Verification and Validation of the AMROC Fluid Solver Framework Coupling with DYNA3D within the Virtual Test Facility Fluid Structure Interaction Suite

Presented by:
Stephen Wood, DOE Fellow (Florida International University, Department of Mechanical and Materials Engineering)

Mentor:
Dr. Ralf Deiterding (Oak Ridge National Laboratory, Computer Science and Mathematics Division, Computational Mathematics Research Group)

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

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

Finite volume scheme

$$Q^{n+1}_{jk} = Q^n_{jk} - \frac{\Delta t}{\Delta x_1}\left(\mathcal{A}^- \Delta_{j+\frac{1}{2},k} + \mathcal{A}^+ \Delta_{j-\frac{1}{2},k}\right) - \frac{\Delta t}{\Delta x_2}\left(\mathcal{B}^- \Delta_{j,k+\frac{1}{2}} + \mathcal{B}^+ \Delta_{j,k-\frac{1}{2}}\right)$$

•AMROC employs a time-explicit finite volume scheme to compute inviscid compressible flows with strong shocks.

Solid Mechanics equations

Finite element scheme

- DYNA3D is based on a finite element discretization of the three spatial dimensions and a finite difference discretization of time
- DYNA3D uses a lumped mass formulation for efficiency. This produces a diagonal mass matrix $\boldsymbol{M}$, which renders the solution of the momentum equation

$$\boldsymbol{M}\boldsymbol{a}_{n+1} = \mathbf{f}^{ext} - \mathbf{f}^{int}$$

trivial at each step in that no simultaneous system of equations must be solved

- The basic continuum finite element in DYNA3D is the eight-node "brick" solid element. This element is valid for large displacements and large strains
- DYNA3D also fully supports truss, beam, and shell structural elements

Kinematic/isotropic plasticity Material Law

$$\phi = \bar{\sigma} - \sigma_y(\bar{\varepsilon}^p)$$

Young's modulus, E
Poisson's ratio, ν
Yield stress, σ_0
Tangent modulus, E_T
Hardening parameter, β

isotropic hardening

$$\bar{\sigma} = \left(\frac{3}{2}s_{ij}s_{ij}\right)^{\frac{1}{2}}$$

kinematic hardening

$$\bar{\sigma} = \left(\frac{3}{2}\eta_{ij}\eta_{ij}\right)^{\frac{1}{2}}$$

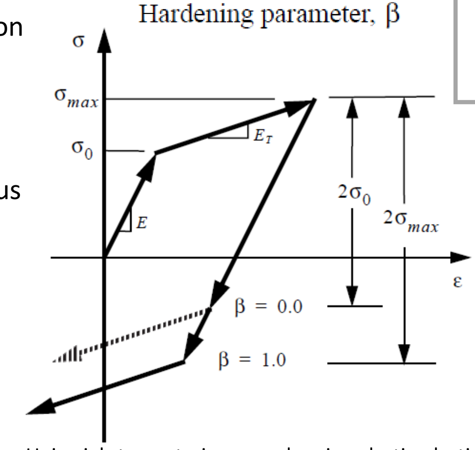
$$\eta_{ij} = s_{ij} - \alpha_{ij}$$

$$\sigma_y = \sigma_0 + \beta E_p \bar{\varepsilon}^p$$

$$\bar{\varepsilon}^p = \int_0^t d\bar{\varepsilon}^p$$

$$d\bar{\varepsilon}^p = \left(\frac{2}{3}d\bar{\varepsilon}^p_{ij}d\bar{\varepsilon}^p_{ij}\right)^{\frac{1}{2}}$$

