Novel Methods in LS-DYNA 980

CFD-Methods CESE and PFEM

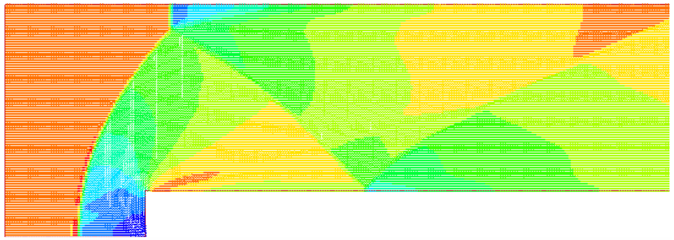
Dynamore GmbH
Industriestraße 2
D 70565 Stuttgart
http://www.dynamore.de

New CFD-capabilities in 980

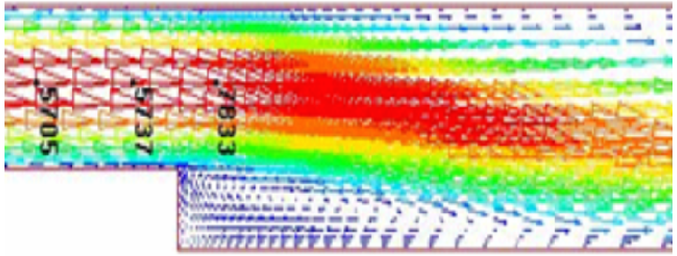
Compressible / Incompressible Flows

- Theoretical Basis are Navier-Stokes Equations
- Different numerical approaches necessary for
  - Compressible flow regime (aerodynamics, Mach number >0.3)
  - Incompressible flow regime (Mach number <0.3)

1. Compressible Flow Solver: CESE



2. Incompressible Flow Solver: PFEM



New CFD-capabilities in 980

Compressible / Incompressible Flows

1. Compressible Flow Applications:

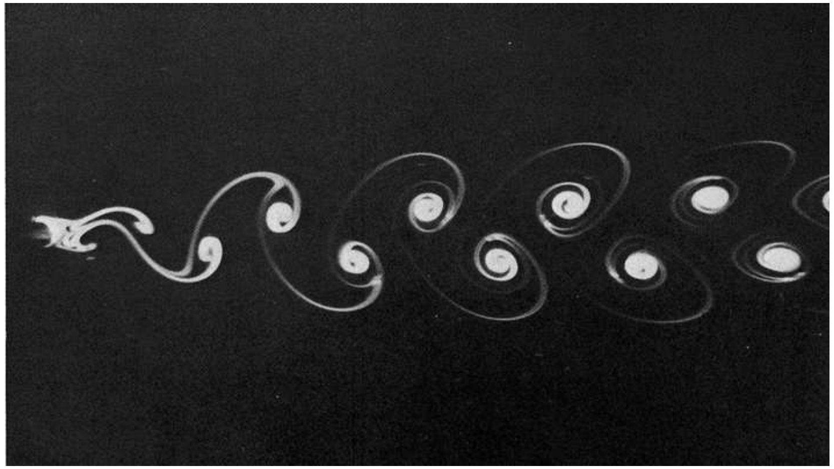
- External aerodynamics - transonic or supersonic flows (flow around airfoils)
- Pneumatic systems
- Flow in an Airbag
- Detonation waves
- Laminar/turbulent flows
- Pressure waves are dominant effect, inviscid calculations show pressure waves (airbag, shock waves etc.)

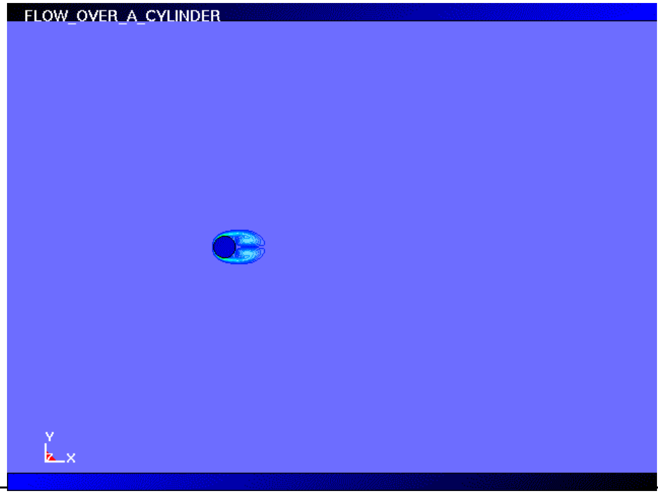
New CFD-capabilities in 980

Compressible / Incompressible Flows

2. Incompressible Flow Applications:

- External aerodynamics - subsonic flows (flow around cars, buildings, etc.)
- Hydraulics
- High viscosity flows
- Internal / External flows
- Due to smaller velocities, boundary layer effects have large influence (->no inviscid calculations)





