Validating the compressible multiphase solver FIRE® and a comparison of off-the-shelf Rayleigh-Plesset based cavitation models

CaFE Innovative Training Networks

HORIZON2020 Marie Skłodowska-Curie actions

Abstract

This poster presents a validation of the compressibility capabilities of the FIRE® multiphase solver and a comparison between four Rayleigh-Plesset based cavitation models.
A liquid/gas shock tube is used as benchmark test case for capturing the pressure waves.
The cavitation models formulations are first presented analytically and then tested on a throttle flow. The numerical simulation results are compared with an experimental visualization.

Introduction

The nozzle flow plays a crucial role for the engine performances and emissions. Today's trend is to increase the injection pressure to obtain a better atomization and finer droplets for improved combustion. This leads to very high velocities in the nozzle. As direct consequence the Mach number reaches values that makes compressibility not negligible. Furthermore the pressure may locally drop below the saturation value, causing cavitation. Flowing to higher pressure regions, the vapor bubbles collapse, thereby initiating shock waves travelling in the liquid, that might cause surface damages and erosion.
Considering these unique flow features, the development of methodologies able to predict the internal flow in fuel injectors has increased significantly in recent years. Numerical methods based on computational fluid dynamics have been considerably improved in these directions [1].

R-P cavitation models

Table 1. R-P cavitation models summary

	Γ_c	$\lvert\dot{R}\rvert$
FIRE linear	$\rho_d \; N \; 4\pi R^2 \lvert\dot{R}\rvert$	$\sqrt{\frac{2}{3}\frac{\lvert\Delta p\rvert}{\rho_c}}$
FIRE nonlinear	$\rho_d \; N \; 4\pi R^2 \lvert\dot{R}\rvert$	$\sqrt{\frac{2}{3}\left\lvert\frac{\Delta p}{\rho_c} - R\ddot{R}\right\rvert}$
Schnerr & Sauer	$\frac{\rho_d \rho_c}{\rho_m} \alpha_c \; N \; 4\pi R^2 \lvert\dot{R}\rvert$	$\sqrt{\frac{2}{3}\frac{\lvert\Delta p\rvert}{\rho_c}}$
Singhal et al.	$\frac{\rho_d \rho_c}{\rho_m} \alpha_c \, \rho_c \frac{\sqrt{k}}{\sigma} \lvert\dot{R}\rvert$	$\sqrt{\frac{2}{3}\frac{\lvert\Delta p\rvert}{\rho_c}}$

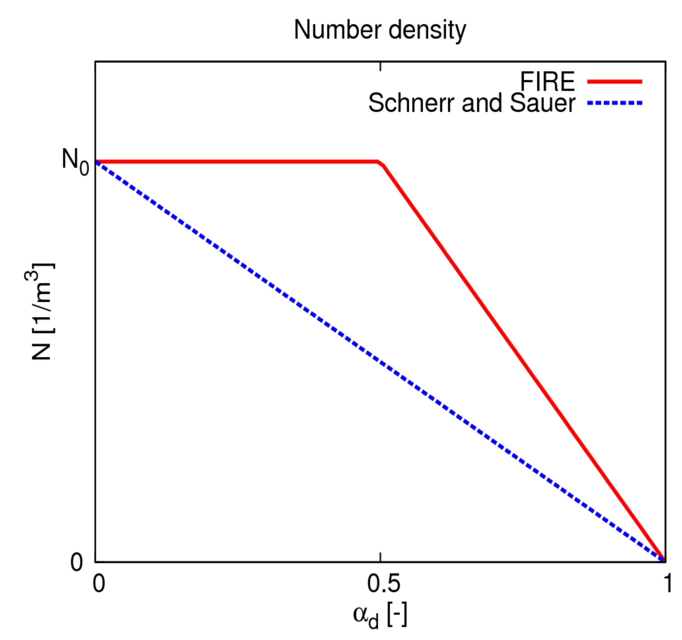

Figure 3. Number density approximation

Local pressure values below the liquid saturation pressure cause cavitation. This phenomenon can be modeled considering the single spherical bubble dynamic equation, known as Rayleigh-Plesset equation. ***Table 1*** shows a summary of some of these cavitation models available in the literature. They approximate the mass transfer rate between liquid and vapor with the averaged bubble mass change rate, multiplied by the number of bubbles.
The *FIRE linear* [6], *FIRE nonlinear* [6], and *Schnerr and Sauer* [7] models require a relation between bubble number density, N [1/m^3], and vapor volume fraction, α_d [-], as presented in ***Fig. 3***.
Singhal et al. model [8] is instead based on the relative velocity between vapor and liquid phases, approximated with the turbulent kinetic energy.