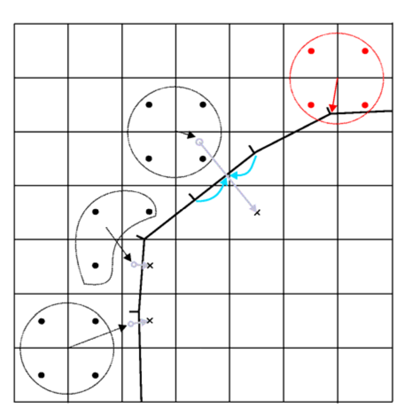
$$E_p = \frac{E E_T}{E - E_T}$$



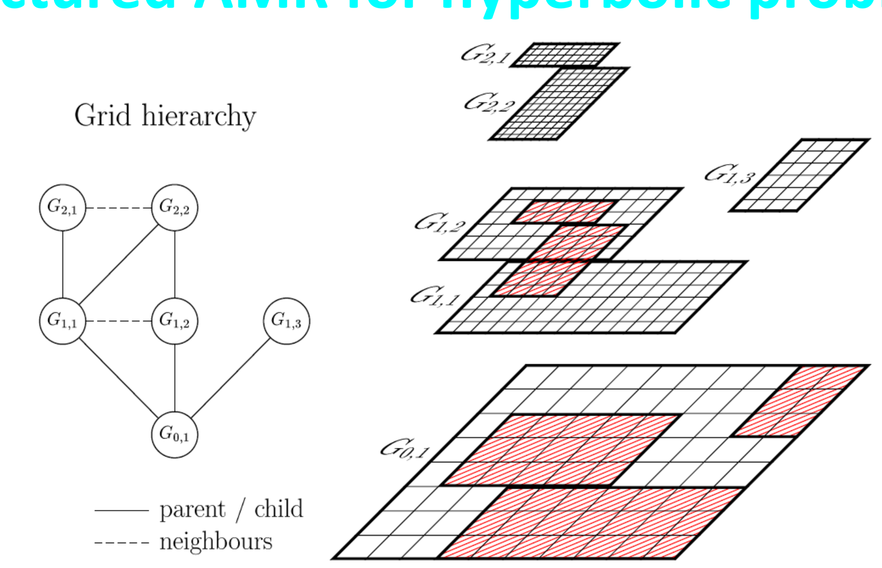
Uni-axial stress strain curve showing elastic-plastic material behavior for kinematic hardening (β = 0.0) and for isotropic hardening (β = 1.0) .

Fluid-structure coupling

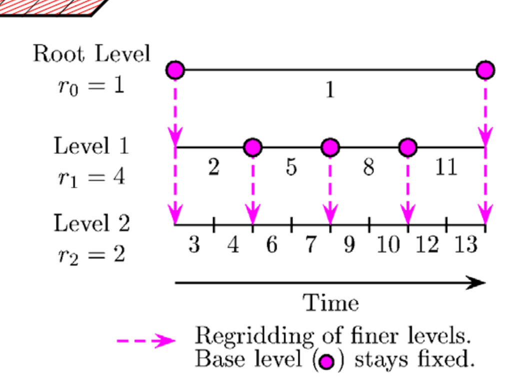
- Compatibility conditions between inviscid fluid and solid at a slip interface
  - *Continuity of normal velocity: $u^S_n = u^F_n$*
  - *Continuity of normal stresses: $\sigma^S_{nn} = -p^F$*
  - *No shear stresses: $\sigma^S_{n\tau} = \sigma^S_{n\omega} = 0$*
- Time-splitting approach for coupling
  - *Fluid:*
    - *Treats evolving solid surface with moving wall boundary conditions in fluid*
    - *Uses solid surface mesh to calculate fluid level set*
    - *Uses nearest velocity values $\boldsymbol{u}^S$ 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



