Ninth International Conference on Computational Fluid Dynamics (ICCFD9), Istanbul, Turkey, July 11-15, 2016

ICCFD9-xxxx

A Well-Posed Two Phase Flow Model and its Numerical Solutions for Reactor Thermal-Fluids Analysis

Samet Y. Kadioglu[1], Ray Berry[1], Richard Martineau[1]

[1] Idaho National Laboratory, Idaho Falls, ID, 83415, USA
Corresponding author: samet.kadioglu@inl.gov

Abstract: A 7-equation two-phase flow model and its numerical implementation is presented for reactor thermal-fluids applications. The equation system is well-posed and treats both phases as compressible flows. The numerical discretization of the equation system is based on the finite element formalism. The numerical algorithm is implemented in the next generation RELAP-7 code (Idaho National Laboratory (INL)'s thermal-fluids code) built on top of an other INL's product, the massively parallel multi-implicit multi-physics object oriented code environment (MOOSE). Some preliminary thermal-fluids computations are presented.

Keywords: Two-Phase Flow, 7-Equation Model, Reactor Thermal-Fluids, RELAP-7, MOOSE.

1 Introduction

Nuclear reactors safety analysis heavily relies on accurate calculations of coupled multiple physics. Thermal fluid dynamics plays an important role in such coupled systems. A typical thermal fluid model has to have a capability of accurately predicting two-phase flow phenomena with heat and mass transfer. Furthermore, such a model must have all-speed flow capability because huge Mach number variations can exist in reactor fluid systems. A commonly used two-phase flow model is called "the 6-equation flow model" that assumes common pressure between the phases. This equation model can be mathematically ill-posed (non-hyperbolic), because it admits complex eigenvalues that can lead to inaccurate calculations of waves. This model requires extra cares such as adjustments of closure relations and numerical dissipations to provide reliable solutions [1]. One important drawback of this model worth mentioning is that although the numerical dissipation guarantees stable calculations, it can over-smear or miss important physics such as shocks or contact discontinuities. Recently, another two-phase flow model has received attention. This new model is based on a 7-equation system of partial differential equations with strong heat and mass transfer terms. It assumes that each fluid phase has its own pressure. This model is mathematically well-posed (unconditionally hyperbolic), since it only admits real eigenvalues [2]. This model guarantees correct calculations of wave speeds together with capturing more accurate physics without any extra effort. Therefore, numerical algorithms can be design to be higher order and lesser dissipative. We are currently developing a numerical algorithm for the new 7-equation model based on a finite element method equipped with a high order entropy viscosity stabilization technique. Our algorithm is being implemented in the next generation RELAP-7 code built on top of the massively parallel multi-implicit multi-physics object oriented code environment (MOOSE). In this paper, we present some preliminary numerical calculations, e.g., a waterhammer and a phase separation test problems.

2 Governing Equations

A general multi-dimensional 7-equation two-phase flow model governing the evolution of mass, momentum, energy, and volume fraction can given as

$$\frac{\partial \alpha_k \rho_k}{\partial t} + \nabla \cdot \alpha_k \rho_k \mathbf{u_k} = \Omega_k^{mass}, \quad (1)$$

$$\frac{\partial \alpha_k \rho_k \mathbf{u_k}}{\partial t} + \nabla \cdot (\alpha_k \rho_k \mathbf{u_k} \otimes \mathbf{u_k}) + \nabla \alpha_k p_k = p_I \nabla \alpha_k + \mathbf{u_I} \Omega_k^{mass} + \lambda(\mathbf{u_m} - \mathbf{u_k}),$$

$$\frac{\partial \alpha_k \rho_k E_k}{\partial t} + \nabla \cdot (\alpha_k \rho_k \mathbf{u_k} H_k) = p_I \mathbf{u_I} \cdot \nabla \alpha_k + E_I \Omega_k^{mass} + \Omega_k^{energy} + \lambda(\mathbf{u_m} - \mathbf{u_k}) \cdot \mathbf{u_I} + \mu(p_k - p_m) p_I,$$

$$\frac{\partial \alpha_k}{\partial t} + \mathbf{u_I} \cdot \nabla \alpha_k = \mu(p_k - p_m) \pm \frac{\Omega_k^{mass}}{\rho_I},$$

