Coupling incompressible and compressible two-phase flow solvers in a numerical wave tank

Based on OpenFOAM & FOAM-Extend projects

Pedro Martínez Ferrer, Ling Qian, Derek Causon, Clive Mingham and Zhihua Ma

CMMFA Research Group,
Manchester Metropolitan University

Streatham Court Theatre,
Exeter,
2016/04/19

Involves:

- Coupling of different specialized solvers: potential, incompressible, compressible...
- Dynamic and overset meshes to accommodate 6DoF floating body motions.
- FSI when the solid deforms.
- Collaborative environment with developers working in different aspects of the wave tank: SoFT project (EPSRC).
The OpenFOAM-based solver *wsFoam* (Virtual Wave Structure Interaction) combines:

- *fnlpFoam* (fully non-linear potential flow solver),
- *interFoam* (incompressible VOF solver),
- *compressibleInterFoam* (compressible VOF),
- a new set of *interface* boundary conditions,
- ...

- Uses a partitioned approach to solve different regions.
- Interface is *implicitly* treated as another face connecting two cells from different regions: two-way coupling!
- Couples regions associated to similar or different solvers.
- Performance is on a par with native solvers: ready to HPC!
Solvers: system of equations

- Incompressible and **compressible** formulations are displayed.
- Mass equations:
  \[
  \frac{\partial \alpha}{\partial t} + \nabla \cdot \mathbf{U}_\alpha + \nabla \cdot \mathbf{U}_c \alpha (1 - \alpha) = -\frac{\alpha}{\rho_w} \frac{D \rho_w}{Dt},
  \]
  \[
  \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0,
  \]
  with \(\rho_w = \rho_{w,0} + \frac{p}{R_w T}\), \(\rho_a = \frac{p}{R_a T}\) (instead of \(\rho_a = \rho_{a,0}\)).

- Similar momentum equation for both solvers
  \[
  \frac{\partial \rho \mathbf{U}}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) - \nabla \cdot (\mu_{\text{eff}} \nabla \mathbf{U}) = \sigma \kappa \nabla \alpha - \mathbf{g} \cdot \mathbf{x} \nabla \rho - \nabla p_d.
  \]
- An energy equation is used by the compressible solver:
  \[
  \frac{\partial \rho T}{\partial t} + \nabla \cdot (\rho \mathbf{U} T) - \Delta (\mu_{\text{eff}} T) = - \left( \frac{\alpha}{c_{v,w}} + \frac{1 - \alpha}{c_{v,a}} \right) \left( \frac{\partial \rho k}{\partial t} + \nabla \cdot (\rho \mathbf{U} k) + \nabla \cdot (\mathbf{U} p) \right).
  \]
- A Poisson equation is derived from mass and momentum equations to solve for the pressure and update velocity.
Coupling interface: boundary conditions

Specify the patch value:

\[ \phi_f = \frac{1}{|d_p - d_N|} (\phi_P |d_N| + \phi_N |d_P|) \]

and the normal gradient patch value:

\[ S_f \cdot \nabla \phi_f = \left( |S_f| \frac{\phi_N - \phi_P}{|d|} \right) \left( \frac{S_f \cdot d}{|S_f||d|} \right), \]
Details of the implementation

wsFiFoam iterative procedure

FOR EACH ITERATION IN THE MAIN LOOP, DO
  1 Calculate the time steps of the I regions
  2 Calculate the time steps of the C regions
  3 Find the global (minimum) time step
  FOR EACH I REGION, DO
    FOR EACH I TRANSPORT EQUATION, DO
      1 Linearize
      2 Apply boundary and interface conditions
      3 Solve
      END I TRANSPORT EQUATION
    END I REGION
  FOR EACH C REGION, DO
    FOR EACH C TRANSPORT EQUATION, DO
      1 Linearize
      2 Apply boundary and interface conditions
      3 Solve
      END C TRANSPORT EQUATION
    END C REGION
  4 Update the simulation time
END MAIN LOOP
**wsiFoam directory structure**

Schematic view of the *wsiFoam* directory (1 of 2).
Details of the implementation

wsiFoam directory structure

Schematic view of the wsiFoam directory (2 of 2).
Validation case: “dam break” (Martin & Moyce, 1952)

Dam break of water simulated with wsiFoam using interFoam (left) and compressibleInterFoam (right).
Validation case: “dam break” (Martin & Moyce, 1952)

Normalized water front position (left) and water column height (right) evolution.

Validation case: “dam break”

Table: Water dam break benchmark: simulation speed up for a mesh of $400 \times 120$ cells; times are normalized by $t_{\text{ref}} = 262.93 \text{s}$.

<table>
<thead>
<tr>
<th>Cores</th>
<th>I</th>
<th>C</th>
<th>I-C</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.25</td>
<td>1.00</td>
<td>1.10</td>
</tr>
<tr>
<td>2</td>
<td>2.10</td>
<td>1.66</td>
<td>1.83</td>
</tr>
<tr>
<td>4</td>
<td>3.18</td>
<td>2.52</td>
<td>2.80</td>
</tr>
</tbody>
</table>

Validation case: “water column drop” (ISOPE, 2010)

Water column drop simulated with one region using `interFoam` (left) and with two regions using `interFoam` and `compressibleInterFoam` (right).
Validation case: “water column drop” (ISOPE, 2010)

Time history of the impact pressure at the centre of the tank floor compared against published results and other OpenFOAM reference solutions.

Coupling FNLP solvers with NS solvers

Linear waves simulated with \texttt{wsiFoam} using \texttt{fnlpFoam} (left) and \texttt{interFoam} (right).
Open-source overset strategy

Oscillating floating box simulated with dynamic mesh (left) and overset mesh (right).
Conclusions

- wsiFoam has been developed according to professional software standards under the SoFT (EPSRC) project: git, doxygen, wiki website, etc.
- Interface boundary conditions have been validated and the results have been published: Martínez Ferrer et al. (2016).
- Coupling between FNLP & NS solvers is being currently investigated.
- Overset strategies are currently being developed.
- FSI to be integrated to account for hydroelasticity effects.
- We aim to keep our contributions open-source to enhance collaboration with the scientific community under the CCP-WSI (EPSRC) project.
Thank you for your attention!

Find us on:

1. Twitter: @CMMFA
2. Facebook: CMMFA
3. Web: http://www.scmdt.mmu.ac.uk/cmmfa/