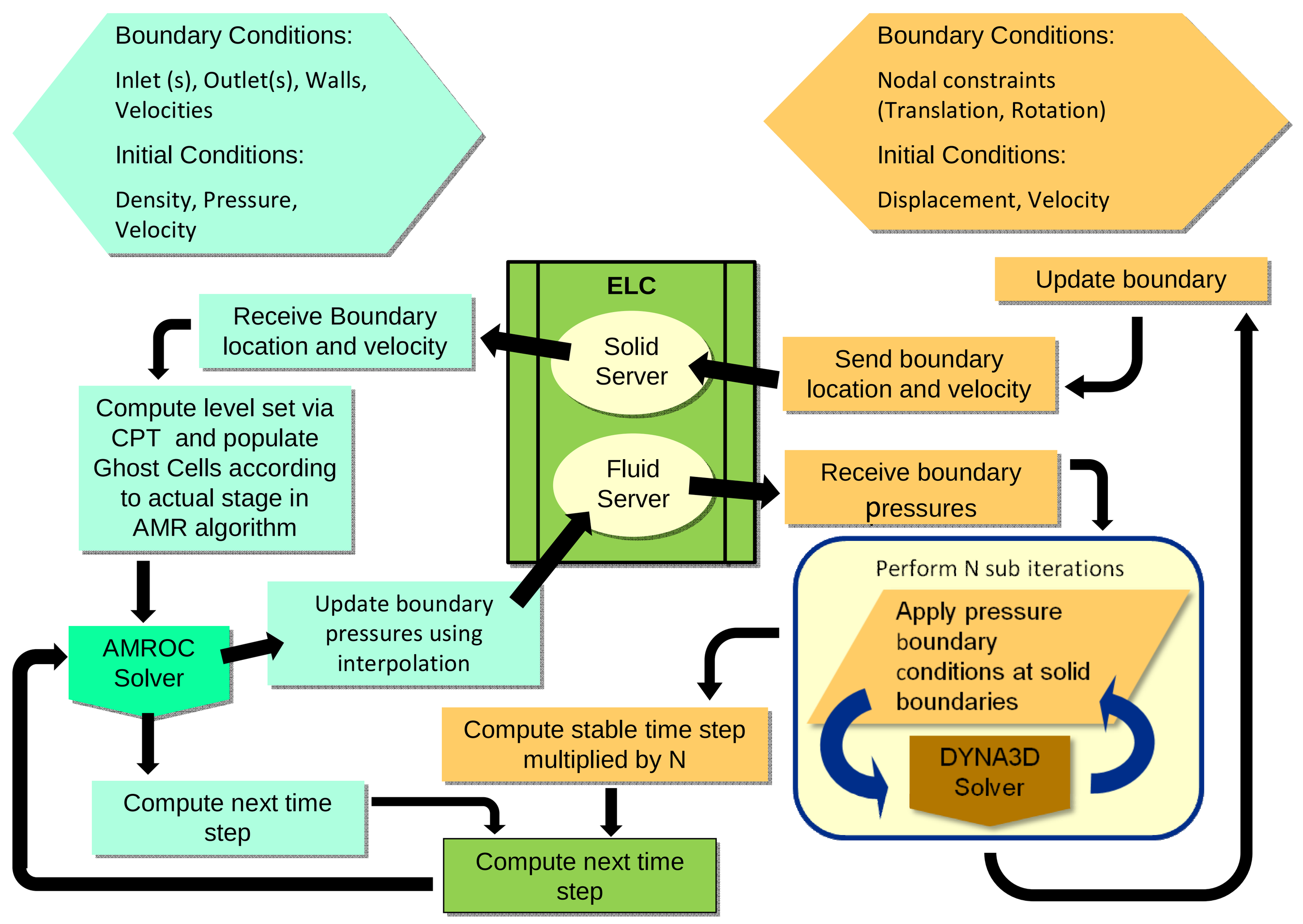
Structured AMR for hyperbolic problems



- 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
- Explicit finite volume scheme
- M. Berger and P. Colella, J. Comput. Phys. 82, 1988.



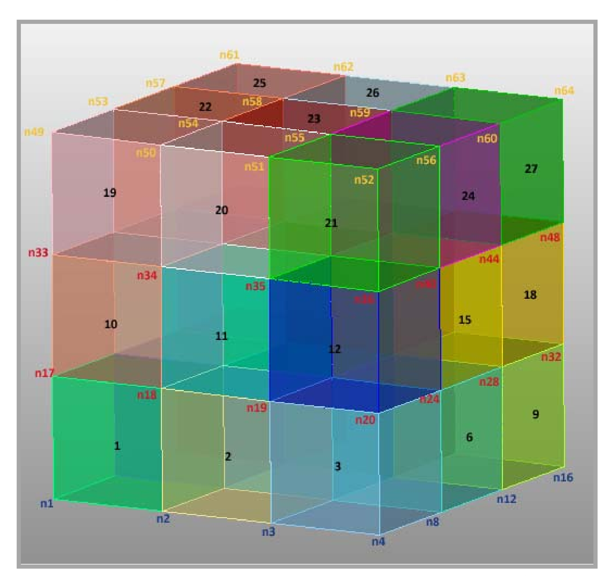
Algorithmic approach for coupling



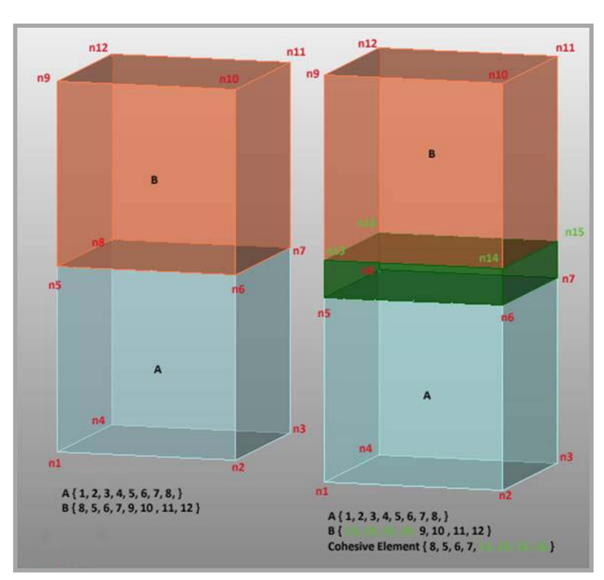
Routines to facilitate AMROC-DYNA

- Geometry Pre-processor routine
  - *Reads output from Cubit 11.0 Mesh Generator and dynamically creates a DYNA3D input file*
  - *Supports hexagonal, tetrahedral, thick shell , and thin shell elements*
  - *Translates node selection sets to apply boundary conditions, pressure loads on included faces, and generates cohesive elements for fracture simulation*