where $\alpha, \rho, \mathbf{u}, p$, and E denote the volume fraction, fluid density, velocity vector, fluid pressure, and fluid energy. k is the phase index (liquid or vapor), m denotes the opposite phase (e.g., if $k = liquid$, then $m = vapor$), I represents inter-facial values, $\pm$ in the last equation takes $+$ if $k = liquid$, $-$ otherwise, $H_k = \frac{\rho_k E_k + p_k}{\rho_k}$ is the enthalpy, Ω_k represents the mass or energy transfer term, λ and μ are the velocity and pressure relaxation coefficients, the two volume fractions sum to unity ($\alpha_{liquid} + \alpha_{vapor} = 1$), the thermodynamical variables are linked by the stiffened gas equation of state ($p_k = (\gamma_k - 1)\rho_k(e_k - q_k) - \gamma_k p_{k,\infty}$ where γ is the gas constant, e is the internal energy, q and p_∞ are the stiffness constants.) We remark that this general system will produce the ill-posed 6-equation model upon instantaneous pressure equilibrium (e.g., the system acquires a single pressure, $p = p_{liquid} = p_{vapor}$). Furthermore, if the velocity equilibrium is additionally enforced, then we obtain the 5-equation model that, like the 7-equation model, is also known to be hyperbolic.

3 Numerical Method

The equation (2) can be rewritten in the following form

$$\mathbf{U}_t + \nabla \cdot \mathbf{F}(\mathbf{U}) - \mathbf{S} = 0, \quad (2)$$

where $\mathbf{F}(\mathbf{U})$ represents the convective flux terms, and $\mathbf{S}$ contains the rest of the terms from (2). A variational finite element statement can be given as

$$\mathbb{R}(\mathbf{U}) = \int_\Omega (\mathbf{U}_t \cdot \mathbf{W} + \mathbf{F}(\mathbf{U}) \nabla \cdot \mathbf{W} - \mathbf{S} \cdot \mathbf{W}) d\Omega + \int_{\partial\Omega} \mathbf{F}(\mathbf{U}) \cdot \mathbf{n} \mathbf{W} d\Gamma = 0, \quad (3)$$

where $\mathbf{W}$ denotes the vector test functions. We seek the finite element solution of the form $\mathbf{U}^h = \sum_j U\phi$ where ϕ's are the basis functions. Then the analytic expression (3) turns into the following numerical approximation

$$\mathbb{R}(\mathbf{U}^h) = \int_\Omega (\mathbf{U}_t^h \cdot \mathbf{W}^h + \mathbf{F}^h(\mathbf{U}^h) \nabla \cdot \mathbf{W}^h - \mathbf{S}^h \cdot \mathbf{W}^h) d\Omega + \int_{\partial\Omega} \mathbf{F}^h(\mathbf{U}^h) \cdot \mathbf{n} \mathbf{W}^h d\Gamma = 0. \quad (4)$$

If we choose the test functions same as the basis functions, we obtain the Galerkin finite element formulation from (4). We use linear basis functions in our calculations. The integral terms in (4) are evaluated by using a numerical quadrature. We note that this finite element setting mimics the central differencing schemes, and therefore can be oscillatory when solving hyperbolic problems that contain discontinuities. The method needs to be stabilized to fix the possible instability issues. Here, we employ an entropy viscosity method as the stabilization scheme [2, 3]. This particular entropy method is based on the addition of artificial dissipation terms to the governing equations while ensuring that the physical principle remains satisfied. The time derivative in (4) is discretized based on either the backward Euler or the second order backward difference formula (BDF-2). After the space and time discretizations, (4) turns into a system of non-linear problem, e.g., $\mathbb{R}(\mathbf{U}^h) = 0$. The MOOSE framework is designed to solve such nonlinear system of equations. The solution technique is based on the Jacobian Free Newton Krylov (JFNK) methodology. The JFNK method guarantees the simultaneous convergences of all unknown components. We note that the efficient implementation of the JFNK method relies on the preconditioning of the linear system that is derived from

the Newton correction step. The framework provides several preconditioning options such as the very popular ILU (incomplete lower upper factorization) method.

