M&C 2017 - International Conference on Mathematics & Computational Methods Applied to Nuclear Science & Engineering, Jeju, Korea, April 16-20, 2017, on USB (2017)

Evaluation of a Hyperbolic Conservative Mixture Model for Thermal-Hydraulics Two-Phase Flows of Nuclear Reactors

Dia Zeidan[†] and Henrik Ström[*]

[†]*School of Basic Sciences and Humanities, German Jordanian University, Amman, Jordan*
[*]*Department of Applied Mechanics, Chalmers University of Technology, Gothenburg, Sweden*
dia.zeidan@gju.edu.jo, henrik.strom@chalmers.se

Abstract - *This paper briefly presents simulation results of a hyperbolic model of the two-phase flow equations written in a conservative form and discretized using finite volume methods. The model presented herein utilizes mixture properties and constitutes a three governing equations model. Such a model able to treat the relative motion between phases independently. Godunov methods of centred type are used for the convection terms. The TVD SLIC scheme is used in which the fluxes are calculated by the Riemann problem. The key results include the ability of the model to simulate two-phase flows with strong gradients in both velocity and phase fractions. In particular, the model is robust and reproduces with fair accuracy the complete wave structure associated with the flow variables. The model and the methods are capable of handling numerical instabilities that may occur during simulations, which is essential in analysis of two-phase flow phenomena in nuclear reactors. The model in hand is considered promising for future consideration of additional sub-models describing complex mechanisms governing gas-liquid mixture two-phase flow including thermodynamic non-equilibrium (subcooled boiling), heterogeneous volume fraction and mass fraction fields for the gas phase.*

I. INTRODUCTION

Gas and liquid multiphase flows are of fundamental importance in the design, operation and safety assessment of nuclear reactors. The study of such reactors is a challenging task both in the mathematical modelling and in designing accurate and robust numerical tools. Typically, these reactors are characterized by thermal-hydraulics and multiphase flows that refers to the formation of gases in liquid flows. Because of the large size of these reactors, the multi-phase or two-phase phenomena within such reactors have been investigated widely with two-fluid models [1, 2, 3, 4, 5]. Such models are derived from conservation laws for each phase for mass, momentum and energy along with phase interaction throughout interfacial terms. Two main issues of concern are related to these models in their standard form. The first one is the occurrence of complex eigenvalues resulting in ill-posed or non-hyperbolic initial-boundary-value problems. The second concern is the non-conservative temporal and spatial terms appearing in their final formulation. While such concerns undoubtedly heralds mathematical, numerical and practical challenges, considerable remedies have been proposed over years in resolving some of these challenges. These include, but are not limited to, incorporating more physical terms such as the viscous stress terms to the momentum equations, the interfacial pressure force terms, using the surface tension force terms or including the relative velocity in the buoyant force term. Although hyperbolicity can be provided with such physical assumptions, the resulting hyperbolic initial-boundary-value problems are inherently non-conservative two-fluid models. Lack of hyperbolicity and conservativity characters of such models results in spurious oscillations in the numerical resolutions which render high-fidelity simulations of small-scale two-phase phenomena in nuclear reactors intractable. Further, these models are in general based on homogeneous equilibrium formulation involving equal or unequal pressure for each phase. A substantial literature exists on the simulations of flow of two fluid phases with currently used hyperbolic and non-hyperbolic non-conservative two-fluid models. See, for example, [3, 6, 7, 8, 9, 10, 11, 12] and the references therein. Despite the maturity that such models have been attained, the complicated coupling terms of physical mechanisms governing two-phase flows within nuclear reactors has implied that the previously mentioned mathematical and numerical challenges remain a crucial interest for scientific investigation. The purpose of this paper is to introduce a recently developed two-phase flow model for thermal-hydraulics systems in nuclear reactors. This model is based on the theory of thermodynamically compatible systems of hyperbolic conservation laws of continuum mechanics [13] and has been used extensively to investigate single and two-phase flow problems, e.g. see [13, 14, 15]. The model is formulated in terms of parameters of state for the mixture and describes gas and liquid phases in a complete non-equilibrium. Formulations with such parameters allows the relative motion between the two phase systems to be expressed by a separate equation. In addition to that, it allows the different phases to be interpenetrated with each other, that is, the volume fractions of phases can be equal to any value between 0 and 1, and the interaction between phases can be expressed in different forms within the resulting mixture equations. Within the context of mixture formulations, the model consists of a mixture mass, mixture momentum and a balance law for the relative velocity between the two phases. A very important key feature of this model is its ability to describe the behaviour of the relative motion between the two phase systems. In practice, this is one of the most important characteristics for two-phase flow mechanisms along with their mathematical