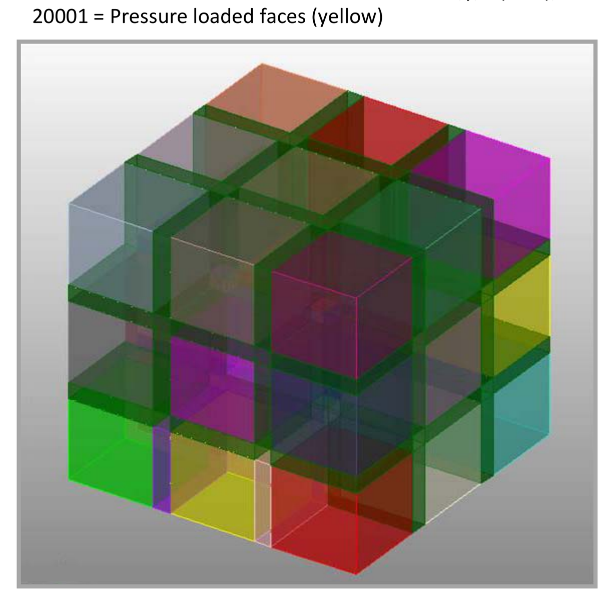
Node selection sets
•1 – 9999 : identify volumes where cohesive elements are to be generated to simulate possible fracture(s)
•10000 – 19999 : identify nodes to which translation and rotation nodal constraints will be applied
•20001 – 29999 : identify nodes on surfaces where pressure loads will be applied
Cohesive elements
•used to simulate cohesion or inter-laminar forces between "parallel" hex elements
•employ traction-displacement relationships to generate nodal forces based upon the projected displacements of the hex element corners in opening (mode I) and in plane shear (mode II) directions



Node selection sets: 1 = Cohesive element generation (all), 10700 = Nodal constraint: translation fixed in x,y,z (blue), 20001 = Pressure loaded faces (yellow)



Generation of new coincident nodes between two arbitrary elements to create cohesive elements prompts an update of elements and faces preserving nodal constraints and pressure loads. Thickness of cohesive element is a visual aide only.



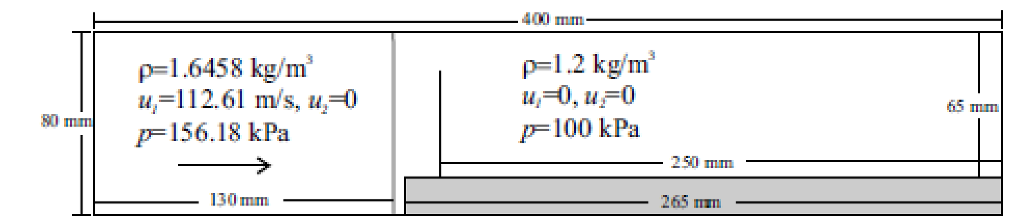
Generated cohesive elements within an arbitrary mesh of solid hexagonal elements to model possible fracture. Thickness of cohesive elements is a visual aide only.

- DYNA3D Post-processor routine
  - *Extracts data from DYNA3D data structures*
  - *Supports hexagonal and thin shell elements*
  - *Outputs displacements, velocities, and stresses in VTK format*

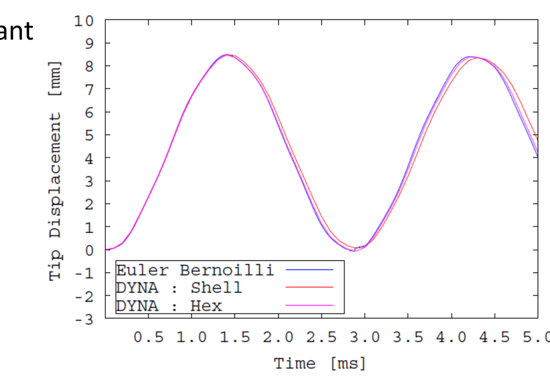
Verification Test Case: Shock-induced panel motion