4 Numerical Results

We present two one-dimensional preliminary calculations. We recast Eqns (2) into the one-dimensional variable area pipe form [2, 4]. The first problem is the classic waterhammer test that is often considered in reactor thermal-fluid study. Basically, we assume sudden flow of water in a pipe system that can not respond acoustically, resulting in pressure surge (waterhammer). The same assumption is valid for the steam (vapor) flow. In the two-phase case, we assume mixture of water and vapor flow. The waterhammer events are transmitted through the system at the wave speed of the water-steam combination. This wave transmission can result in significant dynamic loads to the piping supports as well as the system components, i.e. valves, instrumentation, etc. Therefore, water or steam hammer events have the potential to cause catastrophic damage to the reactor components. In our test case, we consider a $10m$ long horizontal pipe with a cross section area $A = 2.0 \times 10^{-4} m^2$. The initial pressure is $7Mpa$, the initial temperature is set to $513K$, the initial velocity is $12.5m/s$, and the initial liquid volume fraction is 0.999 meaning that the flow is initiated as mostly water. At both ends of the pipe, we impose the solid wall boundary conditions. Figure 1 shows the time history of the pressure solutions. This initial setting creates a compression wave (from the right end) moving into the pipe. We also have a decompression wave propagating from the left side of the pipe (first subfigure). The pressure wave reflects from the left end and further amplifies (third subfigure). In time pressure magnitude diminishes as expected (last subfigure). From these subfigures, we observe that the water and steam pressure are slightly different. This is physically expected at the early stages of the waterhammer test. Figure 1 indicates that our 7-equation model accurately captures this physics. Also, our numerical method performs reasonably well meaning that we do not have spurious oscillations together with well resolved wave fronts considering that we are using only 100 elements in our computation.

The second problem considers a two-phase hydrostatic flow test in which phase separation occurs. We consider a $3.66m$ long vertical pipe with the cross section area $2.0 \times 10^{-4} m^2$. The initial pressure is again set to $7Mpa$, the initial temperature is $513K$, the initial velocity in this time is $0.0m/s$, and the initial liquid volume fraction is 0.5 meaning that the mixture is half steam and half water. Again, at both ends of the pipe, we use the solid wall boundary conditions. Figure 2 shows the time evolution of the mixture pressure field and the liquid volume fraction. The water-steam mixture starts to separate immediately driven by the gravitational force. The heavier fluid (water) accumulates at the bottom half of the pipe, and the steam occupies the top half. The problem reaches the steady state around $time = 400s$. At this time, the two phases are completely separated. The last subfigure shows that the mixture pressure changes hydrostatically but due to the larger density of water is more pronounced in the water phase. Again our numerical method shows no traces of spurious oscillations around the discontinuity which is a good indication that our stabilization technique (the entropy viscosity method) works well. The contact front is captured sharp considering that we are using relatively coarse mesh (100 elements).

5 Conclusion and Future Work

We have presented one dimensional preliminary results. These results indicate that our mathematical model (7-equation, two-phase flow model) provides accurate physical representations. The results also show that our numerical methodology works well, capturing clean and sharp discontinuities. We note that the numerical results are obtained by running the code fully implicit within the JFNK framework. During the transient, we employ time step control for accuracy reasons. This increases computer time considering the amount of function evaluations in both linear and nonlinear iteration procedures. To increase the efficiency, especially for the steady state or slow transient problems (second test in this paper), we plan to utilize our previously invented point implicit (PIM) method [5].

References

[1] Samet Y Kadioglu, Robert Nourgaliev, and Nam Dinh. A Novel Hyperbolization Procedure for the Two-Phase Six-Equation Flow Model. *INL Technical Report*, INL/EXT–11-23551, 2011.

[2] R.A. Berry, L. Zou, H. Zhao, H. Zhang, J.W. Peterson, R.C. Martineau, S.Y. Kadioglu, and D. Andrs. *RELAP-7 Theory Manual. INL Technical Report*, INL/EXT-14-31366 (Revision 2), DOI: 10.13140/rg.2.1.4346.1522, 2016.

[3] Marc O. Delchinia, Jean C. Ragusaa, and Ray A. Berry. Entropy-based viscous regularization for the multi-dimensional Euler equations in low-Mach and transonic flows. *Computers and Fluids*, 118:225–244, 2015.

