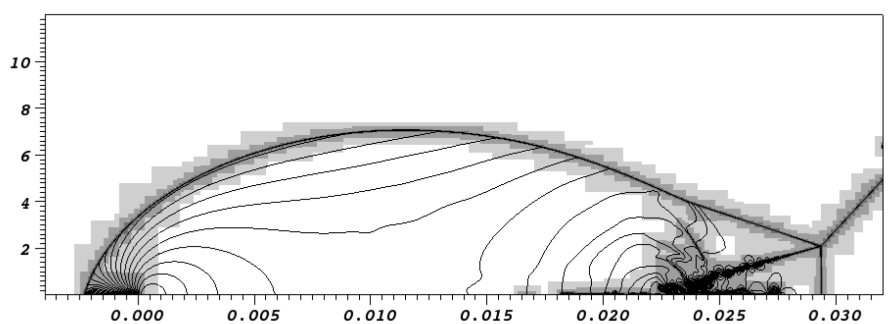


Verification of embedded boundary method

- Reflection of a Mach 2.38 shock in nitrogen at 43^o wedge
- 2nd order MUSCL scheme with Roe solver, 2nd order multidimensional wave propagation

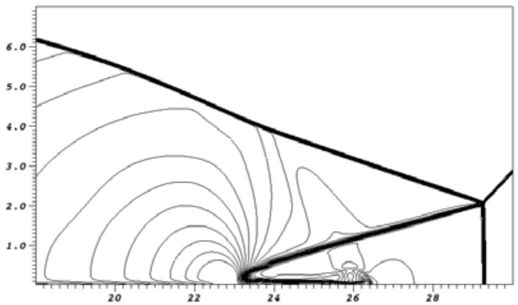


- Cartesian base grid 360x160 cells on domain of 36mm x 16mm with up to 3 refinement levels with refinement factors 2, 4, 4 $\rightarrow$ Δx=3.125 μm, 38h CPU

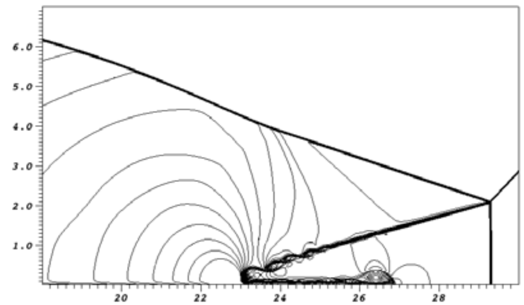


- GFM base grid 390x330 cells on domain of 26mm x 22mm with up to 3 refinement levels with refinement factors 2, 4, 4 $\rightarrow$ Δx_e=2.849 μm, 200h CPU

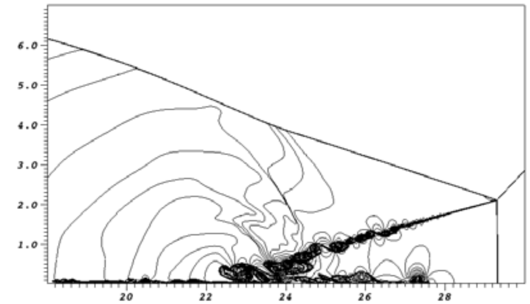
Verification of embedded boundary method, AMR



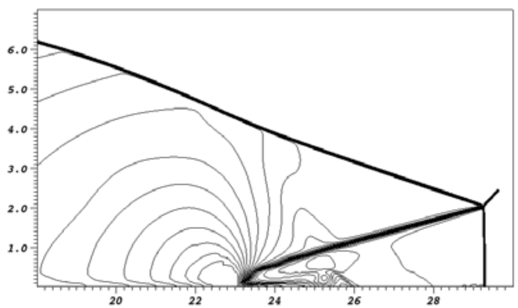
Δx=25µm



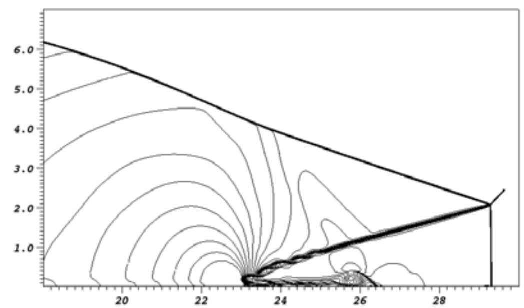
Δx=12.5µm



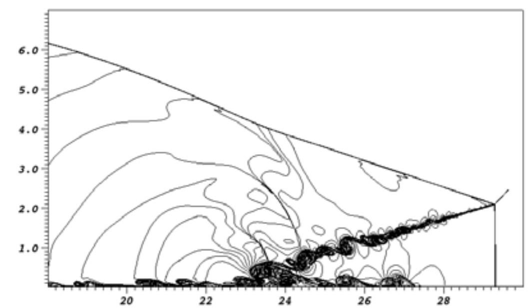
Δx=3.125µm



$\Delta x_e \approx$ 22.8µm

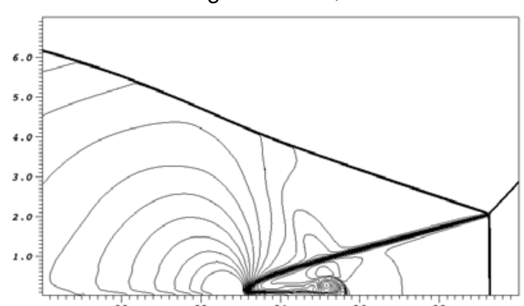


$\Delta x_e \approx$ 11.4µm

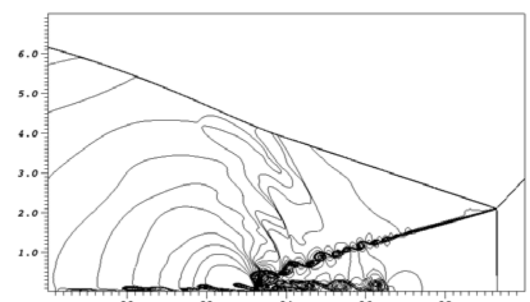


$\Delta x_e \approx$ 2.849µm

- 2nd order MUSCL scheme with Van Leer FVS, dimensional splitting



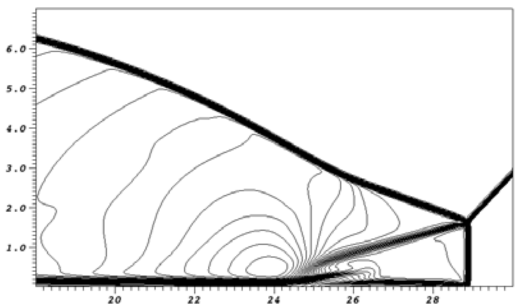
Δx=12.5µm



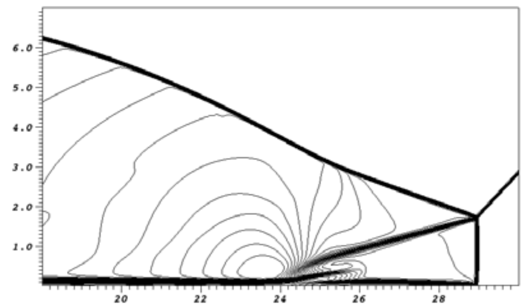
Δx=3.125µm

Navier-Stokes equations

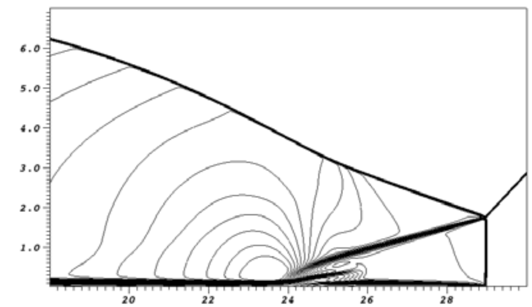
- No-slip boundary conditions, conservative 2nd order centered differences to approximate stress tensor and heat flow



