The Virtual Test Facility: an Infrastructure for Shock-driven Fluid-Structure Interaction Simulation

Ralf Deiterding, Fehmi Cirak, Sean P. Mauch, Daniel I. Meiron

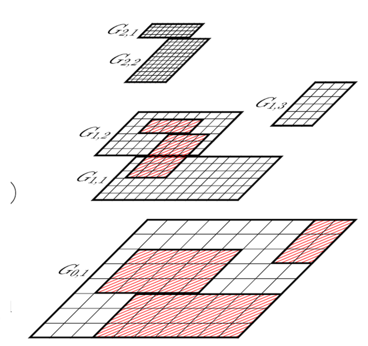
California Institute of Technology

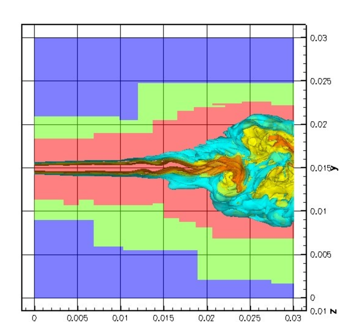
Center for Dynamic Response of Materials
California Institute of Technology
Mail-Code 158-79
Pasadena, CA 91125
http://www.cacr.caltech.edu/asc

Abstract. The Virtual Test Facility is an infrastructure for the coupled simulation of shock- and detonation-wave-driven fluid-structure interaction problems. The key idea of the VTF is to orchestrate the boundary data exchange between Lagrangian solid mechanics solvers and Eulerian fluid mechanics schemes through a ghost fluid approach in which complex embedded boundaries are implicitly represented with evolving level set functions. The level set information is derived on-the-fly with the Closest-point-transform (CPT) algorithm. The VTF software is constructed on top of the block-structured Cartesian SAMR framework Amroc and provides straightforward interfaces to incorporate any unstructured computational solid mechanics (CSD) solver.

Amroc fluid solver framework

- Encapsulates dynamic mesh adaptation and parallelization to the fluid solver developer
  - Numerical scheme only for single block necessary
- Available 3d patch solvers:
  - WENO-TCD LES upwind scheme for turbulence simulations
  - Riemann-solver-based MUSCL and Wave Propagation schemes for simplified and detailed chemistry
- Implementation of Berger-Collela SAMR algorithm coupled to generic ghost-fluid implementation for multiple level sets
- Conservative correction for purely Cartesian problems



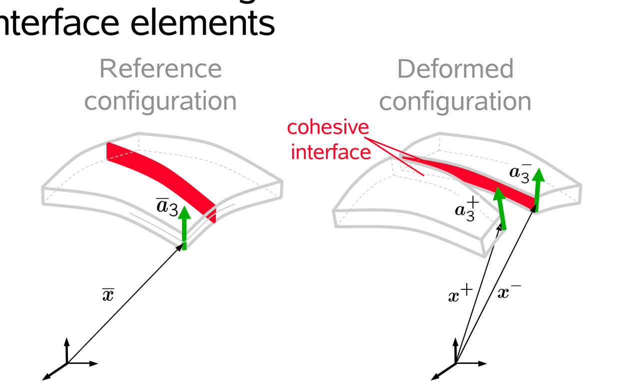


Right: LES of turbulent jet with WENO-TCD patch solver .

Available CSD solvers

SFC finite element solver

- Tetrahedral thin-shell solver in 3d
- Subdivision shell finite elements
  - Stretching and bending resistance
  - Large deformations
- Parallel explicit shell dynamics
  - Fully scalable communications
- Geometric modeling capabilities
- Access to a number of constitutive models
  - Adlib models as well as own implementations
- Parallel contact
- Fracture and fragmentation with cohesive interface elements



Fractured thin-shell kinematics used in SFC.