Figure 4 shows a comparison of the vapor cloud due to cavitation in the throttle flow presented in [9]. Experimental transmission images (up) are compared with numerical iso-surface plots of the vapor volume faction at 0.5 [-] (low). The channel dimensions are 0.3x0.295x1 [mm], and the driving pressures are 300 [bar] inlet and 120 [bar] outlet. Numerical simulations used the RANS *k-ε-ζ-f* turbulence model with a time step of 1 10^{-7} [s] on a 300,000 cells mesh.

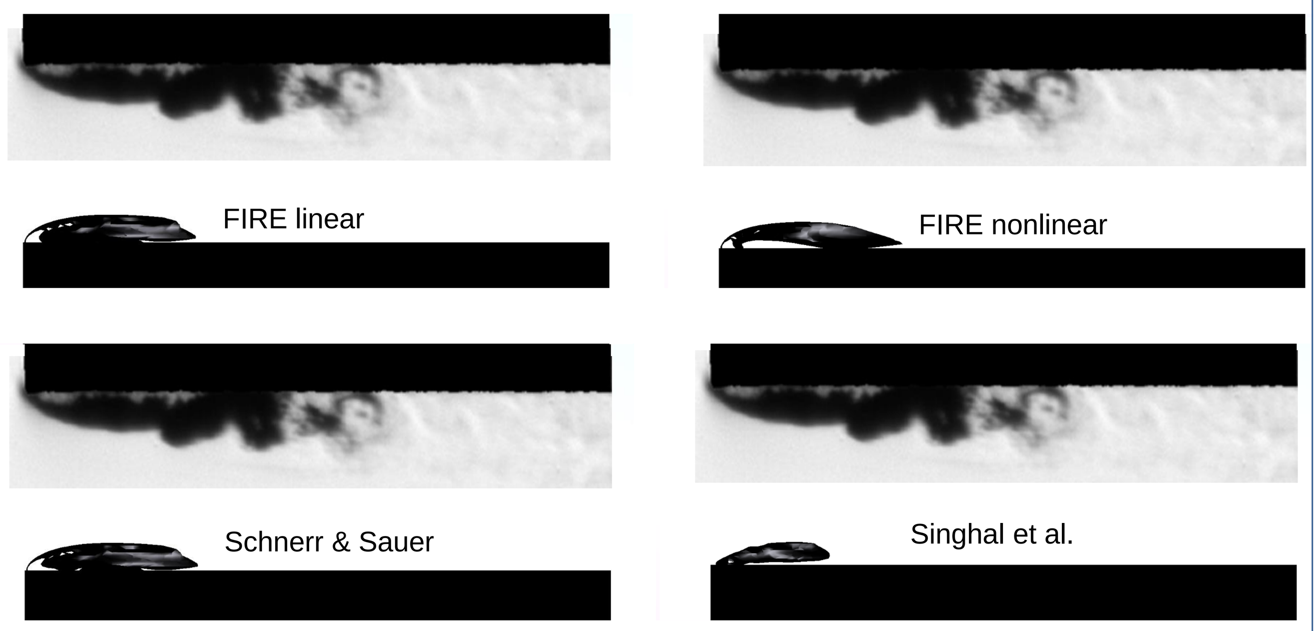

Figure 4. Vapor cloud in a throttle flow with different R-P cavitation models

Liquid/gas shock tube

Due to cavitation, fuel and vapor may coexist in the injector nozzle. The solver capability to resolve shocks and expansion waves in liquid-gas systems is then crucial to compute a reasonable flow field.
A inexpensive but relevant test case to verify the ability of a solver to resolve pressure waves, namely shocks and expansion fans, is the 1D shock tube.
The problem is initialized as a pipe with a fluid at high pressure on the left and one at low pressure on the right, divided by a membrane [2].
Figure 1 shows the characteristic flow field generated after the membrane is removed, extensively described in [3].
The following test case is a liquid/gas shock tube [4,5]. The initial conditions are:

- left: liquid dodecane at 1,000 [bar]
- right: vapor dodecane at 1 [bar]

Results presented in ***Fig. 2*** are in very good agreement with the analytical Riemann solution.