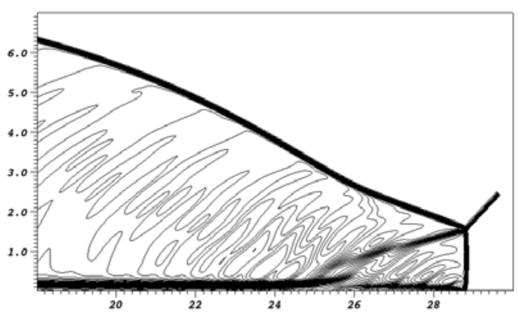
Δx=50μm



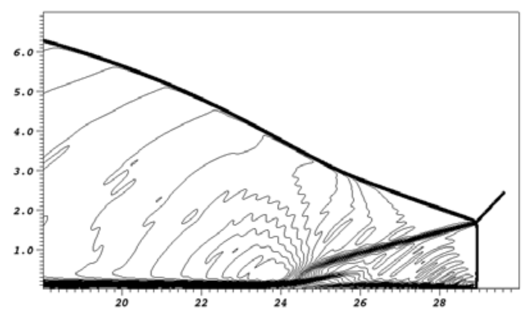
Δx=25μm



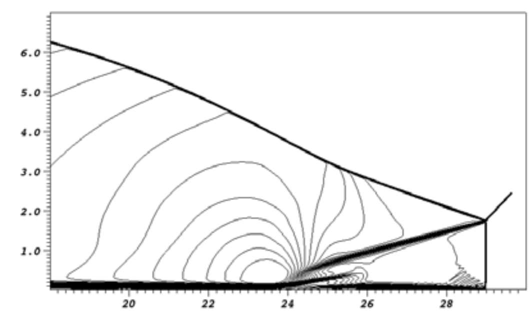
Δx=12.5μm, AMR



$\Delta x_e \approx$ 45.6μm

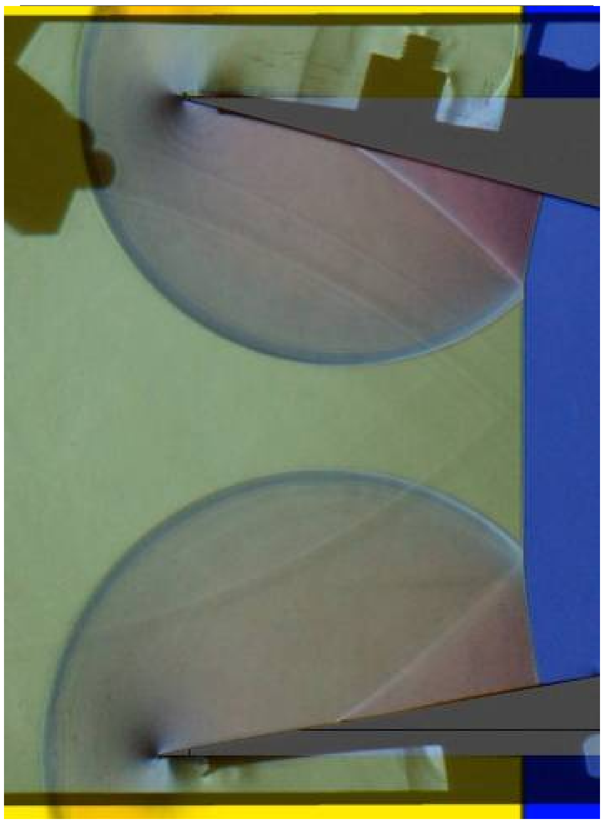


$\Delta x_e \approx$ 22.8μm

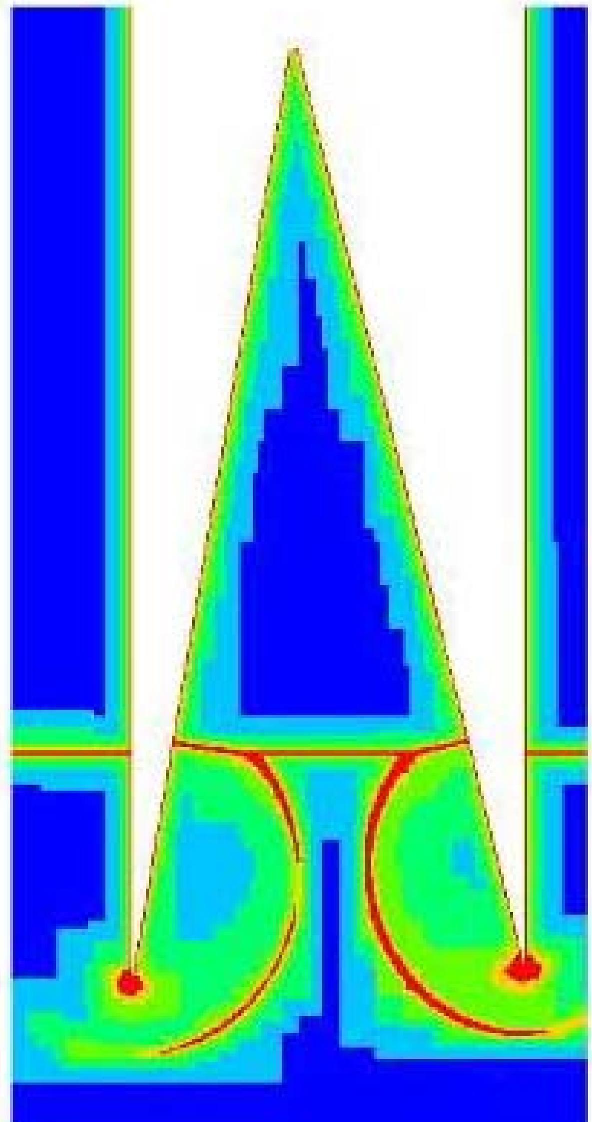


$\Delta x_e \approx$ 11.4μm, AMR

Validation: shock in converging geometry



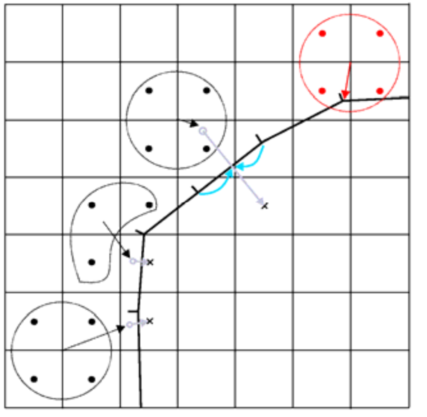
Density contours overlaying Schlieren



- Base mesh 250x100
- 4 additional refinement level, factor 2
- Simulation and graphics D. Hill

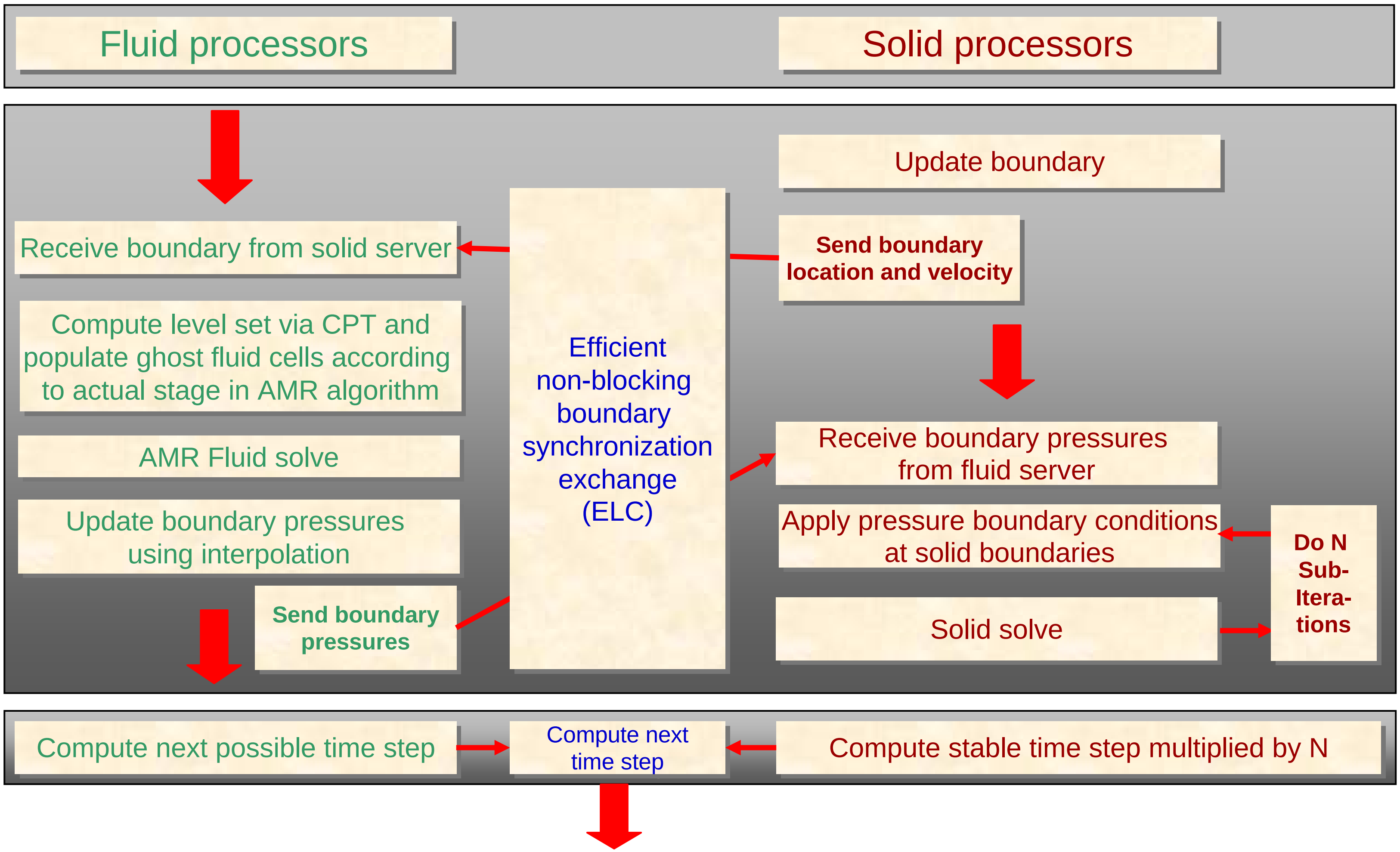
Fluid-structure coupling

- Couple compressible Euler equations to Lagrangian structure mechanics
- Compatibility conditions between inviscid fluid and solid at a slip interface
  - *Continuity of normal velocity:* $u^S_n = u^F_n$
  - *Continuity of normal stresses:* $\sigma^S_{nn} = -p^F$
  - *No shear stresses:* $\sigma^S_{n\tau} = \sigma^S_{n\omega} = 0$



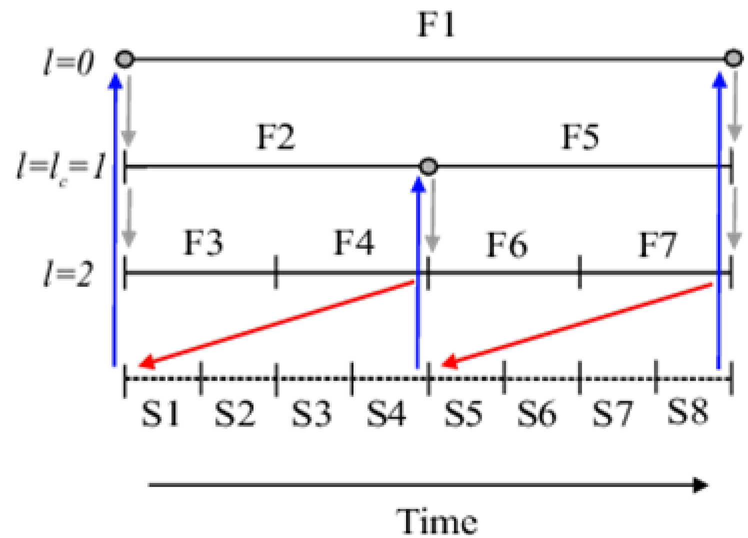
- Time-splitting approach for coupling
  - *Fluid:*
    - *Treat evolving solid surface with moving wall boundary conditions in fluid*
    - *Use solid surface mesh to calculate fluid level set*
    - *Use nearest velocity values* $\mathbf{u}^S$ *on surface facets to impose* u^F_n *in fluid*
  - *Solid:*
    - *Use interpolated hydro-pressure* p^F *to prescribe* σ^S_{nn} *on boundary facets*
- Ad-hoc separation in dedicated fluid and solid processors