Adlib finite element solver

- Hexahedral solver in 3d
- Parallel explicit dynamics
- Solid modeling
- Fully scalable unstructured parallel meshing
- Thermo-mechanical coupling and multi-physics model
- Extensive constitutive library
  - single and polycrystal plasticity
  - ab-initio equation of state
- shock physics, artificial viscosity
- Contact
- Fracture and fragmentation

Dyna3d finite element solver

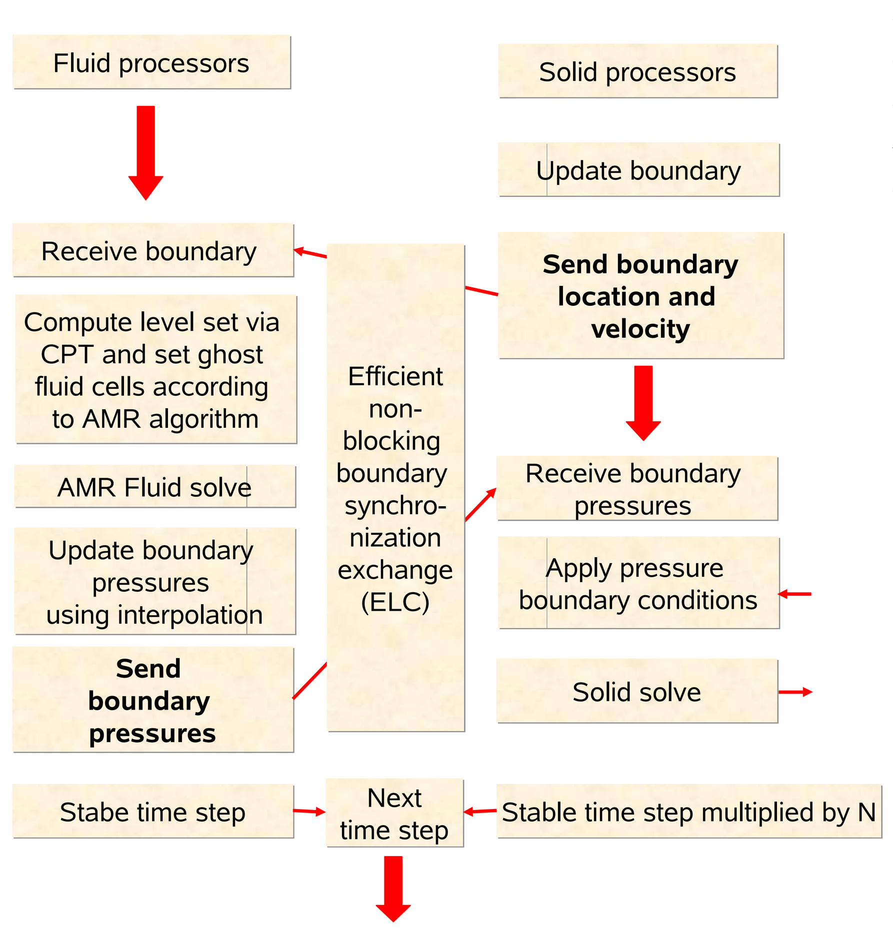
- General-purpose software for complex, real-world problems with mixed elements
- Serial version supported for now

Finite difference beam solver

- Implicit 1d solver for demonstration and verification purposes

Fluid-Structure coupling

- Coupling of Euler equations for compressible flow 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
    $\sigma^S_{n\tau} = \sigma^S_{n\omega} = 0$
- Interpolation operations with solid surface mesh
  - Mirrored fluid density and velocity values $\boldsymbol{u}^F$ into ghost cells
  - Solid velocity values $\boldsymbol{u}^S$ on facets
  - Fluid pressure values in surface points (nodes or face centroids)

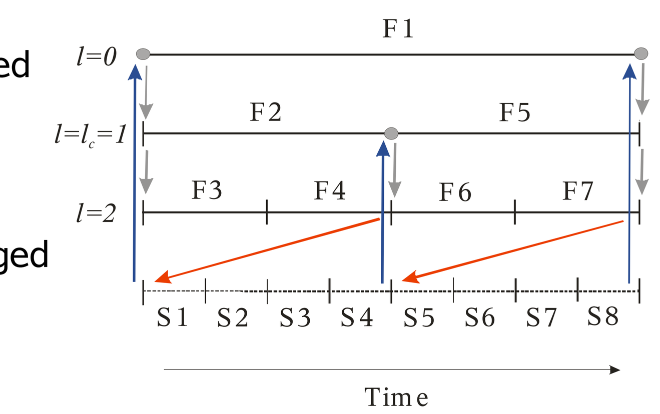


Right: Coupling algorithm: data exchange between dedicated fluid and solid processors.