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

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

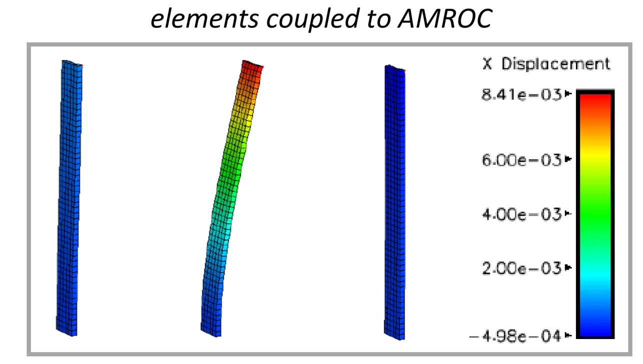


- DYNA3D implementation verification: constant impulsive loading of Δp=100kPa
  - *Euler-Bernoulli Beam Equation*
  - *DYNA : Shell: DYNA3D explicit finite element solver employing thin-shell elements*
  - *DYNA : Hex: DYNA3D explicit finite element solver employing hexagonal elements*

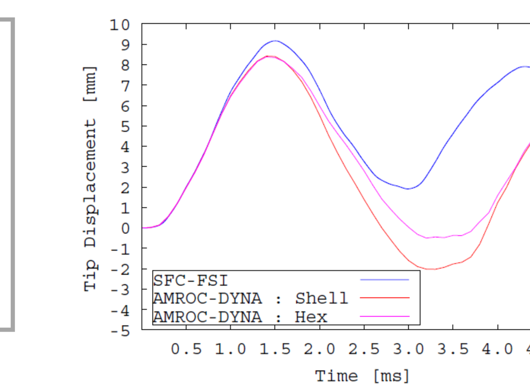


Panel motion – FSI verification

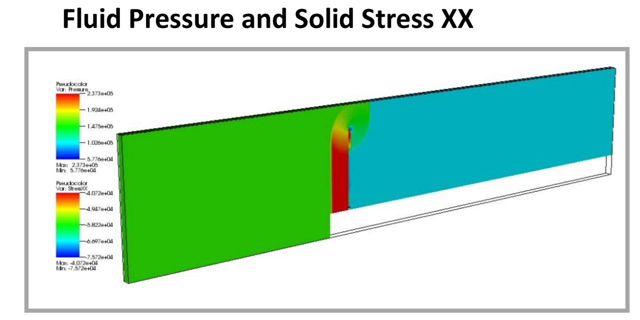
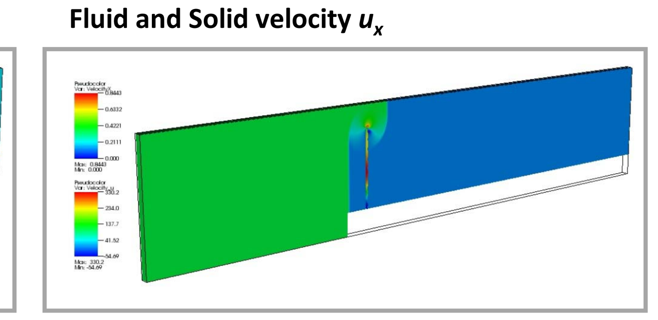
- Forward facing step geometry, reflective boundaries everywhere except inflow at left side, panel 1.5cm behind start of step
- SAMR base mesh 320x64(x2), 2 additional level with factors 2, 2
- Intel 3.4GHz Xeon dual processors connected with Gigabit Ethernet
  - *Beam-FSI: 12.25h CPU on 3 fluid CPU + 1 solid CPU*
  - *FEM-FSI: 322h CPU on 14 fluid CPU + 2 solid CPU*
  - *AMROC-DYNA: 15 fluid CPU + 1 solid CPU; Hex 600h CPU; Shell 450h CPU*
- FSI verification:
  - *SFC-FSI: large displacement thin-shell finite element solver by F.Cirak coupled to FV code*
  - *AMROC-DYNA : Shell: DYNA3D explicit finite element solver employing thin-shell elements coupled to AMROC*
  - *AMROC-DYNA : Hex: DYNA3D explicit finite element solver employing hexagonal elements coupled to AMROC*



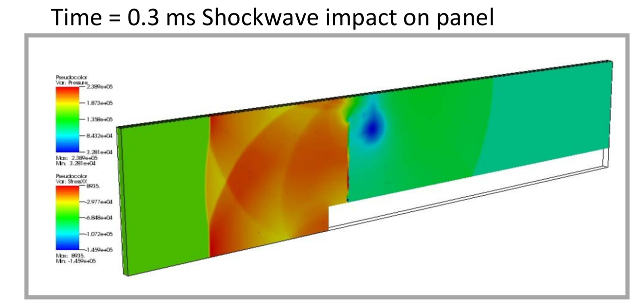
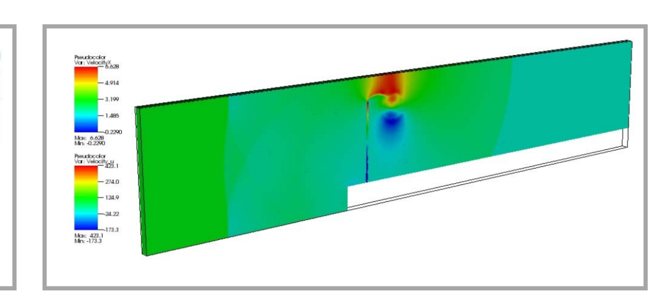
Hexagonal mesh of steel panel colored for x displacement at t= 0.3, 1.4, 3.2 ms