Algorithmic approach for coupling



Incorporation into hierarchical SAMR

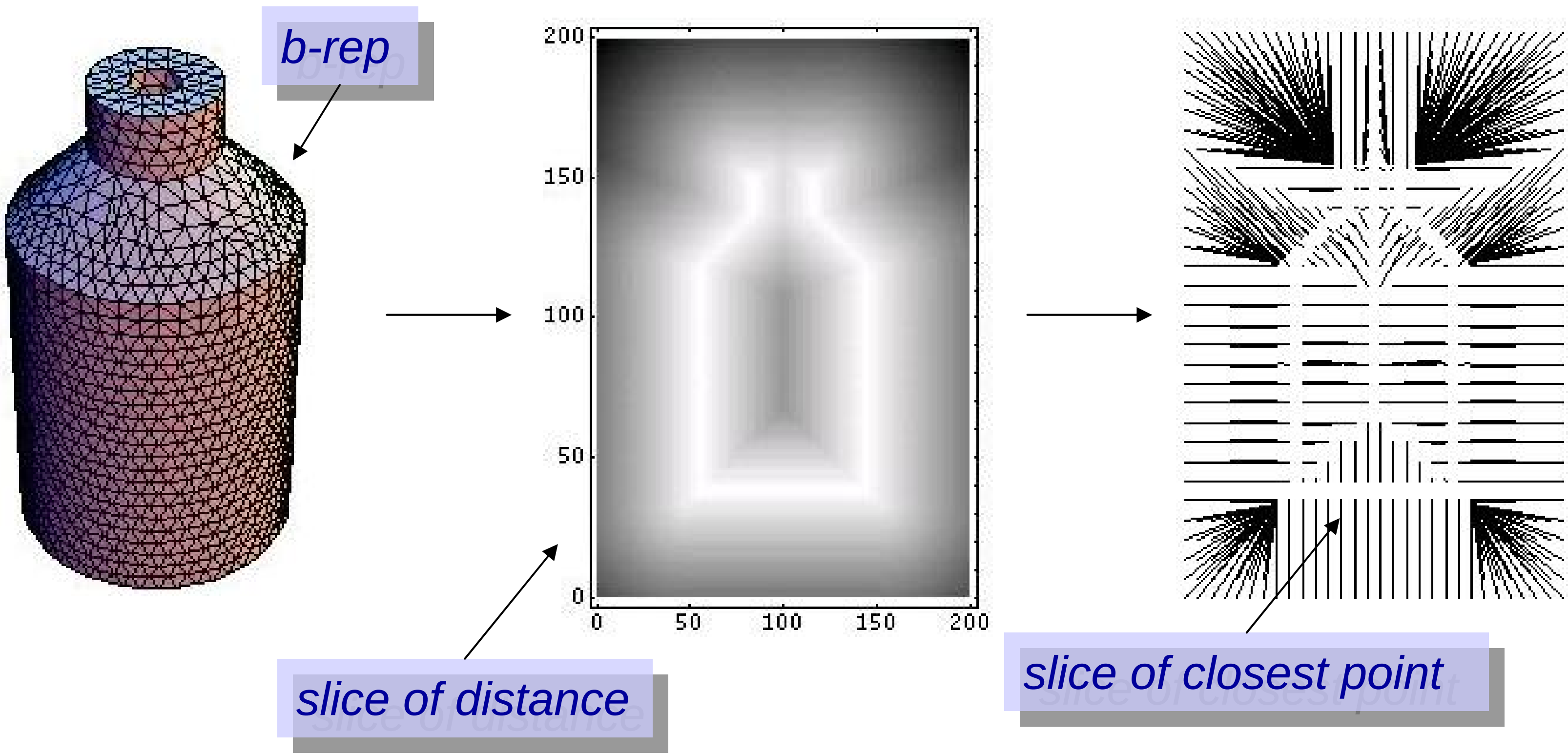
- Eulerian SAMR + non-adaptive Lagrangian FEM scheme
- Exploit SAMR time step refinement for effective coupling to solid solver
  - *Lagrangian simulation is called only at level $l_c < l_{max}$*
  - *SAMR refines solid boundary at least at level l_c*
  - *One additional level reserved to resolve ambiguities in GFM (e.g. thin structures)*
- Nevertheless: Inserting sub-steps accommodates for time step reduction from the solid solver within an SAMR cycle
- Communication strategy
  - *Updated boundary info from solid solver must be received (blue arrow) before regridding operation (gray dots and arrows)*
  - *Boundary data is sent to solid (red arrow) when highest level available*
- Inter-solver communication (point-to-point or globally) managed on the fly by current SAMR partition bounding box information by Eulerian-Lagragian-Coupling module (ELC)



- *When SAMR mesh partitioning is done at runtime, the entire solid mesh must have been received (SAMR partitions must be allowed to change arbitrary)*
- *During strictly local regridding operations only the local portion of the solid mesh has to be received*

Implicit representations of complex surfaces

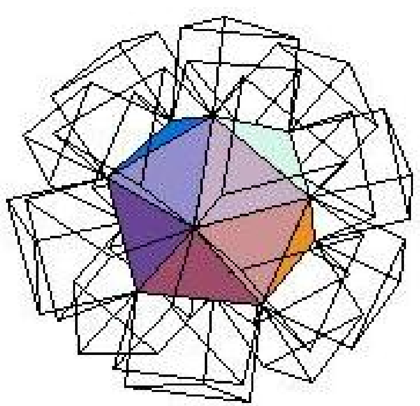
- FEM Solid Solver
  - *Explicit representation of the solid boundary, b-rep*
  - *Triangular faceted surface*
- Cartesian FV Solver
  - *Implicit level set representation*
  - *need closest point on the surface at each grid point*



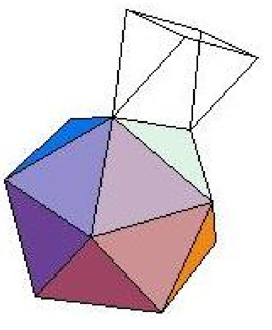
→ Closest point transform algorithm (CPT) by S. Mauch

CPT in linear time

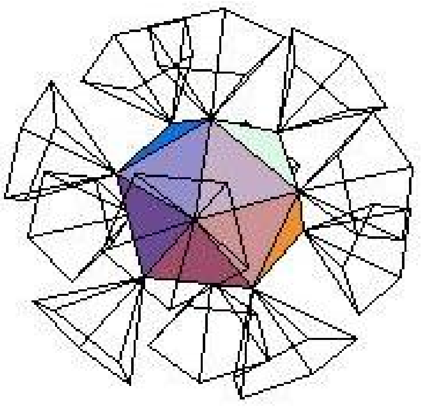
- Problem reduction by evaluation only within specified max. distance
- The characteristic / scan conversion algorithm.
  - *For each face/edge/vertex.*
    - *Scan convert the polyhedron.*
    - *Find distance, closest point to that primitive for the scan converted points.*
- Computational complexity.
  - *O(m) to build the b-rep and the polyhedra.*
  - *O(n) to scan convert the polyhedra and compute the distance, etc.*



Face Polyhedra



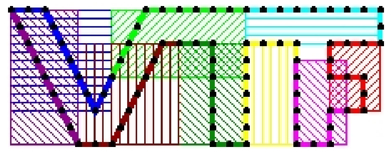
Edge Polyhedra

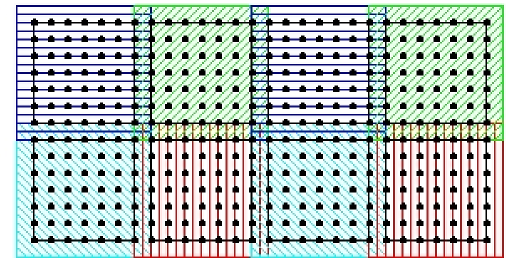


Vertex Polyhedra

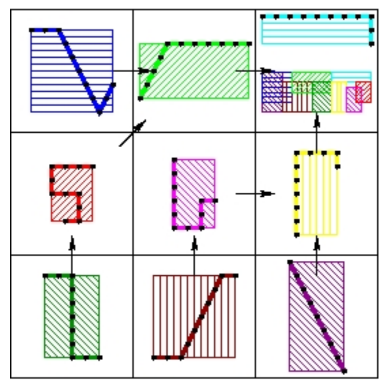
ELC communication module

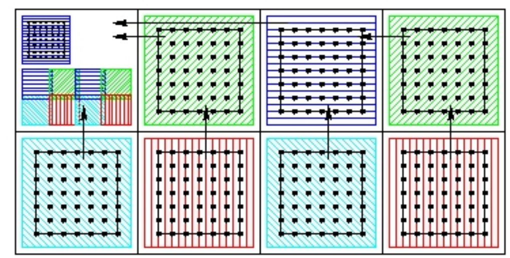
1. Put bounding boxes around each solid processor's piece of the boundary and around each fluid processor's grid.



