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

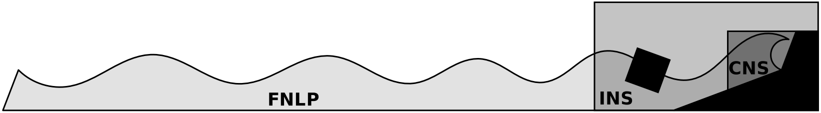
Based on OpenFOAM & FOAM-Extend projects

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

The construction of a numerical wave tank



Virtual Wave Structure Interaction (WSI) Simulation Environment.

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).

`wsiFoam` code development

- The OpenFOAM-based solver `wsiFoam` (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:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot \boldsymbol{U}\alpha + \nabla \cdot \boldsymbol{U}_{\mathrm{c}}\alpha(1-\alpha) = -\frac{\alpha}{\rho_{\mathrm{w}}}\frac{\mathrm{D}\rho_{\mathrm{w}}}{\mathrm{D}t},$$

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{U}) = 0,$$

with $\rho_{\mathrm{w}} = \rho_{\mathrm{w},0} + p/R_{\mathrm{w}}T$, $\rho_{\mathrm{a}} = p/R_{\mathrm{a}}T$ (instead of $\rho_{\mathrm{a}} = \rho_{\mathrm{a},0}$).

- Similar momentum equation for both solvers

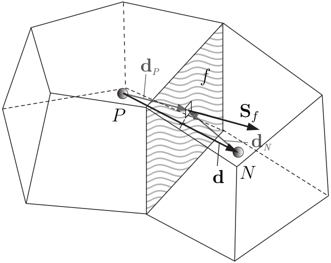
$$\frac{\partial \rho \boldsymbol{U}}{\partial t} + \nabla \cdot (\rho \boldsymbol{U}\boldsymbol{U}) - \nabla \cdot (\mu_{\mathrm{eff}} \nabla \boldsymbol{U}) = \sigma\kappa\nabla\alpha - \boldsymbol{g}\cdot\boldsymbol{x}\nabla\rho - \nabla p_{\mathrm{d}}.$$

- An energy equation is used by the compressible solver:

$$\frac{\partial \rho T}{\partial t} + \nabla \cdot (\rho \boldsymbol{U} T) - \Delta(\mu_{\mathrm{eff}} T) = -\left(\frac{\alpha}{c_{v,\mathrm{w}}} + \frac{1-\alpha}{c_{v,\mathrm{a}}}\right)\left(\frac{\partial \rho k}{\partial t} + \nabla \cdot (\rho \boldsymbol{U} k) + \nabla \cdot (\boldsymbol{U} p)\right).$$

- 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:

$$\phi_{\mathrm{f}} = \frac{1}{|\mathbf{d}_P - \mathbf{d}_N|} \left(\phi_P |\mathbf{d}_N| + \phi_N |\mathbf{d}_P| \right)$$

and the normal gradient patch value:

$$\mathbf{S}_{\mathrm{f}} \cdot \nabla \phi_{\mathrm{f}} = \left(|\mathbf{S}_{\mathrm{f}}| \frac{\phi_N - \phi_P}{|\mathbf{d}|} \right) \left(\frac{\mathbf{S}_{\mathrm{f}} \cdot \mathbf{d}}{|\mathbf{S}_{\mathrm{f}}||\mathbf{d}|} \right),$$

wsiFoam 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

```
wsiFoam
├── applications
│   └── solvers
│       └── wsiFoam
│           ├── compressibleInterFoam
│           │   └── twoPhaseMixtureThermo
│           ├── interFoam
│           ├── multiRegion
│           └── numericalBeach
└── src
```

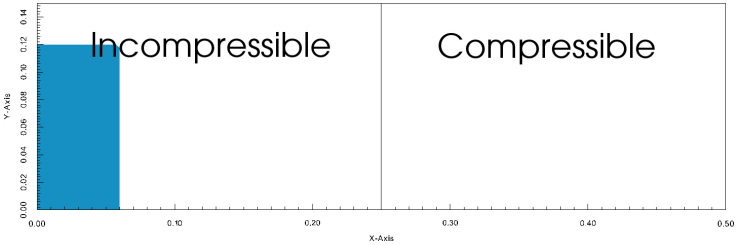
Schematic view of the wsiFoam directory (1 of 2).

wsiFoam directory structure

```
└── src
    ├── finiteVolume
    │   ├── cfdTools
    │   │   └── general
    │   │       └── solutionControl
    │   │           ├── pimpleControl
    │   │           └── solutionControl
    │   └── fields
    │       └── fvPatchFields
    │           ├── basic
    │           │   └── mixedCoupled
    │           └── derived
    │               ├── linearWave
    │               ├── linearWaveVelocity
    │               └── numericalBeachFvPatchField
    ├── regionModels
    │   └── regionModel
    │       ├── regionModel
    │       ├── regionModelFunctionObject
    │       └── regionProperties
    ├── thermophysicalModels
    │   └── basic
    │       ├── basicThermo
    │       ├── fluidThermo
    │       ├── psiThermo
    │       │   └── basicPsiThermo
    │       └── rhoThermo
    │           └── basicRhoThermo
    ├── transportModels
    │   └── twoPhaseMixture
    │       └── twoPhaseMixture
    └── turbulenceModels
        ├── compressible
        │   └── turbulenceModel
        │       └── laminar
        └── incompressible
            └── turbulenceModel
                └── laminar
```