Compressible Flow Solver

Numerical Method CESE

- **CESE** (Conservation Element & Solution Element)
- *Flux* conservations in *space and time* (locally & globally)
- Accuracy
  - 2^{nd} order (for flow variables & their spatial derivatives)
- Novel & simple shock-capturing strategy (transonic shock wave are dominating effect)
- Both strong shocks and small disturbances can be handled very well simultaneously
- Boundary conditions can be imposed easily & accurately

Compressible Flow Solver

Numerical Method CESE

CE – Conservation Element

SE – Solution Element

Chose integral form of conservation equations (CE):

$$\oint_{S(V)} h_m \cdot ds = 0, \qquad m=1, 2, 3, 4, 5$$

SE = FEM-solution in space and time

$$u^{*}(x, y, z, t)_{(i,j,k,n)} = u + u_x(x - x_i) + u_y(y - y_j) + u_z(z - z_k) + u_t(t - t^n)$$

Compressible Flow Solver

Numerical Method CESE

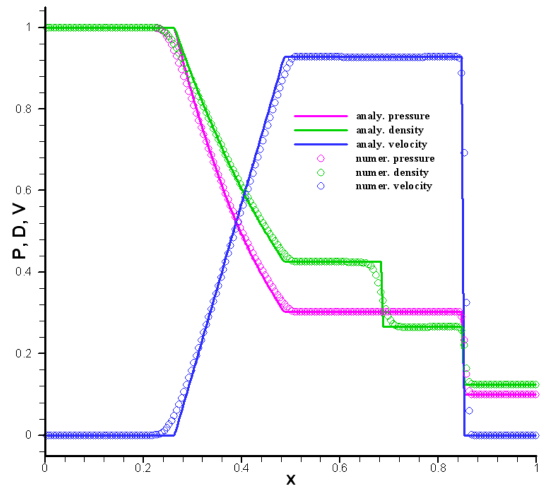
Time stepping scheme:

- Weighted forward/backward time difference:
  - Parameter automatically adjusted (smooth region/shock region)
- Numerical dissipation added only where necessary (shock region)

Compressible Flow Solver

Capabilities and application area

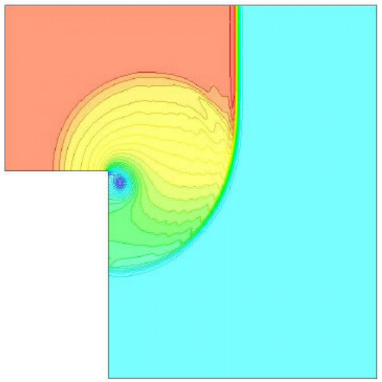
- All speed compressible flows
  - Subsonic, transonic, supersonic flows
  - Supersonic flows with complex shock patterns
- Acoustics
  - Pressure waves can accurately be resolved
  - Strong and small disturbances are resolved
- Chemical reaction flows
- Cavitating flows

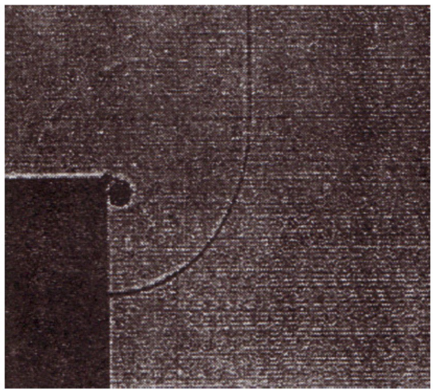


Compressible Flow Solver

Capabilities and application area

- Serial & MPP mode
- Flows: Inviscid & viscous flows
- Meshes: Hexahedra, wedges, tetrahedra or mixture
- Boundary Conditions:
  - Regular boundary conditions
  - solid, open, inflow, outflow, symmetric
  - Moving or rotating solid boundaries for viscous flows
  - 2D option