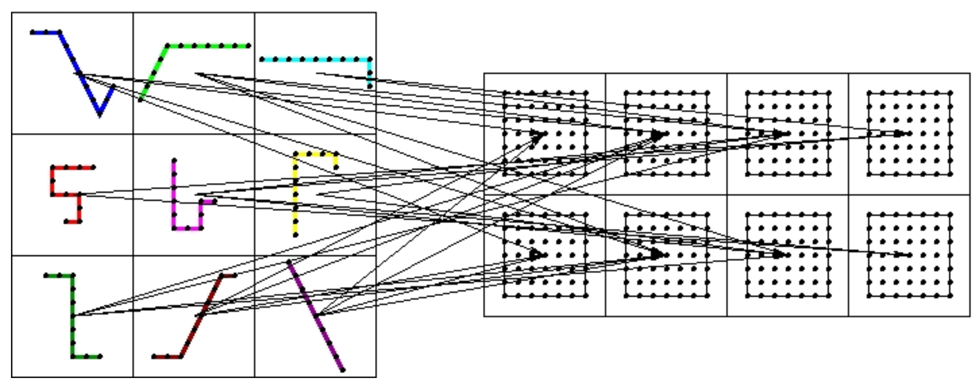


2. Gather, exchange and broadcast of bounding box information





3. Optimal point-to-point communication pattern, non-blocking

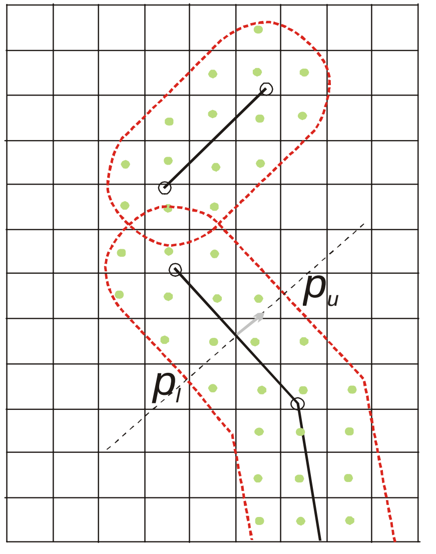


Shock-induced panel motion

- Elastic motion of a thin steel plate (thickness h=1mm, length 50mm)
- Steel plate modeled with finite difference solver using the beam equation

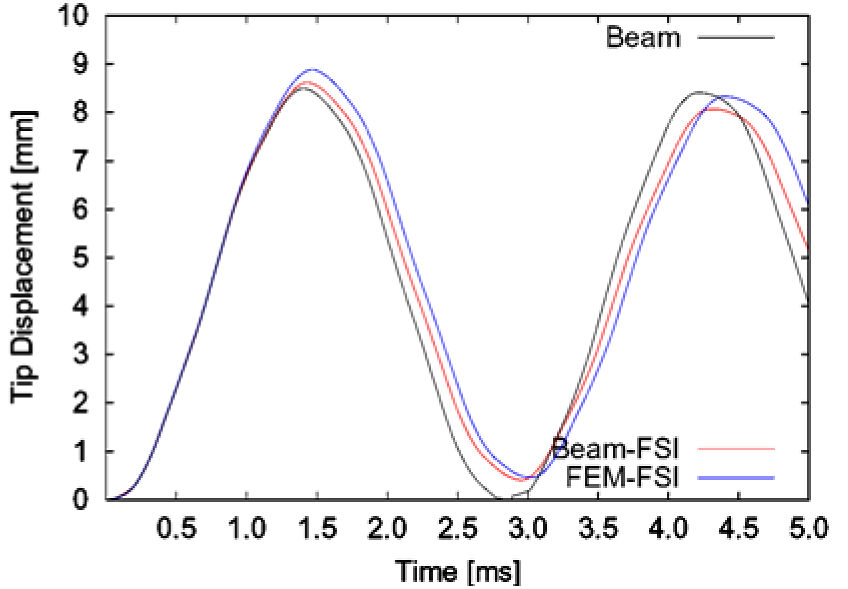
$$\rho h \frac{\partial^2 w}{\partial t^2} + EI \frac{\partial^4 w}{\partial x^4} = p(x,t)$$

- Thin boundary structures or lower-dimensional shells require "thickening" to apply embedded boundary method
  - *Unsigned distance level set function φ*
  - *Treat cells with $0<\varphi<d$ as ghost fluid cells (indicated by green dots)*
  - *Leaving φ unmodified ensures correctness of $\nabla\varphi$*
  - *Refinement criterion based on φ ensures reliable mesh adaptation*
  - *Use face normal in shell element to evaluate in $\Delta p = p_u - p_l$*



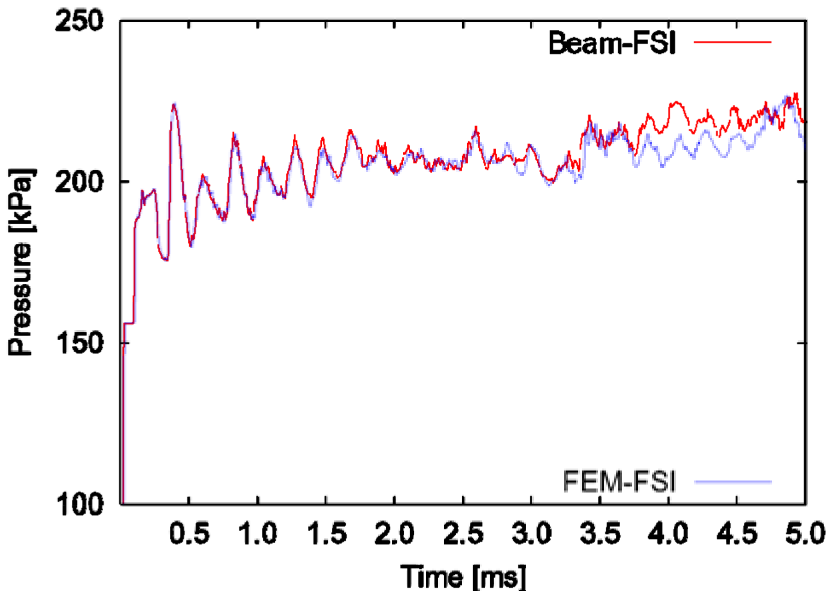
Panel motion – FSI verification

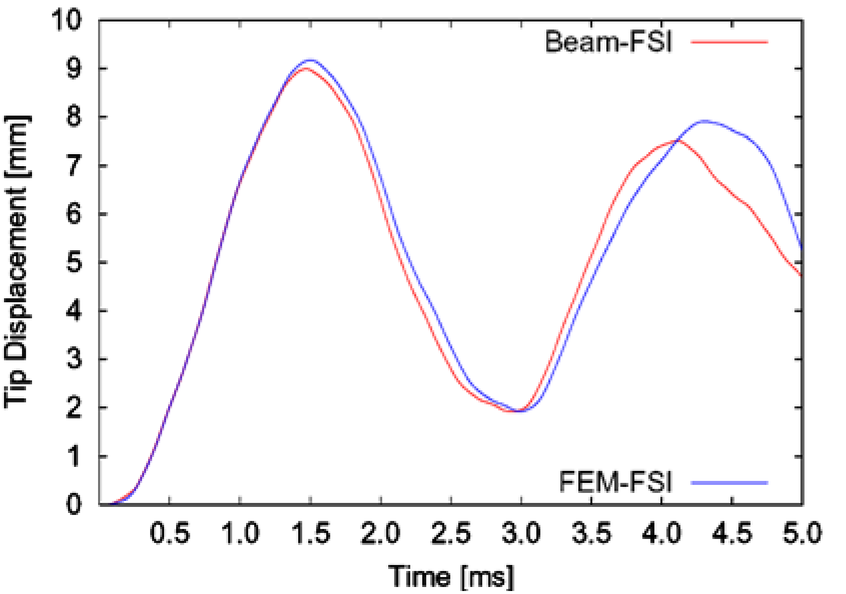
- FSI verification: constant impulsive loading of Δp=100kPa
  - *Beam: No FSI*
  - *Beam-FSI: beam solver with coupled FV code*
  - *FEM-FSI: large displacement thin-shell finite element solver by F.Cirak coupled to FV code*



Panel motion – dynamic impact

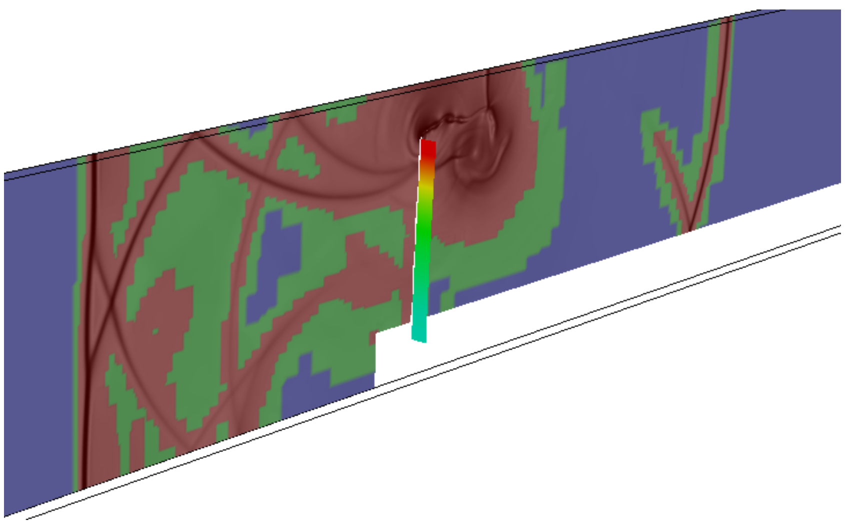
- Forward facing step geometry, reflective boundaries everywhere except inflow at left side, panel 1.5cm behind start of step
- SAMR base mesh 320x64(x2), 2 additional level with factors 2, 2
- Intel 3.4GHz Xeon dual processors connected with Gigabit Ethernet
  - *Beam-FSI: 12.25h CPU on 3 fluid CPU + 1 solid CPU*
  - *FEM-FSI: 322h CPU* on 14 fluid CPU + 2 solid CPU
- Pressure measured at upper boundary, 0.01m before beam





Panel motion – dynamic impact

- Forward facing step geometry, reflective boundaries everywhere except inflow at left side, panel 1.5cm behind start of step
- SAMR base mesh 320x64(x2), 2 additional level with factors 2, 2
- Intel 3.4GHz Xeon dual processors connected with Gigabit Ethernet
  - *Beam-FSI: 12.25h CPU on 3 fluid CPU + 1 solid CPU*
  - *FEM-FSI: 322h CPU* on 14 fluid CPU + 2 solid CPU
- Pressure measured at upper boundary, 0.01m before beam



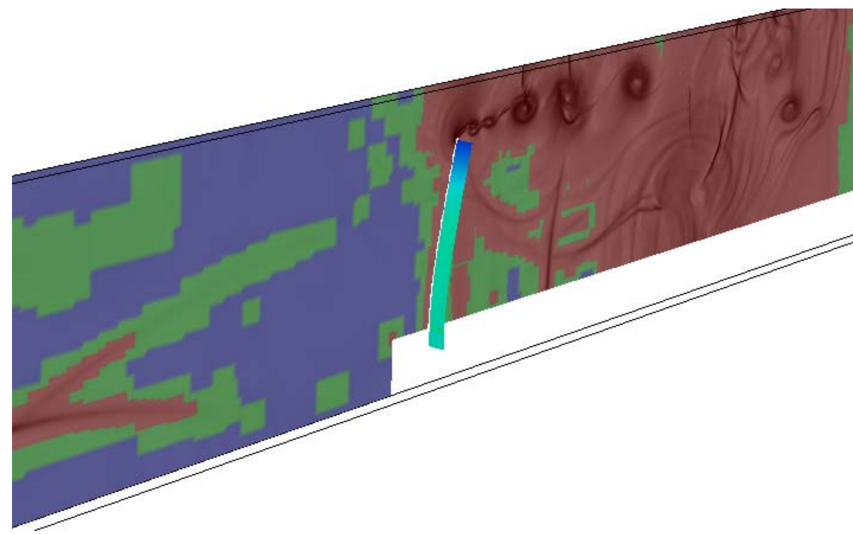


Plate deformation from underwater explosion

- 3d simulation of plastic deformation of air backed aluminum plate with r=85mm from underwater explosion at standoff distance d
- J.Z. Ashani, A.K. Ghamsari, Mat.-wiss. Werkstofftechn. 39(2), 2008.