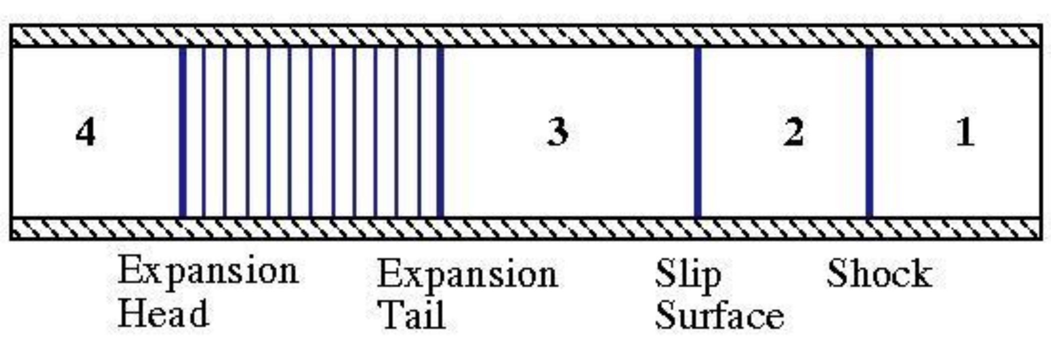

Figure 1. Flow configuration of a shock tube [3]