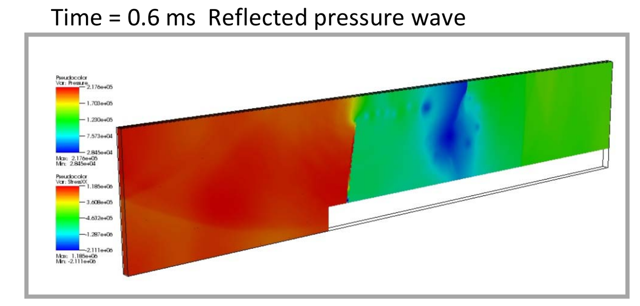
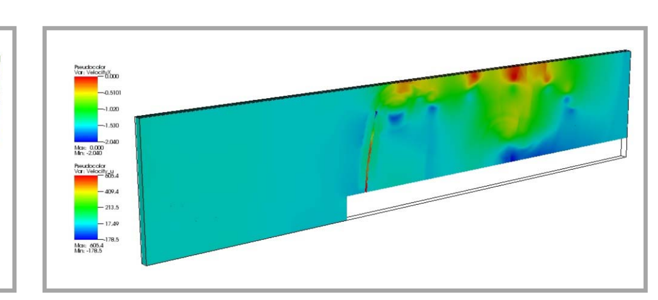
Fluid Pressure and Solid Stress XX | **Fluid and Solid velocity u_x**

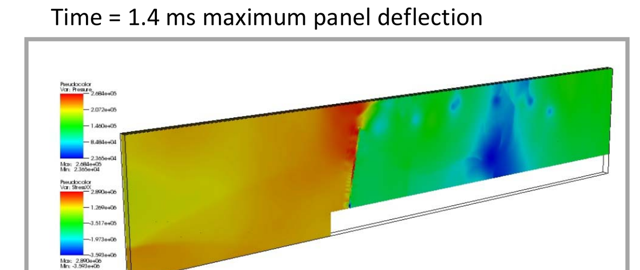
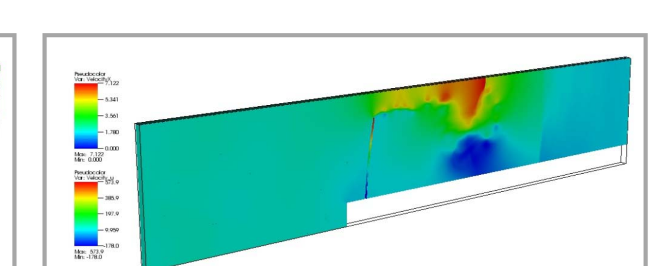
Time = 0.3 ms Shockwave impact on panel