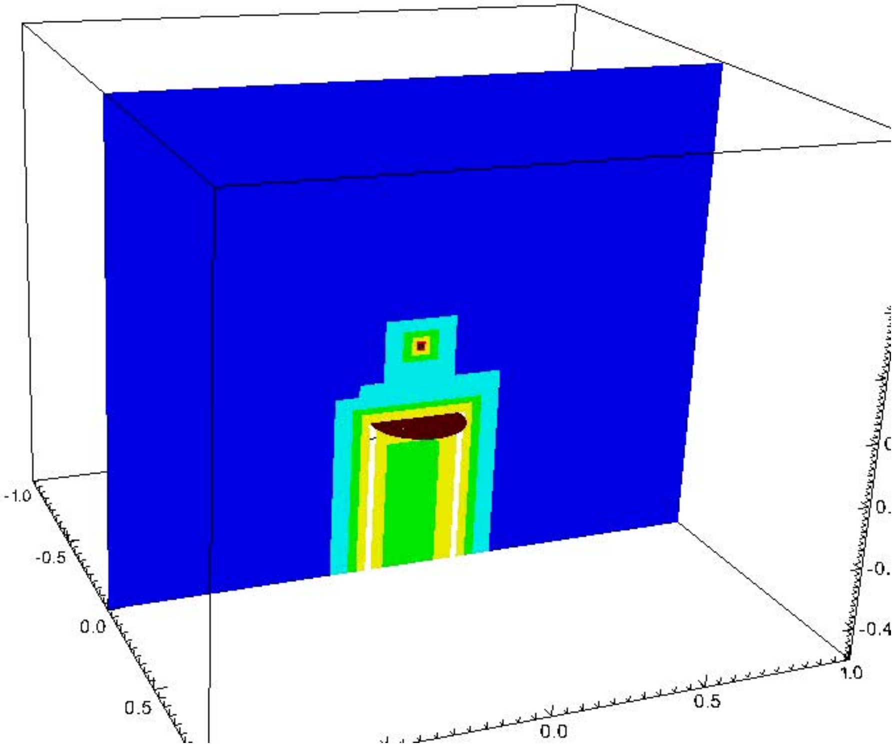
Fluid

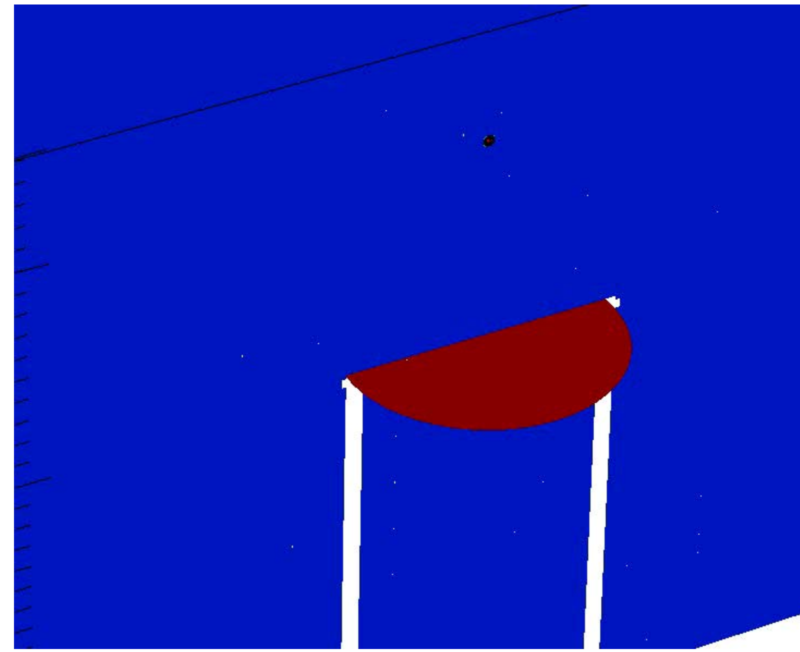
- Domain of 2m x 1.6m x 2m
- Modeling of water with stiffened gas equation of state with γ=7.415, p_∞=296.2 MPa
- Cavitation modeling with pressure cut-off at p=-1 MPa
- Explosion modeled as energy increase in sphere with r=5mm, loading by C4 (15g, 20g, 30g), d=25cm or 30cm above plate
- AMR base grid 50x40x50, 4 additional levels, with 2,2,2,4 (uniform $3.27 \cdot 10^9$ cells), highest level restricted to initial explosion center, 3rd and 4th level to plate vicinity, l_c=3

Solid

- Aluminum plate of 3mm, conventional J2 plasticity model, parameters according to paper
- Triangular mesh with 8148 elements
- Computations of 1296 coupled time steps to t_e=1ms
- 10+2 nodes 3.4 GHz Intel Xeon dual processor, ~130h CPU