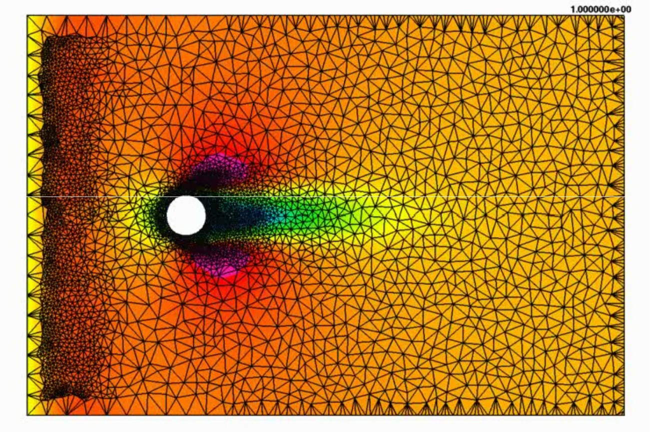


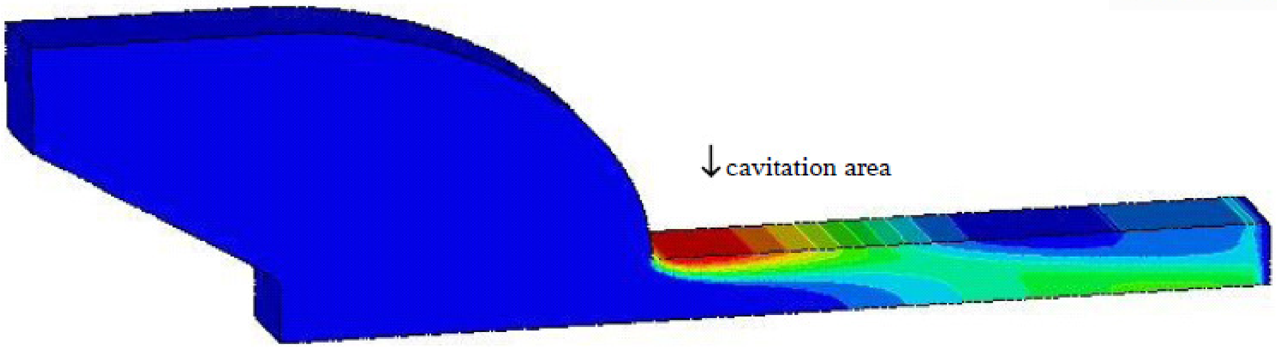


Compressible Flow Solver

Capabilities and application area

- Error Control and adaptive re-meshing MPP implementation.
- Multifluid and Free-Surface flows.
- LES and RANS turbulence models

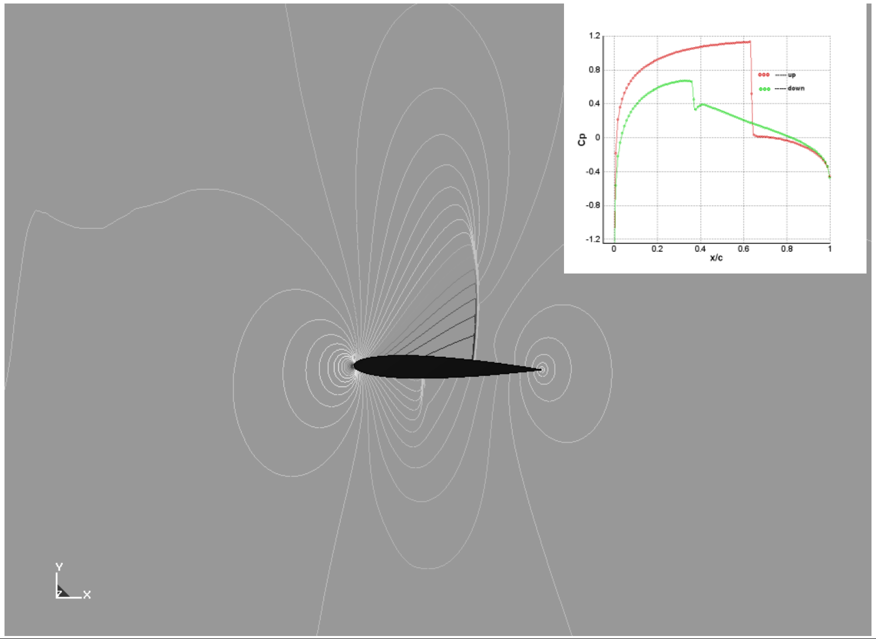




Compressible Flow Solver

Capabilities and application area

- Examples
  - Transonic Flows with sharp shocks
  - Mesh Adaptivity improves sharp shock resolution
  - Multiple fluids (droplet)



