Documentation for evapVOFHardt solver

TECHNISCHE UNIVERSITÄT DARMSTADT

TTD-CSI OpenFOAM database
Authors: Christian Kunkelmann and Stefan Batzdorf

1 Brief description of the solver

The solver **evapVOFHardt** permits to simulate liquid-vapor flows with phase change. The two-phase flow is treated by a volume-of-fluid approach and the phase change is modeled according to Hardt/Wondra (JCP 2008). Some simulation results on boiling flows performed with this solver (combined with an additional model for contact line evaporation) are presented in Kunkelmann/Stephan (NHT, 2009).

2 Detailed information

2.1 OpenFOAM basis and version

The solver is based on OpenFOAM's **interDyMFoam** solver (incompressible two phase flow with dynamic mesh) in OpenFOAM version 2.1.0.

2.2 Major modifications and extensions of basis

The major changes are

- Implementation of energy equation (in file TEqn.H)
- Implementation of phase change (in file calcSourceTerms.H)
- Smooth calculation of interface curvature (in file surfaceTension.H)

2.3 Description of input parameters and added variables

Additional definitions in **transportProperties** file:

- **cp:** specific heat capacity (for each phase)
- **k:** thermal conductivity (for each phase)
- **hEvap:** latent heat of vaporization
- **Rph:** interfacial heat resistance
- **Tsat:** saturation temperature
- **DPsi:** smearing factor for source term distribution (square of smearing length which should be the size of some cells)
- **DAlpha:** smearing factor for curvature calculation (square of smearing length which should be the size of some cells)

Additional variables:

- **alphaS** smeared VOF field (choose zeroGradient BCs)
- **T:** temperature field
- **psi0:** source terms (choose zeroGradient BCs)
- **rhoSource:** mass source terms (output only, no initial conditions required)

3 Related references

- S. Hardt, F. Wondra, 2008. Evaporation model for interfacial flows based on a continuum-field representation of source terms, Journal of Computational Physics, vol. 227, pp. 5871-5895.
- C. Kunkelmann, P Stephan, 2009. CFD simulation of boiling flows using the volume-of-fluid method within OpenFOAM, Numerical Heat Transfer A, vol. 56, pp. 631-646.

4 Description of the example case

The example case is the simulation of a water droplet that rebounces from a hot wall (Leidenfrost phenomenon). It is recommended to run the case on four processors. After compiling the solver **evapVOFHardt** and the initialization utility **initField** the following steps have to be carried out:

- create mesh with **blockMesh**
- initialize droplet with **../initField/initField**
- decompose computational domain (total mesh size is 80,000 cells) with **decomposePar**
- run solver with **mpirun -np 4 ../evapVOFHardt/evapVOFHardt -parallel > log**

On four processors (3.17 GHz), the case runs approximately 2.5 hours for one rebouncing cycle.

5 Usage

The solver has been developed by Christian Kunkelmann and Stefan Batzdorf (batzdorf@ttd.tu-darmstadt.de) at the Institute of Technical Thermodynamics, Technische Universtät Darmstadt, Alarich-Weiss-Straße 10, D-64287 Darmstadt.

Please feel free to use, modify and extend the code according to your requirements. We would only like to ask you to refer to the above mentioned papers.