description designed for the numerical simulations of nuclear power plants thermal-hydraulics. Further, the system of equations exhibits rich mathematical structures including shock and rarefaction waves solutions. These structures also facilitate the construction of approximate Riemann solvers and locally involve them in the framework of well-developed numerical methods for the complete resolution of the model equations. In contrast to considerable recent advances in the development of numerical methods for two-phase flows, extension of such methods to the fully conservative two-phase flow models have remained relatively scarce. In general, however, this extension also is mainly carried out by finite-volume methods such as Godunov-type methods for simulating the model equations. Accordingly, a total variation diminishing (TVD) slope limiter centre (SLIC) numerical scheme is applied and extended for the resolution of the model equations. The TVD SLIC is a Godunov centred type scheme to solve hyperbolic conservation laws that achieves second-order accuracy by using MUSCL extrapolation. See, for example, [16]. Based on finite-volume methods, the model equations are discretized using the TVD SLIC scheme where the solution of the non-linear Riemann problem is fully numerical. The scheme is then applied and tested on different two-phase problems selected from the open literature. The performed numerical simulations clearly demonstrate that both the model equations and numerical methods are capable of predicting the two-phase flows wave phenomena with a high degree of accuracy, robustness and without any oscillation and additional diffusion. Simulations show that the results of the model and the methods presented here compare well with other numerical methods available in the literature.

In this article, the model equations are presented in the next section. The numerical methods are explained in the section after. The ability of this hyperbolic and conservative mixture model to resolve steep gradients in the velocity and phase fraction fields, without experiencing severe oscillations or exaggerated numerical smearing, will be assessed in the fourth section. Concluding remarks are given in the last section.

II. MATHEMATICAL MODEL

The aim of the present work is to address the accuracy and robustness obtainable with a fully hyperbolic and fully conservative two-phase flow model for thermal-hydraulic analysis of nuclear reactors. At the present stage, the two-phase flow of interest is, therefore, primarily a relatively dilute disperse gas-liquid flow in which small steam bubbles have just been formed in a continuous water phase. This physical situation, for instance, is found in the region after the boiling onset within a boiling water reactor (BWR). In such a flow situation, local equilibrium between the two phases is attained over short spatial scales and a mixture model formulation may appropriately employed to simplify the complete two-phase flow equations. Thus, mixture parameters of state [14, 15] are introduced, and the mathematical model under consideration, in terms of conserved variables, reads as follows:

- Continuity equation of gas-liquid mixture

$$\frac{\partial}{\partial t}(\rho) + \frac{\partial}{\partial x}(\rho u) = 0. \tag{1}$$

- Momentum equation of gas-liquid mixture

$$\frac{\partial}{\partial t}(\rho u) + \frac{\partial}{\partial x}(\rho u^2 + P + \rho c(1-c)u_r^2) = \mathcal{S}. \tag{2}$$

- Relative velocity equation of gas-liquid mixture

$$\frac{\partial}{\partial t}(u_r) + \frac{\partial}{\partial x}\Big(uu_r + (1-2c)\frac{u_r^2}{2} + \psi(P)\Big) = \pi. \tag{3}$$

The symbols in the equations presented above have the following meanings: $t \in \mathbb{R}_0^+$ denotes time and $x \in \mathbb{R}$ is spatial position, ρ, u, P, c and u_r denote, respectively, the mixture density, mixture velocity, mixture pressure, gas mass void fraction and relative velocity between the two phase systems. These mixture variables are related through the gas volume fraction, α, in the following form

$$\rho = \alpha\rho_2 + (1-\alpha)\rho_1 \quad \text{and} \quad \rho u = \alpha\rho_2 u_2 + (1-\alpha)\rho_1 u_1,$$
$$P = \alpha P_2 + (1-\alpha)P_1,\ c = \alpha\rho_2\rho^{-1} \quad \text{and} \quad u_r = u_2 - u_1.$$

Here two, 2, and one, 1, are subscripts denotes the gas and liquid phases, P_k, ρ_k and u_k are the pressure, velocity and density of phase k, where $k = 1, 2$, respectively, and α is between 0 and 1 and satisfies the constraint $\alpha + (1-\alpha) = 1$. In the present work, we have assumed isentropic conditions within the equations and that no mass transfer between phases occurs. Further, the basic physics is that the two phases are coupled by the relative velocity equation rather than the phase momentum equations. In this regard, additional terms arise in the framework of velocity non-equilibrium behaviour showing the benefits of using mixture formulations. Based on such formulations, the term $\psi(P)$ is a function that depends on phase equations of state (EOS) for isentropic and non-isentropic laws defined as

$$\psi(P) = e_2 + \frac{P_2}{\rho_2} - e_1 - \frac{P_1}{\rho_1} = \psi_2(P_2) - \psi_1(P_1). \tag{4}$$

The source terms on the right-hand side of equations of (2) and (3) represent the interphase exchange processes. The different forms of these terms arising in a range of flow processes have been discussed extensively in the literature (see [8, 10, 15, 17] and references therein). Although an adequate treatment of the interphase exchange terms will be important in a final application, the effects of including various exchange models on the stability and overall behaviour of the numerical problem would render the current assessment of the intrinsic model capabilities impossible [8, 18]. These various forms for modelling the source terms therefore fall outside the authors' interest in this paper, and have been disregarded for the sake of clarity and simplicity.