Time = 0.6 ms Reflected pressure wave

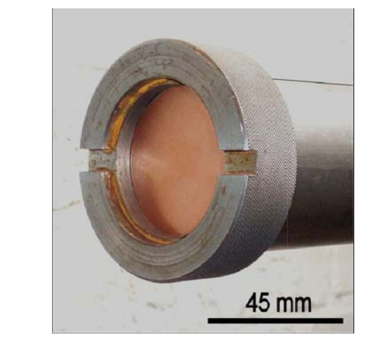
Time = 1.4 ms maximum panel deflection

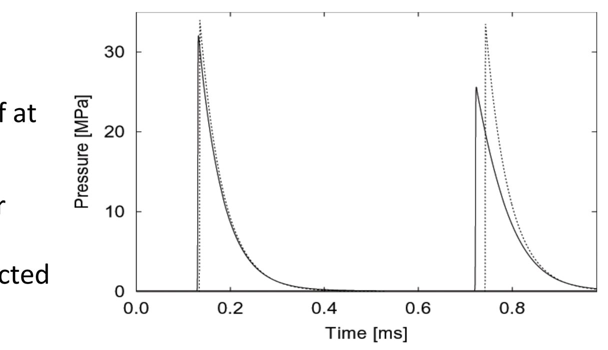



Time = 3.2 ms maximum panel rebound

Validation Test Case: Plate deformation from water hammer

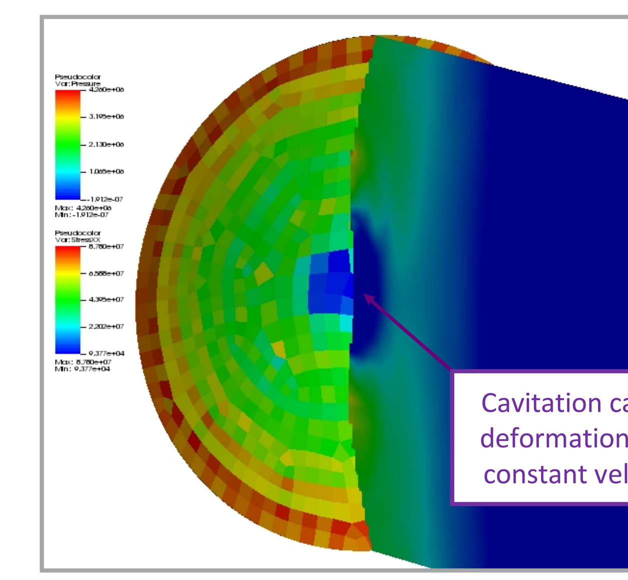
- 3d simulation of plastic deformation of thin copper plate attached to the end of a pipe due to water hammer
- Strong over-pressure wave in water is induced by rapid piston motion at end of tube
- Experiments from 'An underwater shock simulator', V.S. Deshpande et al., Proc. Royal Soc. A 462, 2006.
- Two-component model based on "stiffened" gas equation of state
- Computation uses γ^{Air}=1.4, p_1^{Air}=0, γ^{Water}=7.415, p_1^{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
- Intel 3.4GHz Xeon dual processors connected with Gigabit Ethernet
  - *SFC-FSI: 130h CPU on 8 nodes*
  - *AMROC-DYNA: 15 fluid CPU + 1 solid CPU; Hex 206h CPU; Shell 97h CPU*



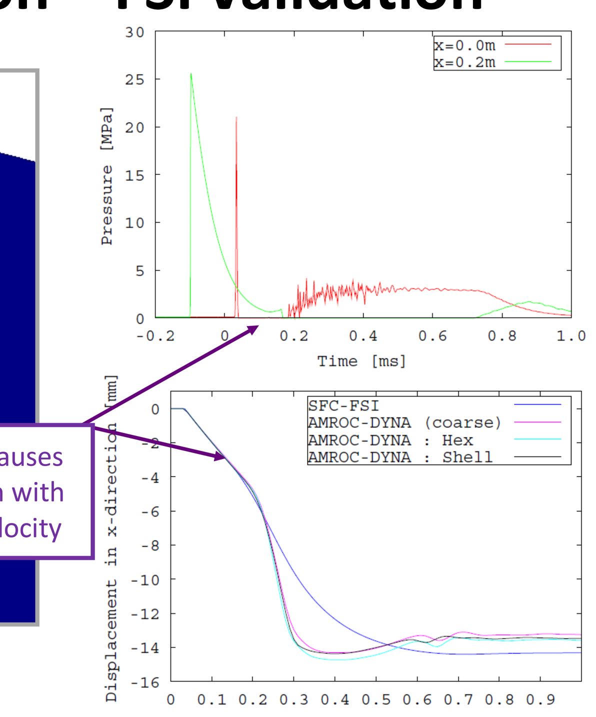


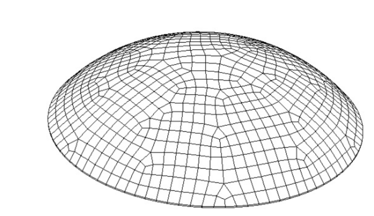
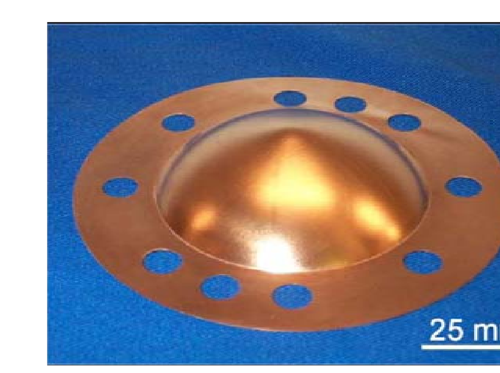
Comparison of the traveling wave approximation (dotted) with computed pressure traces (solid) at x_1 = 1.1 m (left) and x_1 = 0.2 m (right).