Incompressible Flow Solver

Numerical Method PFEM – Particle Finite Element Method

Particle Method – “Meshless” Finite Element Method MFEM, similar to EFG

1. Discretize domains with a finite element mesh (Delaunay). Particles are mesh nodes
2. Identify external boundaries (free surfaces)
3. Compute the state variables velocities, pressure and viscous stresses
4. Move mesh nodes to a new position (Lagrangian Step)
5. Generate a new mesh if needed. The mesh regeneration process can take place (automated remeshing process)

Incompressible Flow Solver

Numerical Method PFEM – Particle Finite Element Method

Scientific background

- Based on work of Onate, Idelson, Del Pin
- Extended Delaunay Algorithm – fast remeshing (every time step)
- MFEM approach is done for solids+fluids
- Free surface+Fluid-Structure-Interaction naturally included

Incompressible Flow Solver

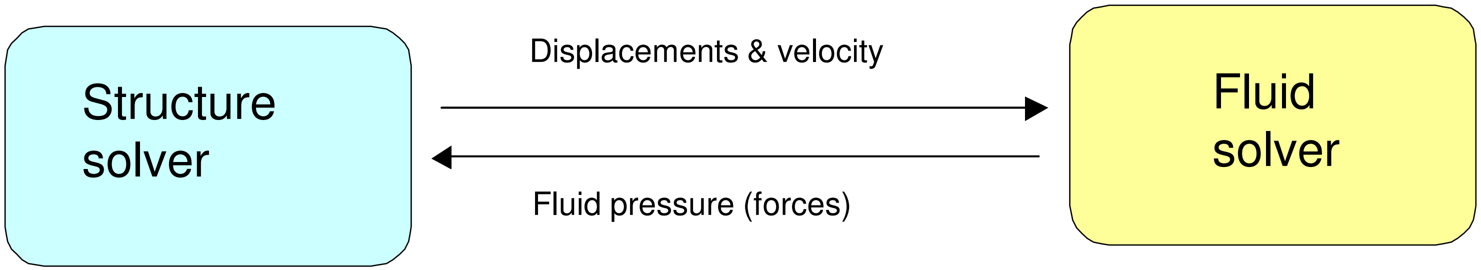
Capabilities and application area

- Laminar+turbulent flows
  - Smagorinsky LES
  - K-ε-Turbulenz-Modell
- Conjugate heat transfer
  - Solve fluid-thermal problem coupling
  - Heat transfer from fluid to solid can be investigated
- Multiple fluids
- Boundary layer meshing for external aerodynamics

Fluid-Structure-Interaction

Coupling algorithm

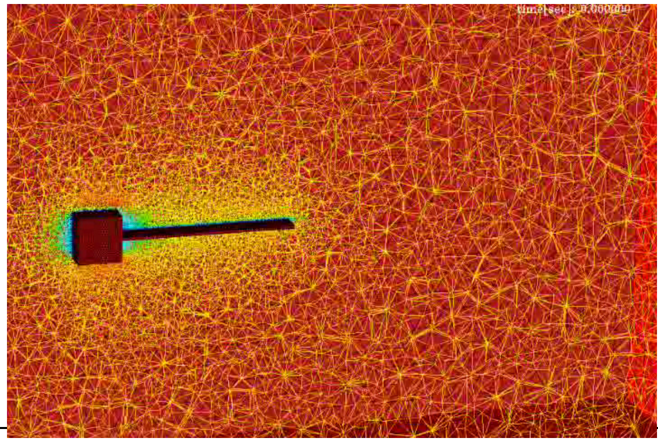
- Structure and fluid solvers
  - Structure solver —— FEM (Lagrangian)
  - Fluid solver —— CESE / PFEM