Coupling of Eulerian SAMR to a non-adaptive CSD solver

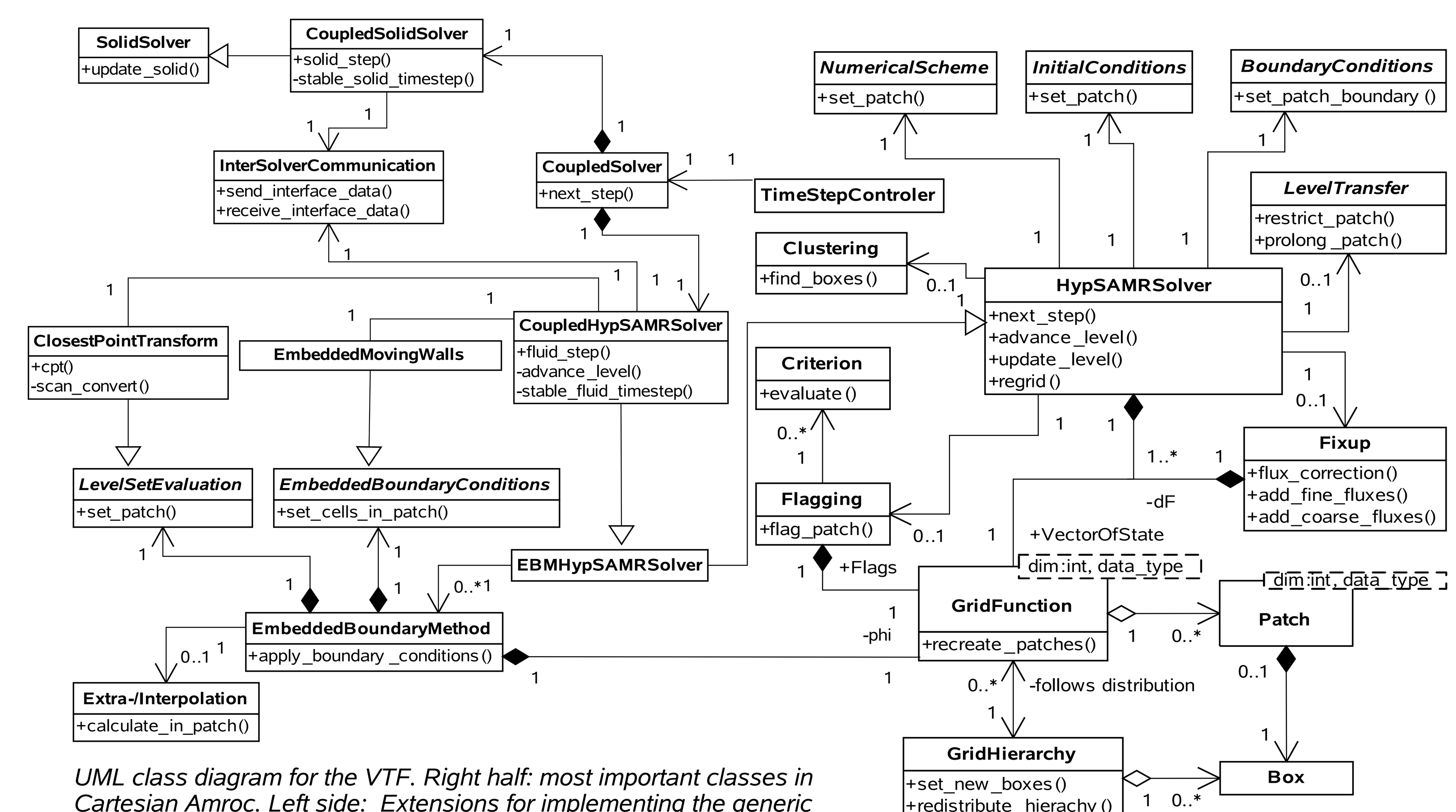
Exploit SAMR time step refinement for effective coupling to solid solver
- Lagrangian simulation is called only at level $l_c < l_{max}$
- SAMR refines solid boundary at least at level l_c
- One additional level reserved to resolve ambiguities in GFM (e.g. thin structures)

- Nevertheless: Inserting sub-steps accommodates for time step reduction from time-explicit CSD solvers 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 Eulerian-Lagrangian-Coupling module (ELC)