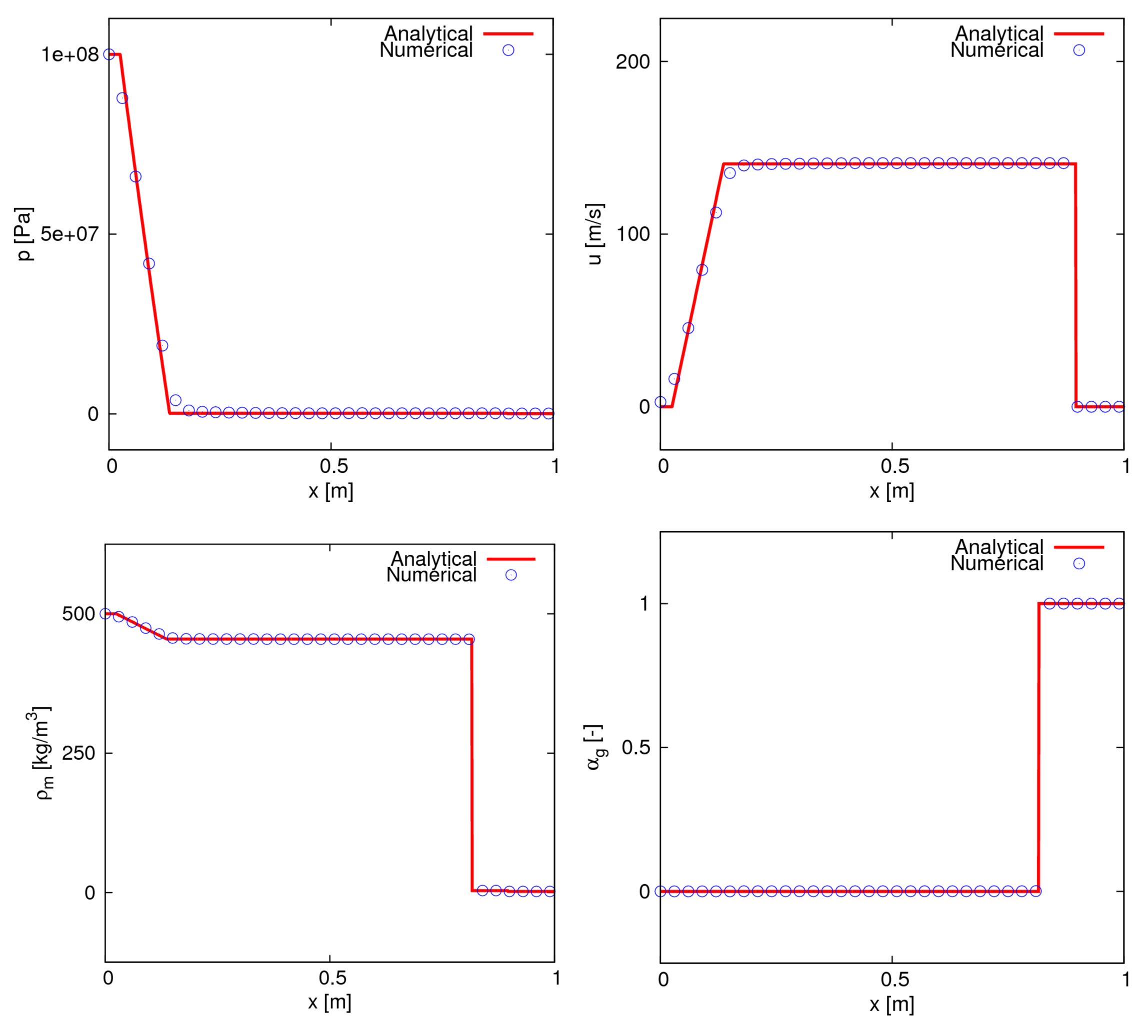

Figure 2. Liquid gas shock tube results

Euler equations are solved on 10,000 cells. The selected time step in 2 10^{-7} [s], corresponding to a CFL number of 0.3 [-]. The solver is Eulerian/Eulerian multi-phase pressure based, and it uses the SIMPLE algorithm. Pressure and velocity fields are shared between phases, while no mass and heat transfers are included. The solution is obtained proceeding in time with the 1st order accuracy and the MINMOD scheme. The stiffened gas equation of state is used for both phases [4,5].

Conclusions

- The pressure based FIRE® multiphase solver performs well in presence of pressure waves, travelling both in liquid and gas phases.
- The cavitation model originally available in the solver FIRE® [6] is very similar to the one developed by Schnerr and Sauer [7]. The nonlinear treatment of the bubble radius derivative provides more inertia to the single bubble and to the whole vapor cloud, accordingly.

Future studies

- Use the compressible multiphase solver with a cavitation model for the single bubble collapse simulation.
- Apply the model to engineering relevant cases, e.g. injector flow.

Acknowledgments

This project has received funding from the European Union's Horizon 2020 research and innovation programme under the Marie Sklodowska-Curie grant agreement No 642536.
The computations were performed at AVL List GmbH, Graz, Austria.

Contact

Marco Cristofaro
Marco.Cristofaro@avl.com
AVL List GmbH, Graz, Austria

PhD supervisors

Prof. Manolis Gavaises
M.Gavaises@city.ac.uk
City University London, UK

Dr. Wilfried Edelbauer
Wilfried.Edelbauer@avl.com
AVL List GmbH, Graz, Austria

References

1.Koukouvinis P., Gavaises M., Li J. Z. and Wang L., "Large Eddy Simulation of Diesel injector including cavitation effects and correlation to erosion damage", *Fuel*, 175, 2016
2.Sod G. A., "A survey of several finite difference methods for systems of nonlinear hyperbolic conservation laws", *Journal of computational physics*, 27.1, 1978
3.Anderson J.D., "Modern Compressible Flow", *McGraw Hill Inc.*, 1984
4.Saurel R., Petitpas F., and Abgrall R., "Modelling phase transition in metastable liquids: application to cavitating and flashing flows", *Journal of Fluid Mechanics*, 607, 2008
5. Zein A., Hantke M., and Warnecke G., "Modeling phase transition for compressible two--phase flows applied to metastable liquids", *Journal of Computational Physics*, 229, 2010
6. Grogger H. A. and Alajbegović A., "Calculation Of The Cavitating Flow In Venturi Geometries Using Two Fluid Model", *ASME Fluids Engineering Division Summer Meeting*, 1998
7.Schnerr G. and Sauer J., "Physical and numerical modeling of unsteady cavitation dynamics", *Fourth International Conference on Multiphase*, 2001
8. Singhal A. K., Athavale M. M., Li H., and Jiang Y., "Mathematical Basis and Validation of the Full Cavitation Model", *Journal of Fluids Engineering*, 124, 2002
9.Edelbauer W., Strucl J., and Morozov A., "Large Eddy Simulation of Cavitating Throttle Flow", *SimHydro: Modelling of rapid transitory flows*, 2014