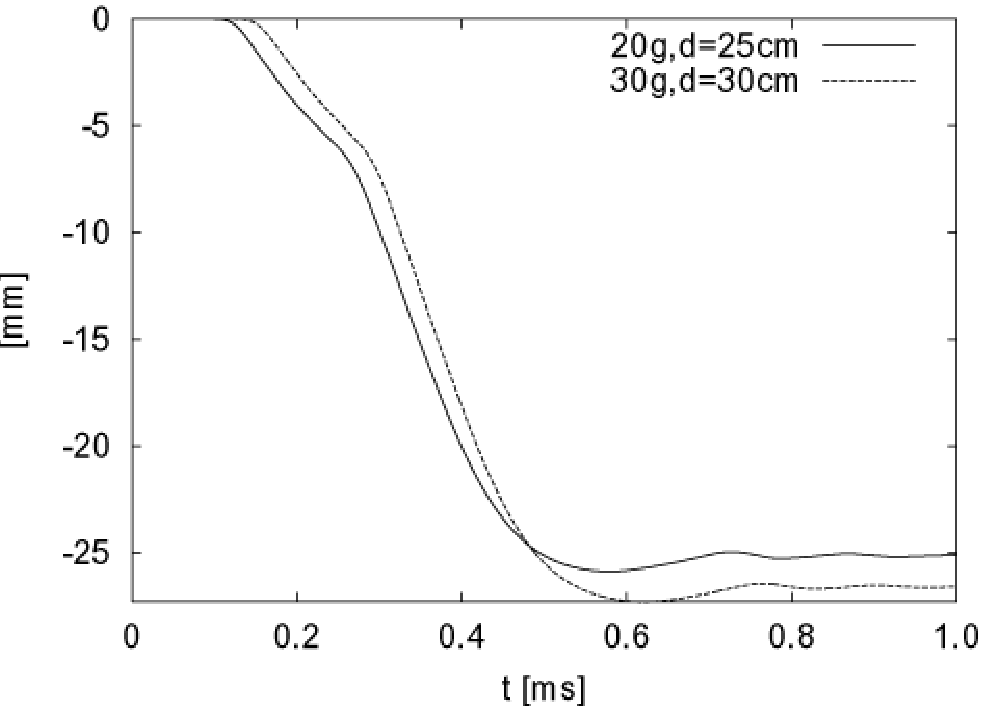
Underwater explosion: Results

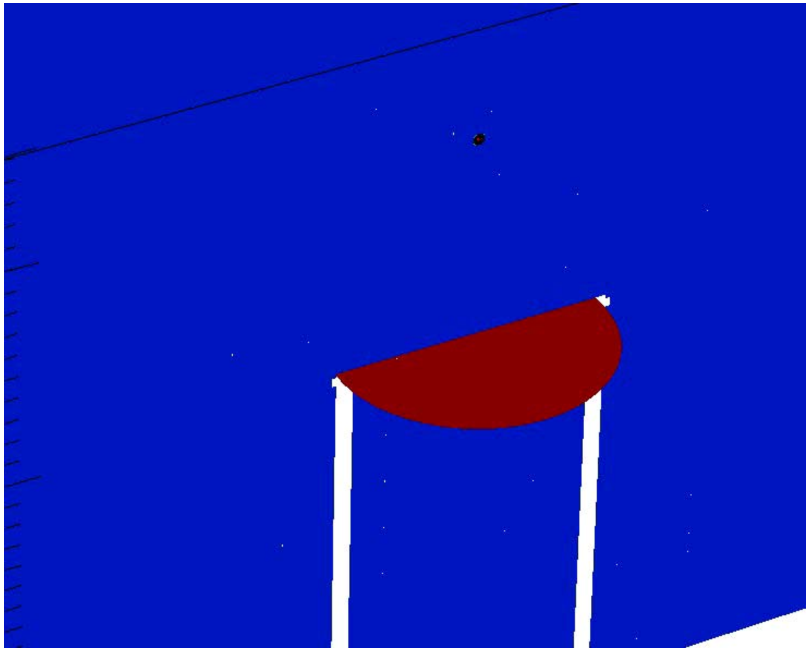




Isolines of pressure on refinement levels

Underwater explosion: Results

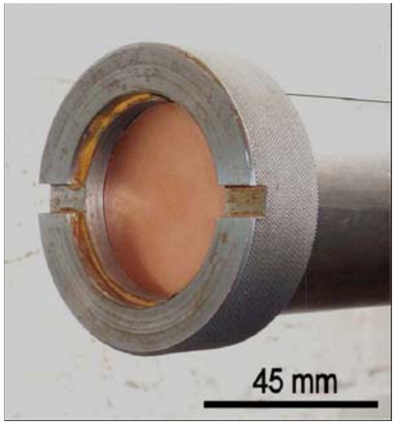


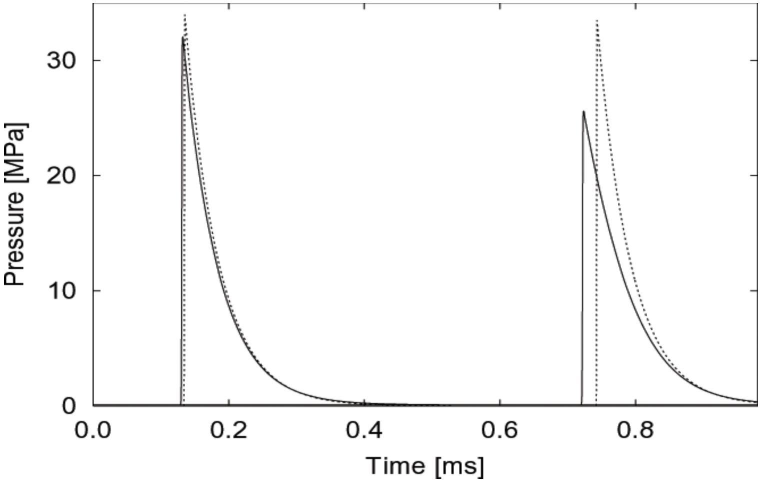


Maximal deflection	Experiment [mm]	Simulation [mm]
20g, d=25cm	28.83	25.88
30g, d=30cm	30.09	27.31

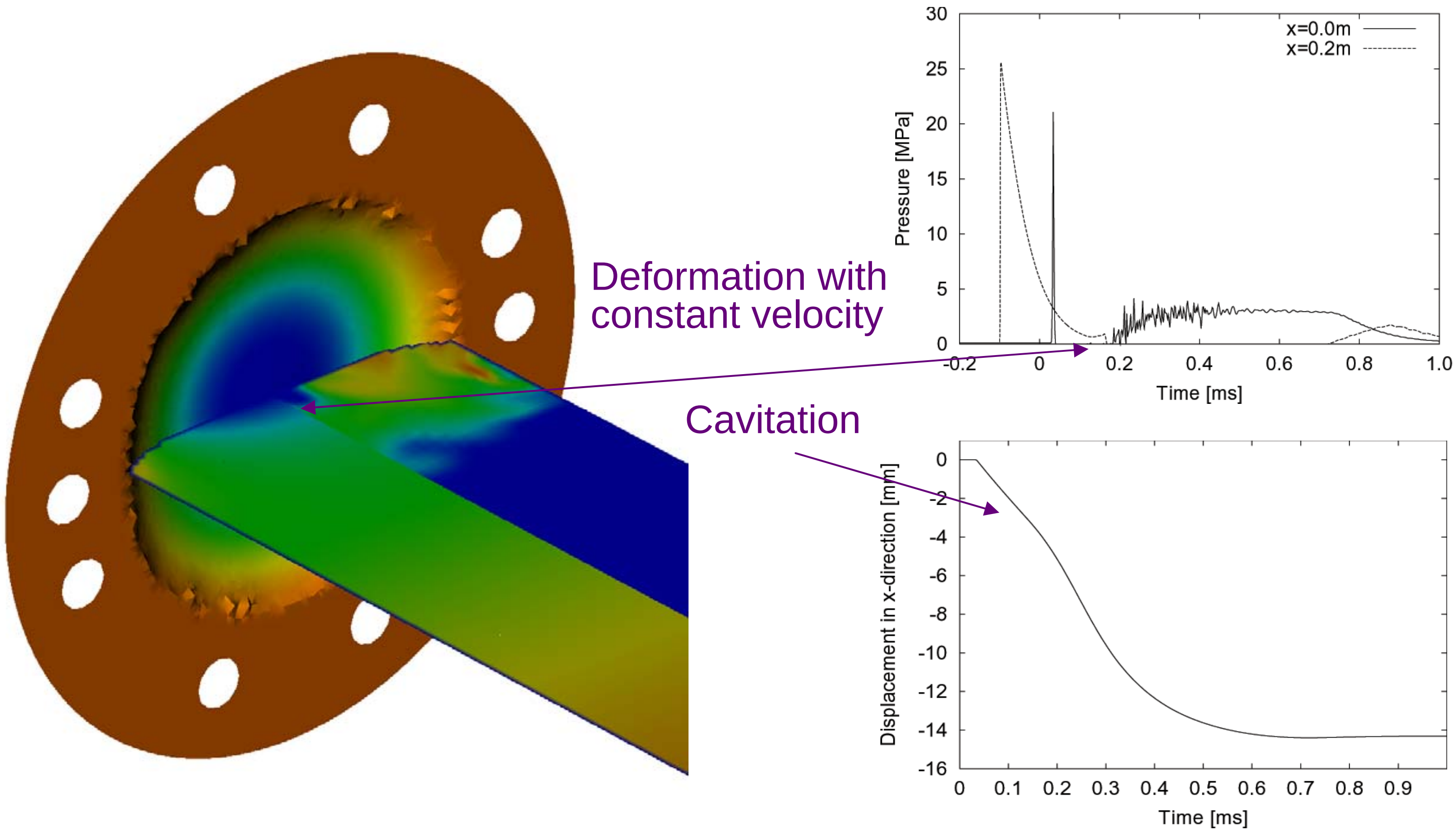
Plate deformation from water hammer

- 3d simulation of plastic deformation of thin copper plate attached to the end of a pipe due to water hammer
- Strong over-pressure wave in water is induced by rapid piston motion at end of tube
- Experiments from ‘An underwater shock simulator’, V.S. Deshpande et al., Proc. Royal Soc. A 462, 2006.
- Two-component model based on “stiffened” gas equation of state
- Computation uses $\gamma^{Air}=1.4$, $p_\infty^{Air}=0$, $\gamma^{Water}=7.415$, $p_\infty^{Water}=2962$ bar
- Cavitation modeling with pressure cut-off at p=0 MPa, no surface tension
- Realistic pressure loading in simulations created by solving equation of motion for piston
- Left: comparison of simulated pressure wave compared to analytically derived traveling wave solution (dotted) for initial pressure p_0=34 MPa



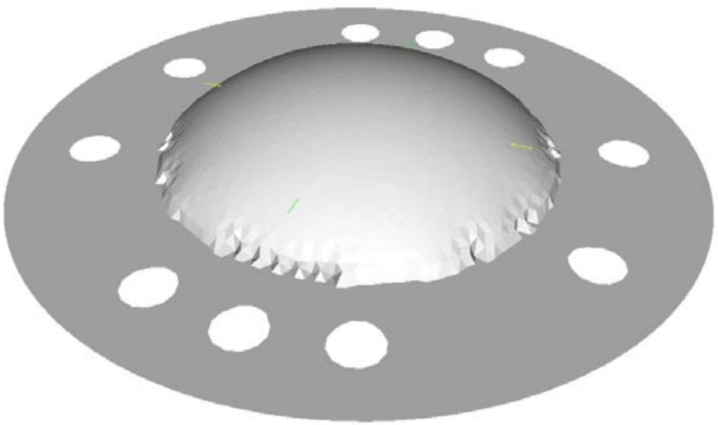


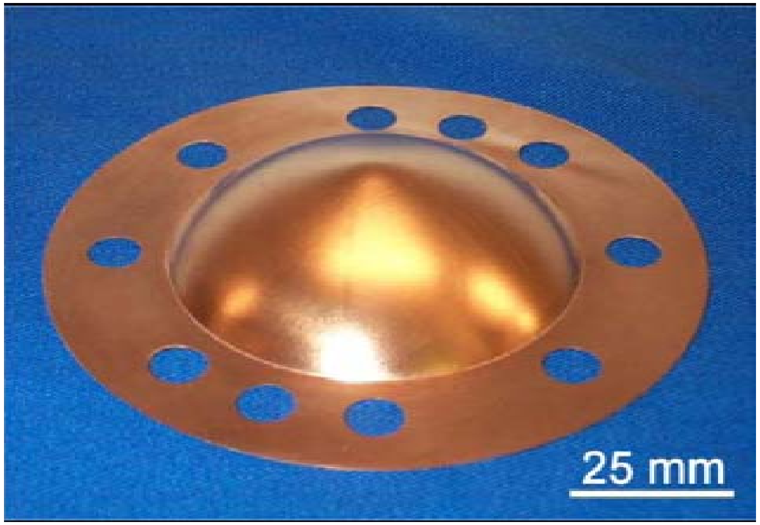
Plastic deformation



- p_0=34 MPa
- *8 nodes 3.4 GHz Intel Xeon dual processor, ~130h CPU*

Plate deformation

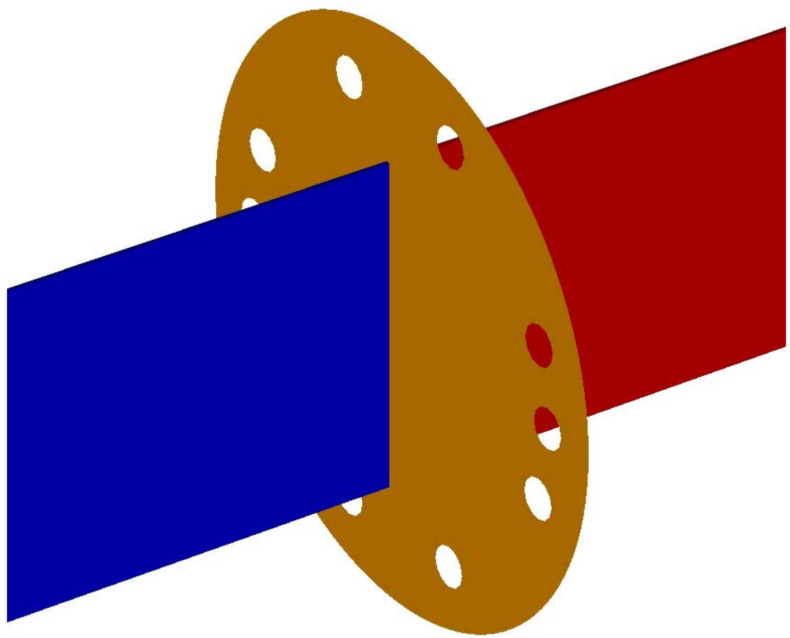




Comparison of plate at end of simulation and experiment.