[4] Richard Saurel, Fabien Petitpas, and Ray A. Berry. Simple and efficient relaxation methods for interfaces separating compressible fluids, cavitating flows and shocks in multiphase mixtures. *Journal of Computational Physics*, 228:1678–1712, 2009.

[5] S.Y. Kadioglu, R.A. Berry, and R.C. Marineau. A point implicit time integration technique for slow transient flow problems. *Nuclear Engineering and Design*, 286:130–138, 2015.

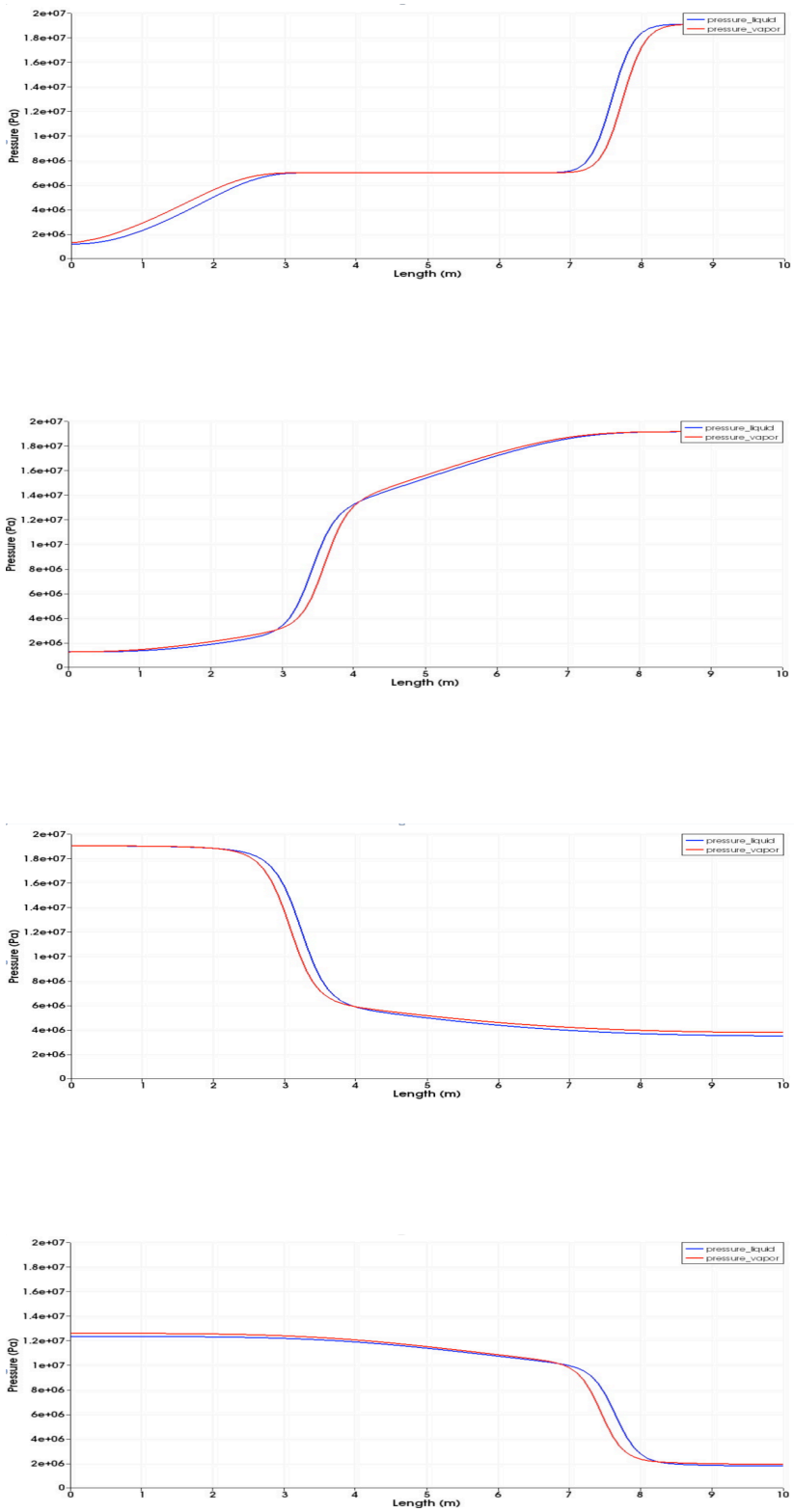

Figure 1: Time = 5ms, 10ms, 20ms, 30ms

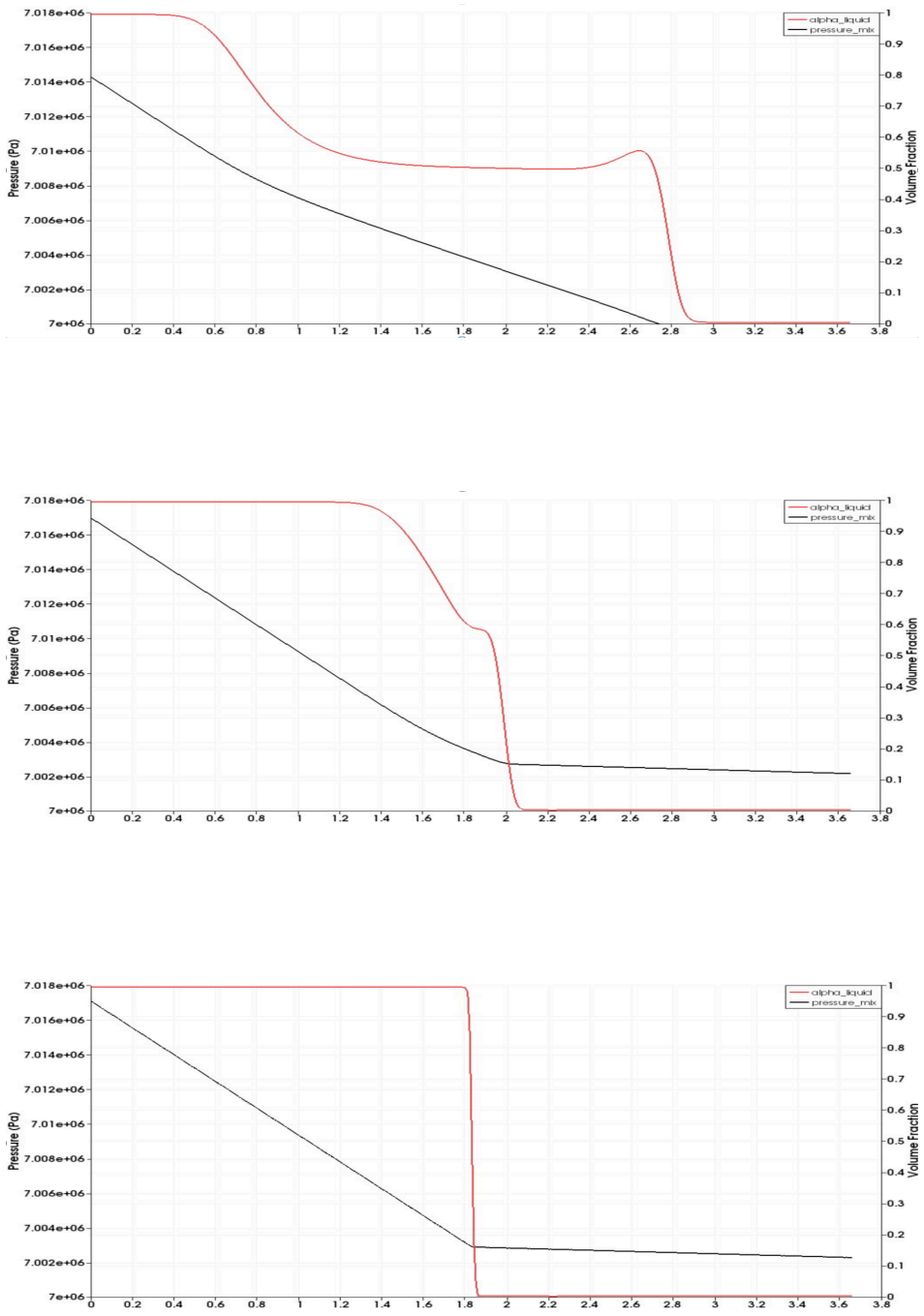

Figure 2: Time = 100s, 200s, 600s.