Brief Description of the Multiphase OpenFOAM Solver cavitatingFoam

22 November 2017

1 Introduction

CavitatingFoam, developed by Kärrholm, is a multiphase solver within the OpenFOAM library that bases itself on the Homogeneous Equation Model (HEM).[1, 7, 5, 3] The solver employs a one-fluid approach with the assumption that homogeneous mixing occurs along the interface between the liquid and vapour phases that is described by a volume fraction term. The single-phase code includes a variable density that can be depend on pressure and/or temperature; cavitatingFoam assumes constant temperature so compressibility is assumed to be barotropic. Thus, a barotropic equilibrium equation of state is used to describe the mixtures fluid properties which can be modelled in a number of ways. Non-linear models such as the Wallis equation can be used to obtain the liquid and vapour compressibilies, however a linear compressbility model is typically implemented in cavitatingFoam due to its stability even in high velocity flows.[6, 7] The solver assumes friction-free compressible flow so that momentum equations are solved for velocity only, excluding density. The continuity and momentum equations divergence terms can be treated by variety of available interpolation techniques such as the Gauss upwind numerical scheme, better suited for high velocity flows, or a total variation diminishing scheme called MUSCLE.[6, 7, 4, 5] Eventhough minor developments have been made to the solver and the OpenFOAM library, the fundmental methodology of cavitatingFoam has remained consistent.

2 Numerical Method

The following numerical method is the derivation described by Kärrholm and has been since validated as good cavitation solver for high velocity flow investigations:[6, 7, 3, 2, 4, 5]

cavitatingFoam bases itself on a non-equilibrium differential barotropic equation of state (EoS):

$$\frac{D\rho}{Dt} = \psi \frac{Dp}{Dt}$$

Linear equations of states are used so that consistency is achieved for the liquid and vapour states at the limits (pure liquid or vapour) and mixture states:

$$\rho_v = \psi_v p$$
$$\rho_l = \rho_l^0 + \psi_l p$$

Before the inital iteration begins, one of three barotropic compressbility models available in OpenFOAM is established from the file thermoproperties.H: the three available models are the linear compressibility model, the Wallis equation, and the Chung equation.

The iteration process intiates by solving the continuity equation for ρ.

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

The initial density is used to determine the preliminary values for the volume fraction α and the mixture compressibility ψ. The α term is determined with the mixture density and material saturation density properties:

$$\alpha = \frac{\rho - \rho_{l,sat}}{\rho_{v,sat} - \rho_{l,sat}}$$

The mixture's equilibrium EoS can be defined as:

$$\rho = (1-\alpha)\rho_l^0 + (\alpha\psi_v + (1-\alpha)\psi_l)p_{sat} + \psi\alpha(p - p_{sat})$$

A linear compressibility model is commonly used for VoF investigations as viscosity and mass fraction are also determined linearly. The linear model is given by:

$$\psi = \alpha\psi_v + (1-\alpha)\psi_l$$

when a linear compressibility model is used, the mixture's equilibrium EoS can be simplified to:

$$\rho = (1-\alpha)\rho_l^0 + \psi p$$

The momentum equations are established to determine the matrices used to calculate the pressure-free velocity before a PISO loop is initiated. The momentum equations can be defined as:

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U U) = -\nabla p + \nabla(\mu_f \nabla U)$$

The continuity equation is transformed into a pressure equation before being solved in conjuction with the momentum equations in a PISO loop and is given by:

$$\frac{\partial \psi p}{\partial t} - (\rho_l^0 \frac{\partial \alpha}{\partial t} + (\psi_l - \psi_v)p_{sat})\frac{\partial \alpha}{\partial t} - p_{sat}\frac{\partial \psi}{\partial t} + \nabla\cdot(\rho U) = 0$$

The PISO loop will only cease once continuity has been reached where the remaining fluid properties α, ρ and ψ are updated according to the mixtures EoS. Due to the nature of the given model, the time step is limited by the Courant number so that convergence is reached. The Courant number is defined as:

$$Co = max(\frac{|U|}{\Delta x})\Delta t$$

Although it has less of an influence determining the time step, the acounstic Courant number is also considered and is defined by:

$$Co_{acoustic} = max(\frac{1}{\sqrt{\psi}\Delta x})\Delta t$$

References

[1] B. Biçer. Implementation of transport model into cavitatingfoam to simulate the cavitation in diesel injector nozzle. 2014.

[2] A. M. K. P. Taylor D. Pearce, K. Vogiatzaki and Y. Hardalupas. Investigation of the effects of domain representation and boundary condition selection in numerical simulations of micro scale flows with phase change. In *ILASSEurope 2017, 28th Conference on Liquid Atomization and Spray Systems*, September 2017.

[3] Y. Hardalupas D. Pearce and A. M. K. P. Taylor. Near nozzle field conditions in diesel fuel injector testing. *SAE Int.*, 2015.

[4] M. D. Roselló F. J. Salvador, J. V. Romero and J. Martínez-López. Validation of a code for modeling cavitation phenomena in diesel injector nozzles. *Mathematical and Computer Modelling*, 52:1123–1132, 2010.

[5] R. Novella F. J. Salvador, S. Hoyas and J. Martínez-López. Numerical simulation and extended validation of two-phase compressible flow in diesel injector nozzles. *Proc. IMechE*, 225, 2010.

[6] H. Weller F. P. Kärrholm and N. Nordin. Modelling injector flow including cavitation effects for diesel applications. In *5th Joint ASME/JSME Fluids Engineering Conference*, August 2007.

[7] F. P. Kärrholm. *Numerical Modelling of Diesel Spray Injection, Turbulence Interaction and Combustion.* PhD thesis, Chalmers University of Technology, 2008.