Fluid-Structure-Interaction

Coupling algorithm

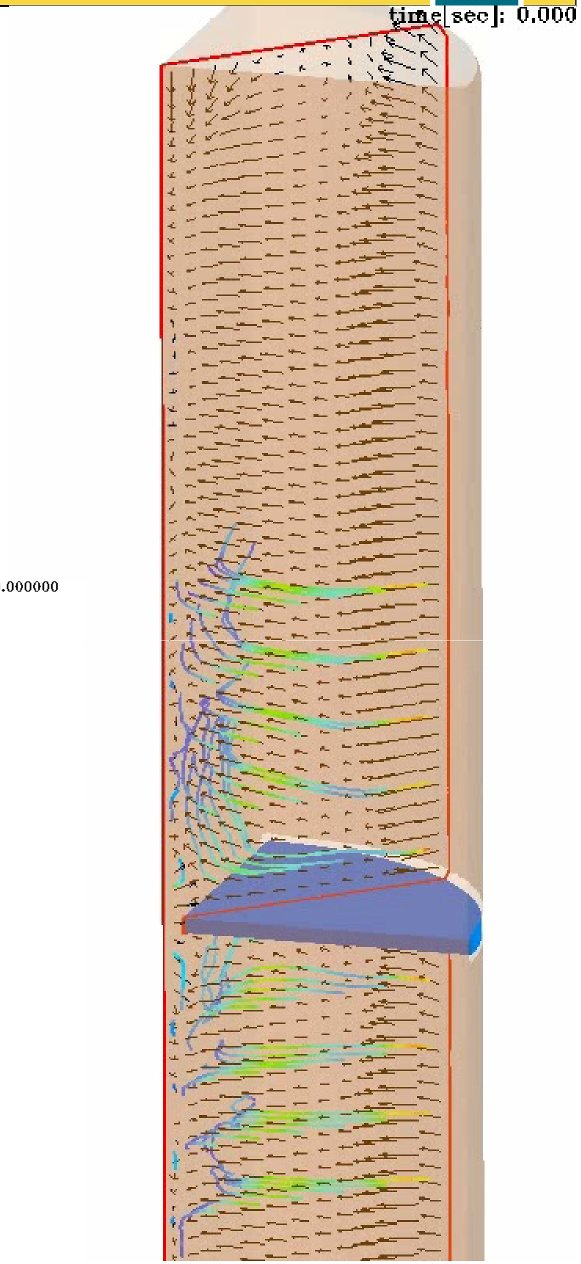
- structures can be shell and/or solid volume elements
- Fluid mesh is independent of the structures
- For some applications (e.g. airbag), users have the option to only calculate the inside of the bag or both sides (using the same fluid material or different ones)
- Allows weak and strong FSI coupling depending upon the problem.
- Coupling to explicit and implicit structural solvers



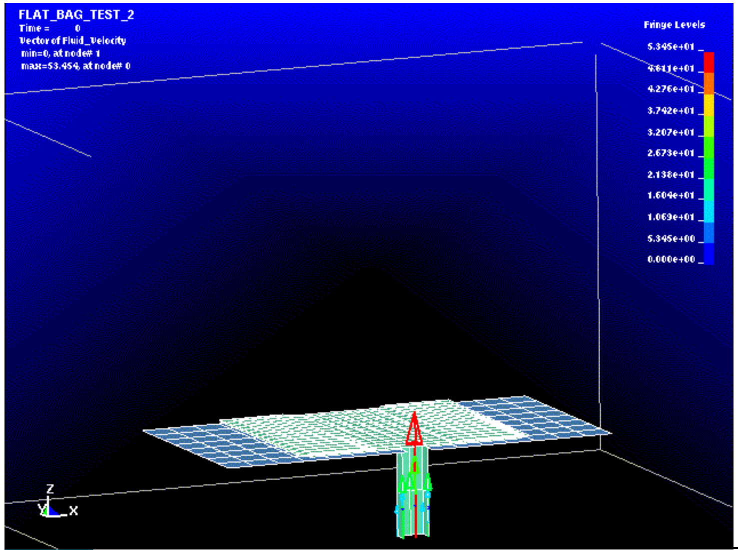
Fluid-Structure-Interaction

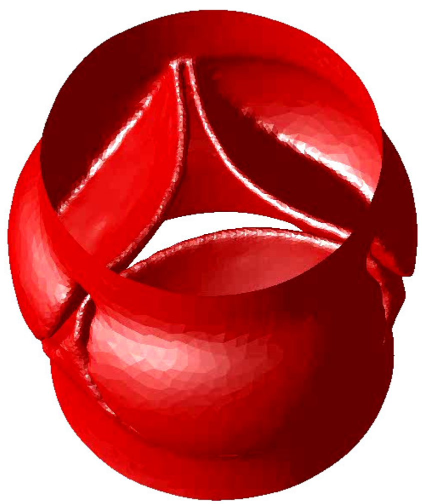
Coupling example

- Opening of a vessel
- Heart valve
- Airbag deflation



time[sec]: 0.000000





Thank you for your attention!

Your DYNA distributor and *more*