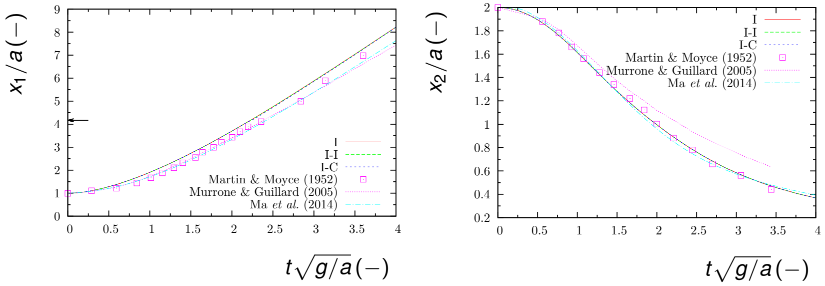
Schematic view of the wsiFoam directory (2 of 2).

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.

*P. J. Martínez Ferrer, D. M. Causon, L. Qian, C. G. Mingham, Z. H. Ma. A multi-region coupling scheme for compressible and incompressible flow solvers for two-phase flow in a numerical wave tank. *Computers & Fluids* **125**, 116–129 (2016).

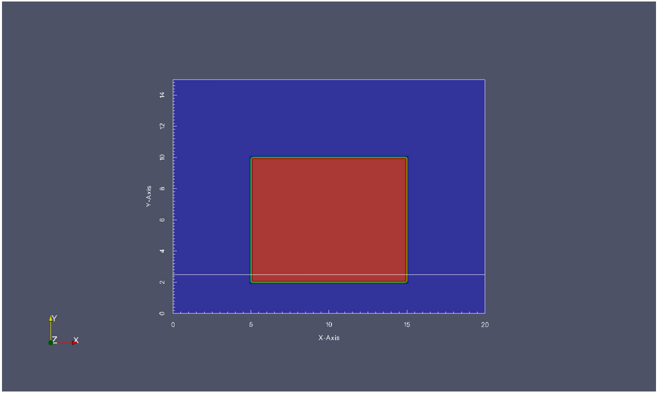
Validation case: “dam break”

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

Cores	I	C	I-C
1	1.25	1.00	1.10
2	2.10	1.66	1.83
4	3.18	2.52	2.80

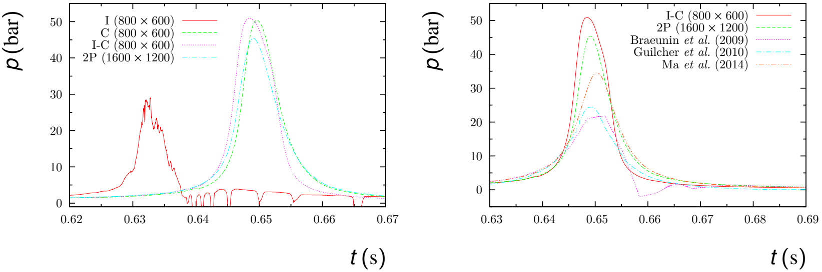
*P. J. Martínez Ferrer, D. M. Causon, L. Qian, C. G. Mingham, Z. H. Ma. A multi-region coupling scheme for compressible and incompressible flow solvers for two-phase flow in a numerical wave tank. *Computers & Fluids* **125**, 116–129 (2016).

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).

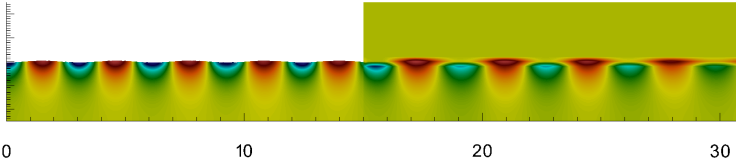
Validation case: “water column drop” (ISOPE, 2010)



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

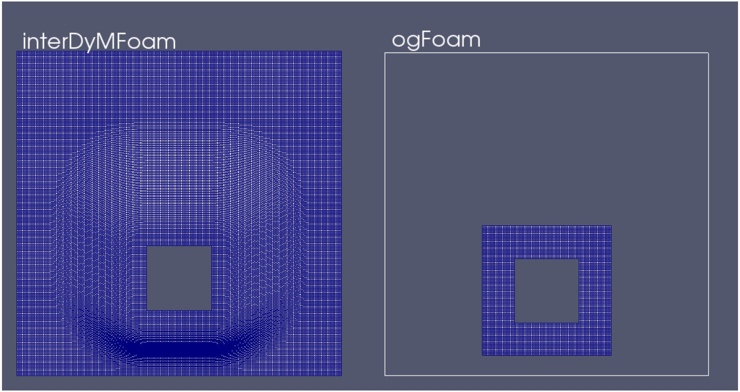
*P. J. Martínez Ferrer, D. M. Causon, L. Qian, C. G. Mingham, Z. H. Ma. A multi-region coupling scheme for compressible and incompressible flow solvers for two-phase flow in a numerical wave tank. *Computers & Fluids* **125**, 116–129 (2016).

Coupling FNLP solvers with NS solvers



Linear waves simulated with `wsiFoam` using `fnlpFoam` (left) and `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

Thank you for your attention!

Find us on:

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