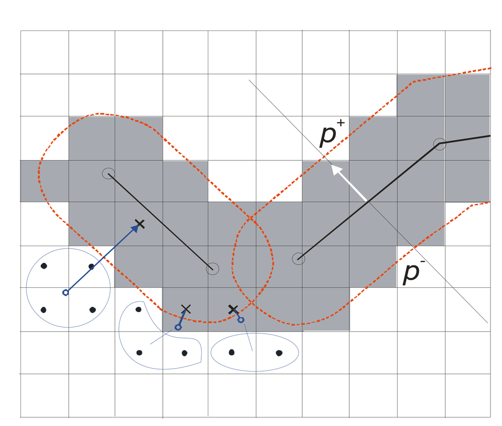


VTF software design

The VTF is constructed on top of the Amroc class hierarchy in a C++ framework approach.
- The design acknowledges the greater complexity on the Eulerian side in our coupling approach.
- Applications use one generic main program that instantiates predetermined objects.
- Objects can be extensively customized by C++ class derivation.
- Problem-specific routines (initial conditions, boundary conditions) are provided in F77/90 or C/C++.



UML class diagram for the VTF. Right half: most important classes in Cartesian Amroc. Left side: Extensions for implementing the generic ghost fluid method and coupling with a CSD solver.



Treatment of lower-dimensional thin-shells

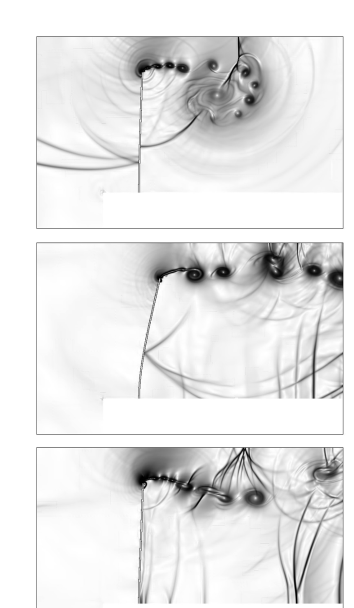
- Thin boundary structures require "thickening"
- Unsigned distance level set function ϕ
- Treat cells with $0<\phi<d$ as ghost fluid cells (gray)
- Leaving level set unmodified ensures correctness of $\nabla\phi$
- Refinement criterion based on ϕ ensures reliable mesh adaptation
- Use face normal in shell element to evaluate in $\Delta p = p^+ - p^-$

Shock-induced motion of a thin panel

- Verification and demo simulation
- Mach-1.21 inflow in air (γ=1.4, P_0=100kPa, 293K) impinges on thin steel panel of 1mm thickness and 50mm length

Fluid
- Two-dimensional 0.4m x 0.08m flow domain with forward facing step geometry, reflective boundaries everywhere except inflow on left side.
- AMR base mesh 320x64, 2 additional levels with factors 2, 4

Solid – 1d beam solver using Euler-Bernoulli theory
- 80 finite difference points in beam middle axis
- Panel situated 1.5cm behind start of step, elastic material
- 4 nodes 3.4 GHz Intel Xeon dual processor, Gigabit Ethernet network, ca. 54h CPU (7 fluid processors, 1 solid processor)



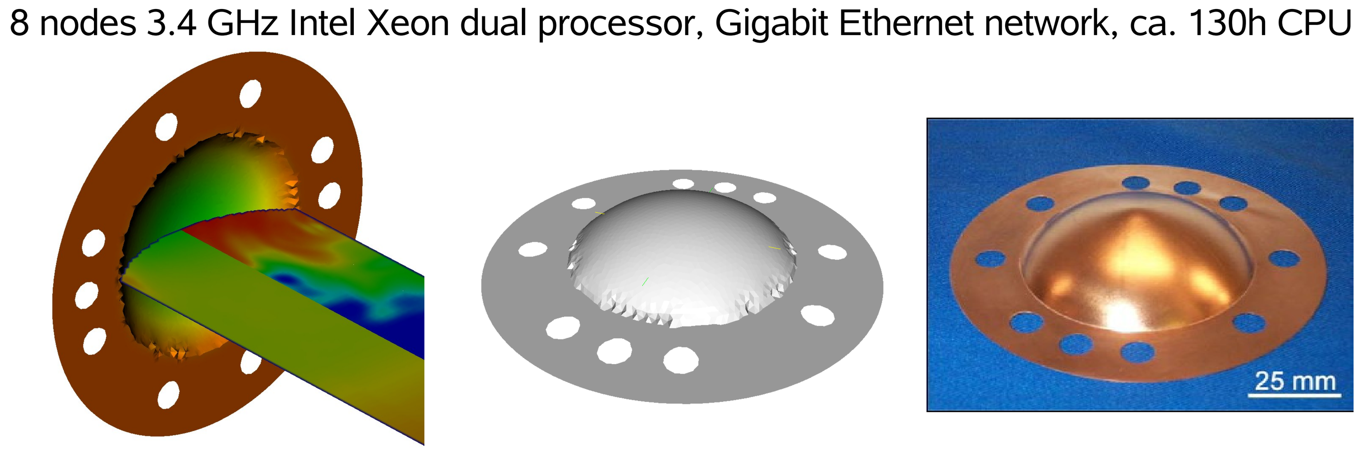
Right: Schlieren plot of fluid density in the vicinity of the thin steel panel. Snapshots for t=0.51ms, 1.63ms, and 3.02ms (from top to bottom). Vortex shedding from panel tip and beam vibration induced by shock impact.