Plastic deformation – FSI validation



Fluid pressure distribution revealing cavitation and stressXX distribution within the copper plate at t = 0.15 ms

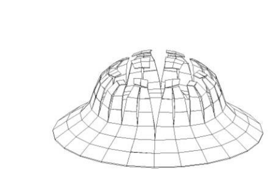


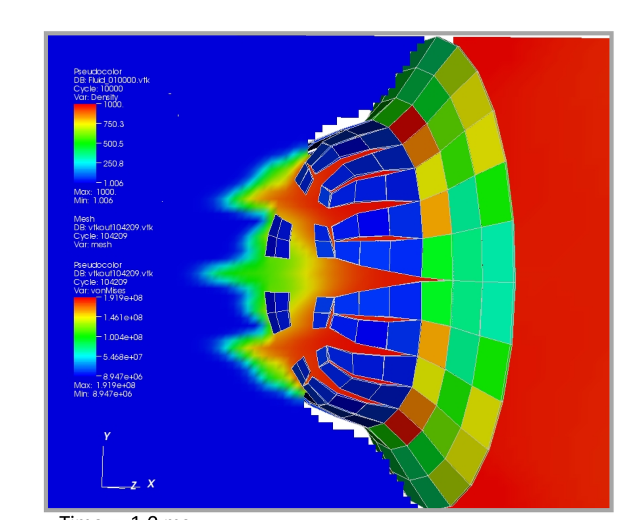



Comparison of plate at end of simulation and experiment show good agreement of shape and maximum deflection.

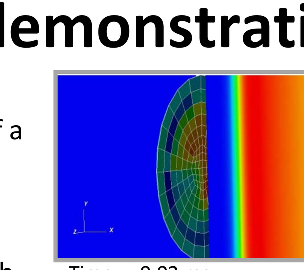
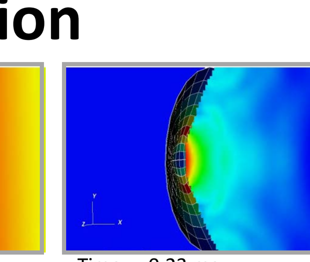
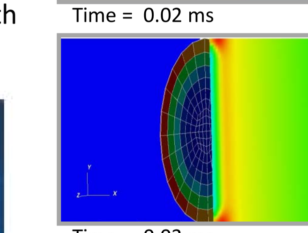
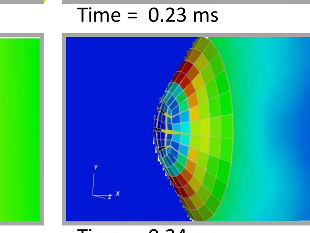
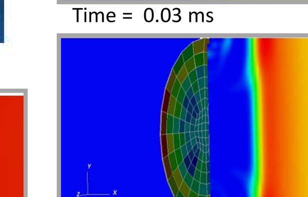
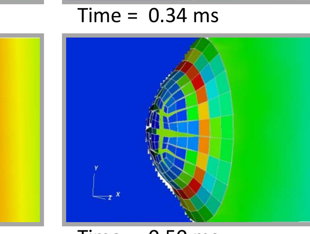
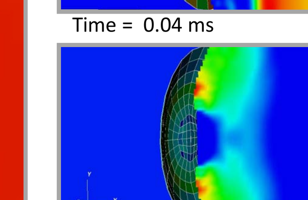
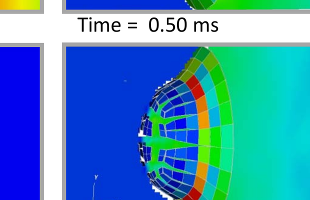
Fracture demonstration

- 3d simulation of plastic deformation of thin copper plate attached to the end of a pipe due to water hammer
- Modeled cohesive elements and sliding contact between solid elements
- Preliminary results show agreement with experimental results



Time = 1.0 ms


Time = 0.02 ms

Time = 0.23 ms

Time = 0.03 ms

Time = 0.34 ms

Time = 0.04 ms

Time = 0.50 ms

Time = 0.16 ms

Time = 0.75 ms

Left: fluid density and plate von Mises stress
Above: fluid pressure and plate von Mises stress from behind the plate in left column and from in front of the plate in the right column.

U.S. DEPARTMENT OF ENERGY
OAK RIDGE NATIONAL LABORATORY
HERE
HIGHER EDUCATION RESEARCH EXPERIENCES
ORISE
OAK RIDGE INSTITUTE FOR SCIENCE AND EDUCATION
OAK RIDGE National Laboratory
FIU Engineering & Computing
FLORIDA INTERNATIONAL UNIVERSITY