Plate fracture

- Unsigned distance function to accommodate topology changes
- Copper plate of 0.25mm, J2 plasticity model with hardening, rate sensitivity, and thermal softening, cohesive interface model
- Fluid base mesh: 374x20x20, 2 additional levels, refinement factor 2,2, solid mesh: 8896 triangles, ~1250 time steps to t=1.0 ms
- 6+6 nodes 3.4 GHz Intel Xeon dual processor, ~800h CPU



Initial pressure
p_0=64 MPa

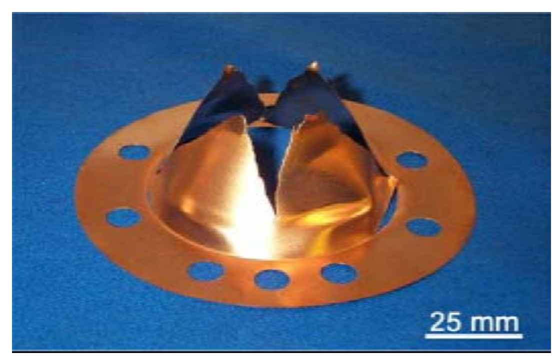
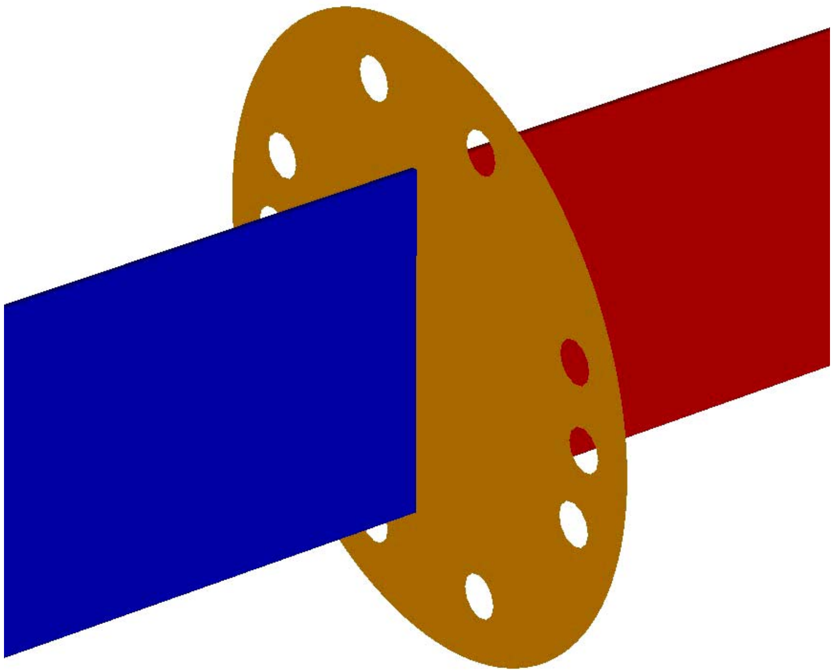


Plate fracture

- Unsigned distance function to accommodate topology changes
- Copper plate of 0.25mm, J2 plasticity model with hardening, rate sensitivity, and thermal softening, cohesive interface model
- Fluid base mesh: 374x20x20, 2 additional levels, refinement factor 2,2, solid mesh: 8896 triangles, ~1250 time steps to t=1.0 ms
- 6+6 nodes 3.4 GHz Intel Xeon dual processor, ~800h CPU



p_0=173 MPa

Fracture at hinges!

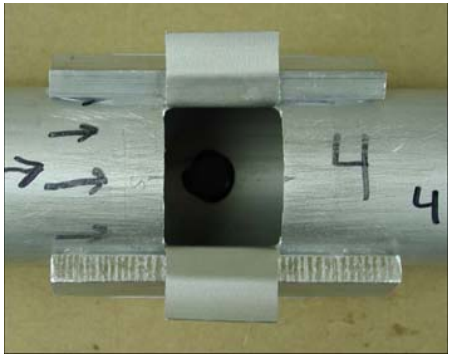
Benchmarking of fluid solver

- Calculation of coupled 50 time steps on 3.4 GHz Intel Xeon dual processor
- *F* processors used for fluid solver, *S* processors for solid solver
- AMR base level: 350x20x20, 2 additional levels, refinement factor 2,2
- l_c =2, 5 sub-iterations in solid solver

Task	F=6, S=2 %	F=12, S=4 %	F=24, S=8 %	F=48, S=16 %	F=96, S=32 %
Integration	27.3	22.4	16.1	12.5	8.8
Boundary sync	41.3	39.6	48.1	50.1	47.3
Recomposition	3.3	5.5	6.4	8.5	10.4
Interpolation	1.0	1.0	0.7	0.5	0.4
Regridding	0.7	0.9	0.6	0.4	0.4
GFM Find cells	3.2	2.7	2.0	1.6	1.2
GFM Interpolation	10.0	8.6	6.1	4.7	3.5
GFM Overhead	1.6	0.7	0.3	0.5	0.3
CPT	0.5	0.6	0.7	0.9	1.3
Level set sync	2.6	7.3	8.6	9.1	11.3
ELC	5.7	7.4	7.6	8.0	10.8
Coupling data calc	0.5	0.3	0.3	0.3	0.2
Misc	2.3	3.0	2.5	2.9	4.1
Time per step [s]	**23**	**14**	**11**	**7**	**5**

Fluid-structure interaction validation – tube with flaps

- Experiments by T. Chao, J. C. Krok, J. Karnesky, F. Pintgen, J.E. Shepherd
- $C_2 H_4 + 3 O_2$ CJ detonation for p_0=100kPa drives plastic opening of pre-cut flaps

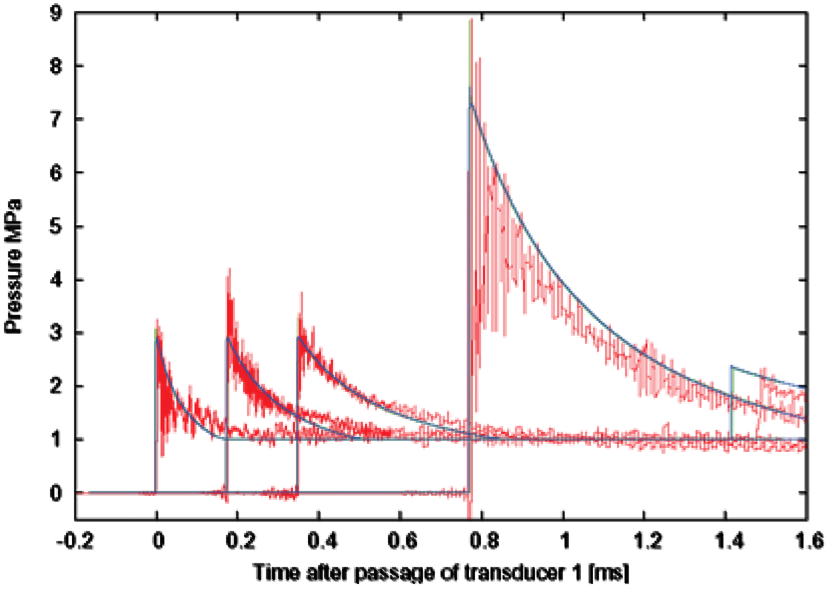


Fluid

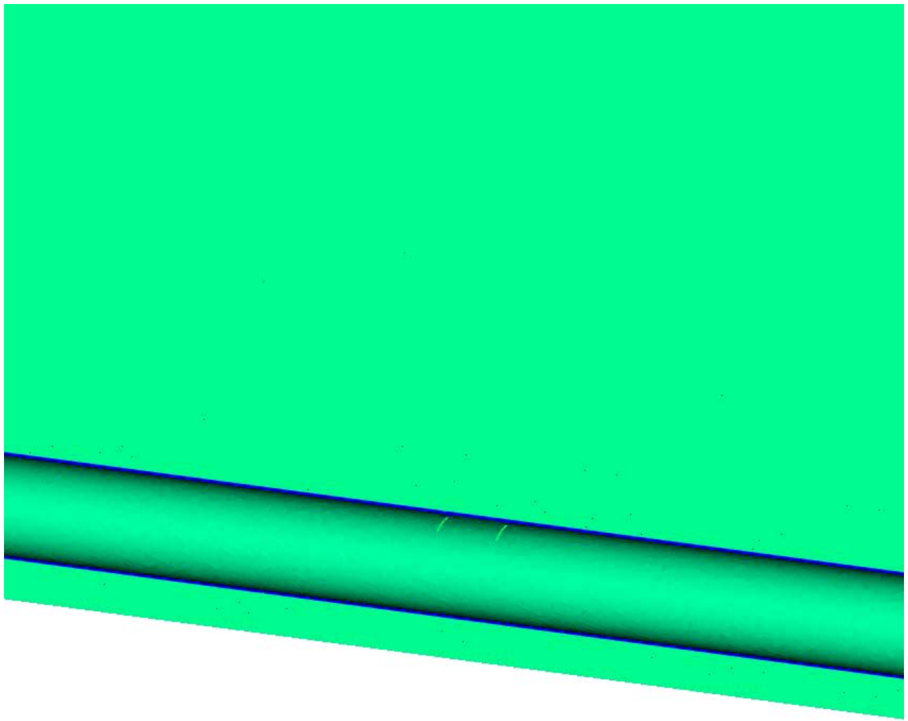
- Constant volume burn detonation model
- AMR base level: 104x80x242, 3 additional levels, factors 2,2,4
- Approx. $4 \cdot 10^7$ cells instead of $7.9 \cdot 10^9$ cells (uniform)
- Tube and detonation fully refined
- Thickening of 2d mesh: 0.81mm on both sides (real thickness on both sides 0.445mm)