Water-hammer-driven plate deformation

Piston-induced strong pressure wave in water shock tube impinges on thin copper plate.

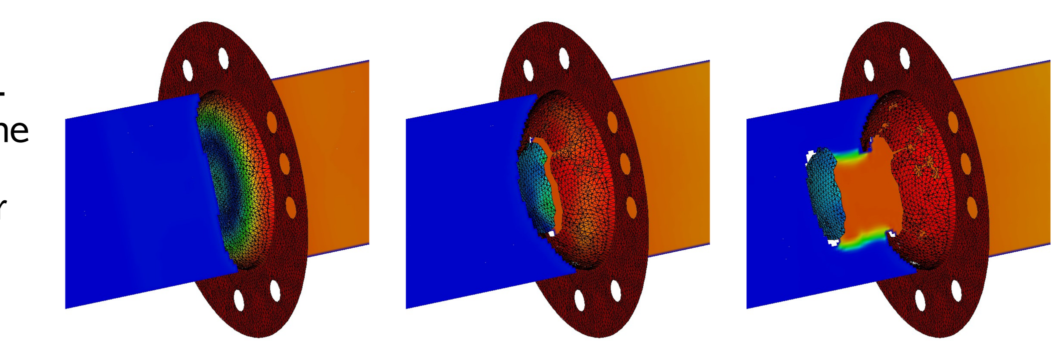
Fluid
- Correct pressure wave profile generated by propagating upper tube boundary according to equation of motion for piston
- Water shock tube of 1.3m length, 64mm diameter
- Two-component solver, modeling of water with stiffened gas EOS with γ=7.415, p_∞=296.2 MPa, air with ideal gas EOS with γ=1.4
- No tensile stresses allowed (cavitation threshold set to 0.0 Pa).
- AMR base level: 350x20x20, 2 additional levels, refinement factors 2, 2
- Approx. $1.2 \cdot 10^6$ cells used in fluid on average instead of $9 \cdot 10^6$ (uniform)

Solid - SFC
- Copper plate of 0.25mm thickness, J2 plasticity model with hardening, rate sensitivity, and thermal softening
- Shell mesh: 4675 nodes, 8896 elements
- 8 nodes 3.4 GHz Intel Xeon dual processor, Gigabit Ethernet network, ca. 130h CPU



Left: velocity in direction of tube axis in plate and fluid (lower half of plane) and fluid pressure (upper half of plane) 1.62ms after piston impact. Right: plate at end of simulation and after experiment.

Right: Preliminary simulation with plate fracture. The fluid density in the mid-plane visualizes the water splash. The solid mesh displays the velocity in direction of the tube axis.

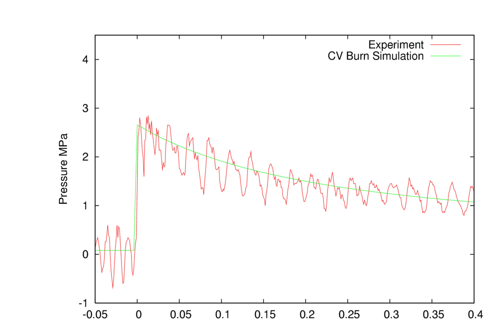


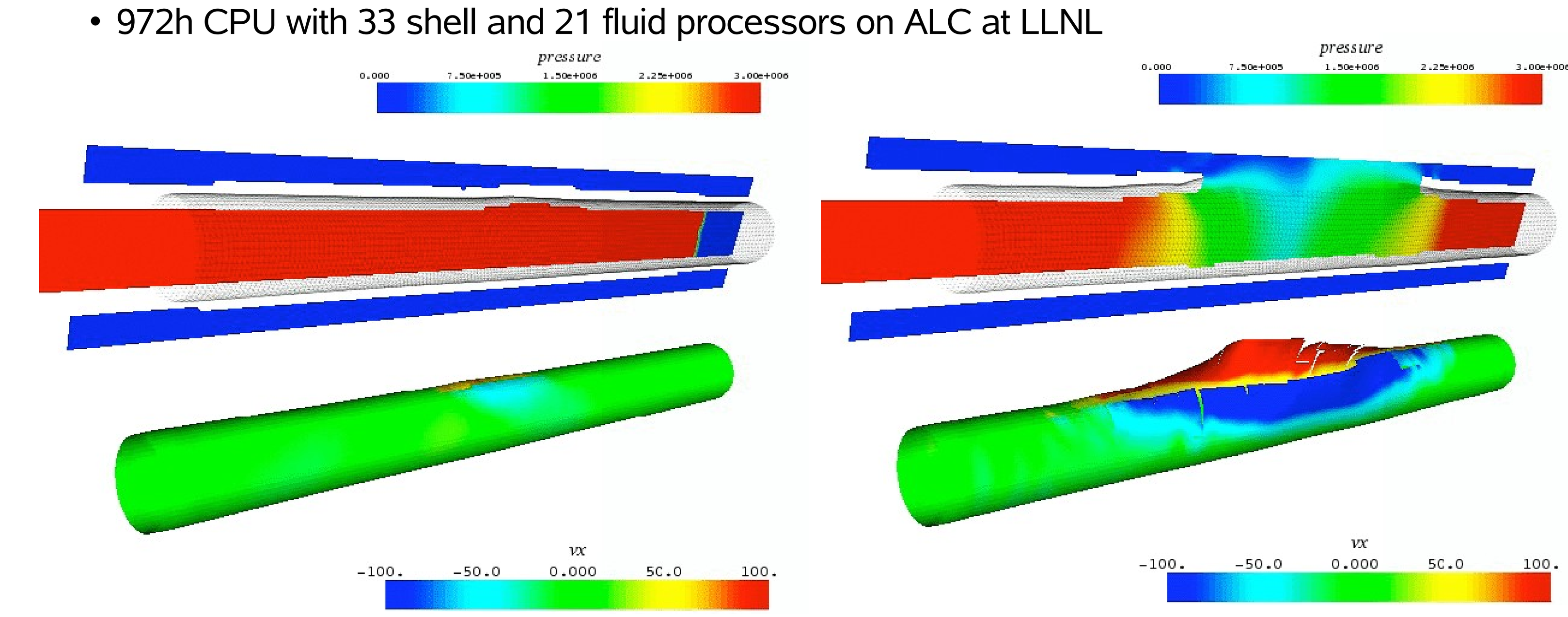
Detonation-driven fracture of thin aluminum tubes

Motivation: Validate VTF for complex fluid-structure interaction problem
- Interaction of detonation, ductile deformation, fracture

Fluid
- Constant volume burn model with γ=1.24, P_{CJ} =6.1MPa, D_{CJ}=2402m/s (Right: flow field before specimen)
- 40x40x725 cells unigrid

Solid - SFC
- Material model for cohesive interface: linearly decreasing envelope
- Shell mesh: 206,208 nodes
- 972h CPU with 33 shell and 21 fluid processors on ALC at LLNL





Conclusions. A flexible C++ framework for fluid-structure interaction has been developed. It uses components in Fortran 77/90 (fluid patch solvers) and C (bulk material models). The approach is tailored to high-speed events (shock waves, detonations) with large material deformation and strongly coupled response. Three-dimensional fluid-structure interaction simulations with arbitrary topology evolutions allowing for fracture and fragmentation are possible. Dynamic mesh adaptation can be used in the fluid to decrease the computational costs significantly. Scalable MPI parallelization allows the effective utilization of the ASC machines to achieve results in minimal real time. Further information and links to publications can be found under *http://www.cacr.caltech.edu/asc*.