Solid

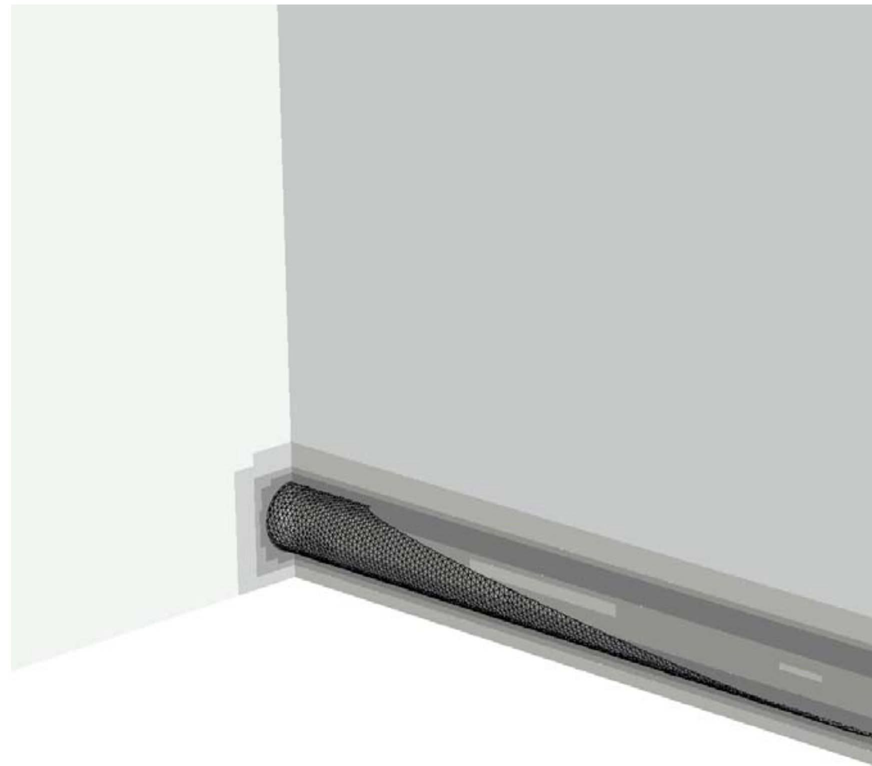
- Aluminum, J2 plasticity with hardening, rate sensitivity, and thermal softening
- Mesh: 8577 nodes, 17056 elements
- 16+2 nodes 2.2 GHz AMD Opteron quad processor, ~ 4320h CPU to t=450 μs



Tube with flaps - Results



Fluid density and diplacement in y-direction in solid



Schlieren plot of fluid density on refinement levels

Coupled fracture simulation

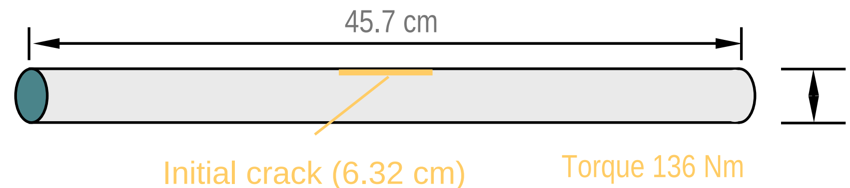
- C2 H4+3 O2 CJ detonation for p_0=180kPa drives tube fracture
- Motivation*:* Full configuration

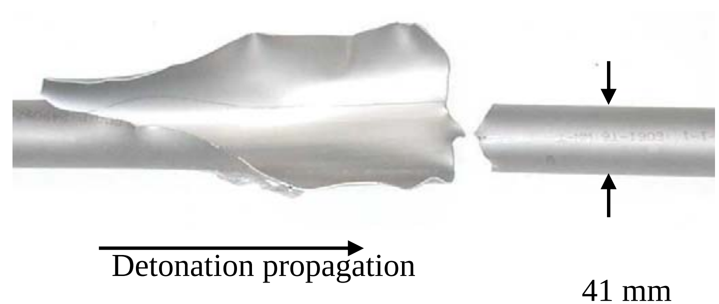
Fluid

- Constant volume burn model
- 40x40x725 cells unigrid

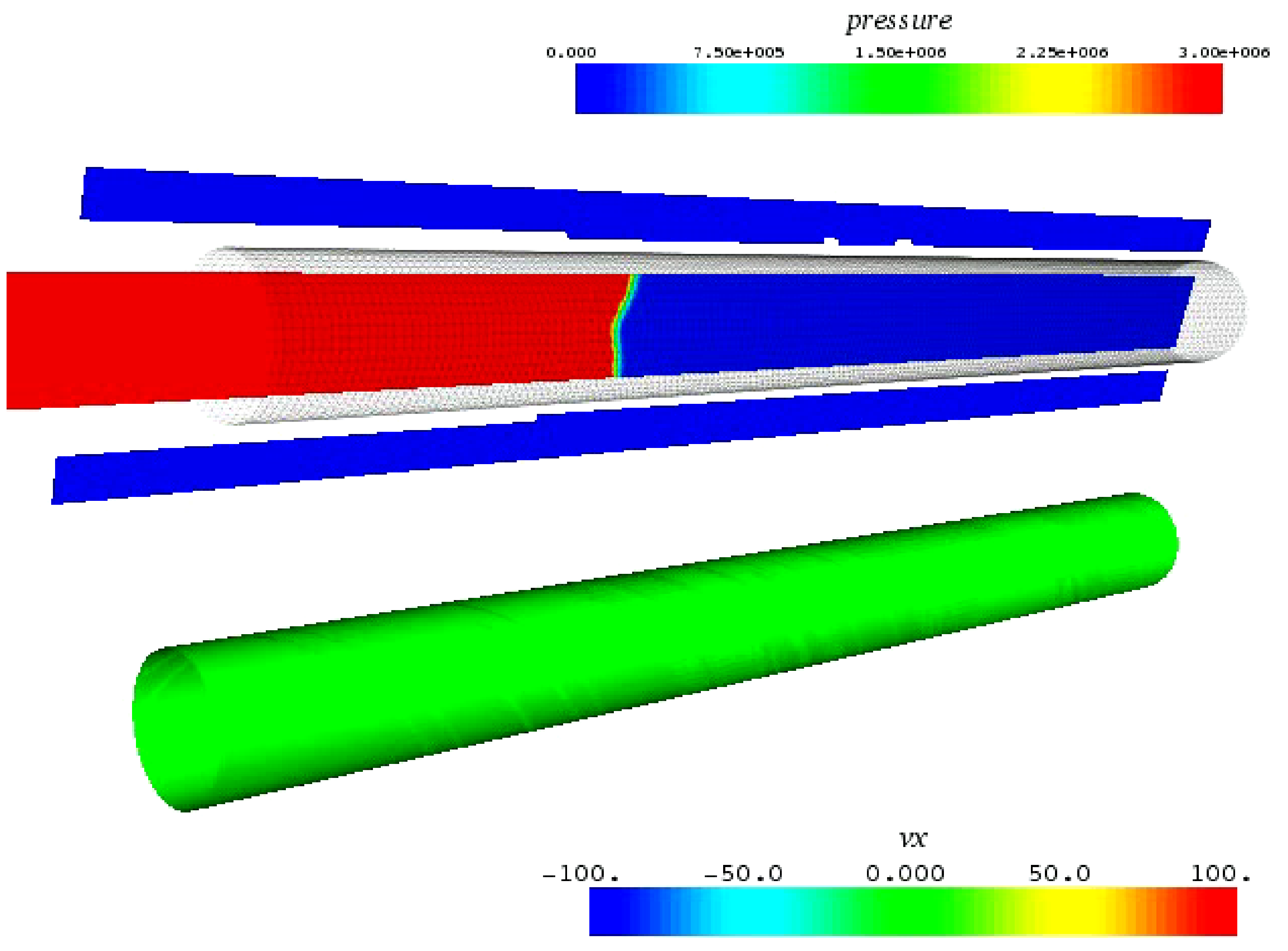
Solid

- Aluminum, J2 plasticity with hardening, rate sensitivity, and thermal softening
- Material model for cohesive interface: Linearly decreasing envelope
- Mesh: 206,208 nodes
- 27 nodes Pentium 4-dual-processor with 33 shell and 21 fluid processors
- Ca. 972h CPU



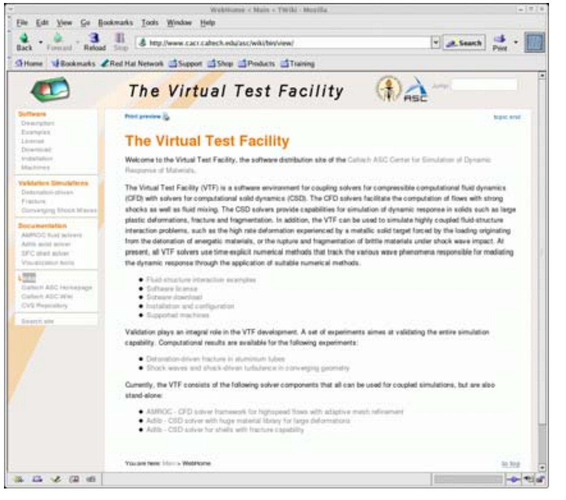


Coupled fracture simulation



Virtual Test Facility software

- Make majority of software available to open source community
  - *Release fully functional solvers with large number of single-solver applications plus several fully functional FSI applications*
  - *Include solver and FSI unit tests plus reference results to allow users to verify software integrity*
- Language: object-oriented C++ with components in C, F77, F90
- autoconf / automake environment with support for typical parallel high-performance systems
- Webpage: http://www.cacr.caltech.edu/asc
  - *Installation, configuration, examples*
  - *Scientific and technical papers*
  - *Archival of key simulation and experimental results*
  - *Source code documentation*
  - *Downloadable software with example simulations*



Summary

- Developed Cartesian Eulerian fluid solver framework that allows easy coupling to Lagrangian solid mechanics solver
- Focus is currently on time-explicit schemes and weak coupling
- Generic embedded boundary implementation allows consideration of moving embedded boundaries in any Cartesian finite volume method
- Boundaries can be described directly through level set functions or arbitrary triangulated surface meshes
- Structured mesh adaptation ensures accuracy
- Several interfaces to finite element solvers already implemented, e.g. LLNL Dyna3d
- All components parallelized with MPI for distributed memory machines