The system of equations (1)-(3) can be written in the following vector conservative form:

$$\frac{\partial \mathbb{U}}{\partial t} + \frac{\partial \mathbb{F}(\mathbb{U})}{\partial x} = \mathbb{S}(\mathbb{U}), \tag{5}$$

where the vector representing the conserved flow variables, flux and source term vectors, respectively, are given by

$$\mathbb{U} = \begin{pmatrix} \rho \\ \rho u \\ u_r \end{pmatrix}, \quad \mathbb{F}(\mathbb{U}) = \begin{pmatrix} \rho u \\ \rho u^2 + P + \rho c(1-c)u_r^2 \\ u u_r + (1-2c)\frac{u_r^2}{2} + \psi(P) \end{pmatrix}$$

and $\mathbb{S} = \Big(0, \mathcal{S}, \pi\Big)\cdot$

If we set $u_2 = u_1$ and $\alpha = 1$ in the above system, then one recover single-phase flow equations. System (5) also has been compared with currently used two-fluid models. See [14, 15] for further details. As can be seen in equations (1)-(3), system (5) represent a fully conservative initial-boundary-value problem. Further, the eigenvalues of system (5) in terms of parameters of state for the mixture are

$$\lambda_1 = u - a_m + \mathcal{Y} u_r, \quad \lambda_2 = u \quad \text{and} \quad \lambda_3 = u + a_m + \mathcal{Y} u_r,$$

where $\mathcal{Y}$ is a function connects the mixture variables to each other and a_m is the speed of sound of the mixture. For further details, see [15]. With the complete hyperbolic nature and the conservative form in hand, system (5) admits discontinuous solutions. More specifically, it admits a linearly degenerate field, associated with the middle wave, and two genuinely non-linear fields, associated to the left and right waves. This has the consequence that the model equations can be solved using any standard numerical method without any specific numerical design.

III. NUMERICAL APPROXIMATION

Although system (1)-(3) is independent of the type of numerical method used to resolve it, the equations of the mixture model are discretized for processes without dissipation, the source terms are zero, using finite-volume methods. The computational domain discretization is based on a mesh of length Δx and the temporal domain t into intervals of duration Δt. For the i^{th} mesh, this discretization results as

$$\mathbb{U}_i^{n+1} = \mathbb{U}_i^n - \frac{\Delta t}{\Delta x}\Big[\mathbb{F}_{i+\frac{1}{2}} - \mathbb{F}_{i-\frac{1}{2}}\Big]. \tag{6}$$

In equation (6), the time step is determined by

$$\Delta t = \text{CFL} \cdot \frac{\Delta x}{S_{\max}^{(n)}},$$

where the Courant number coefficient CFL is between 0 and 1 and the maximum wave speed $S_{\max}^{(n)}$ is selected at the current time level n as

$$S_{\max}^{(n)} = \max_i \{|\lambda_i|\},$$

and λ_i are eigenvalues corresponding to sound waves. Further, the determination of $\mathbb{U}$ at the new time step $n+1$ necessitates the computation of the numerical flux $\mathbb{F}$ at the cell interfaces at the previous time n. For the evaluation of the numerical fluxes of the model equations in finite volume methods, one need to solve the Riemann problem at the cell interfaces together with the initial conditions

$$\mathbb{U}(x, t=0) = \begin{cases} \mathbb{U}_L, & \text{if} \quad x \le x_0, \\ \\ \mathbb{U}_R, & \text{if} \quad x > x_0, \end{cases} \tag{7}$$

where $\mathbb{U}_L$ and $\mathbb{U}_R$ are given left and right constant states of the gas and liquid flows, respectively, separated by a discontinuity at $x = x_0$. The solution to the Riemann problem for system (1)-(3) can be analytical or numerical. Analytical solution of the model equations, however, faces a practical barrier due to the equations of state and to the relative velocity equation between the two phases. Although exact Riemann solvers are available for specific mixture and two-fluid models, they have been developed using certain simplifications such as mathematical and physical assumptions related to one of the phases. For a review, see e.g. [15] and the references therein. This entails the numerical approximation of two-phase flow equations where the solution of the Riemann problem is fully numerical. In the present work, we consider Godunov methods of centred type, namely the second-order Slope Limiter Centred (SLIC) scheme, which have been studied in [16] for single-phase flows. This method gives high-resolution of large-gradient regions that are free from spurious oscillations. Further, it involves the following three steps.

- Data reconstruction. In this step, the cell averaged values $\mathbb{U}_i^n$ are locally restored by piecewise linear function in every cell as

$$\mathbb{U}_i(x) = \mathbb{U}_i^n(x) + \frac{(x - x_i)}{\Delta x}\Delta_i. \tag{8}$$

  The values $\mathbb{U}_i^n$ then are transformed to the boundary extrapolated values as

$$\mathbb{U}_i^{L,R} = \mathbb{U}_i^n \pm \frac{\Delta_i}{2}, \tag{9}$$

  where Δ_i defines a suitable slope limiter of the linear function set in every cell.

- Evolution. The boundary extrapolated values are then evolved by a half time step, $\frac{1}{2}\Delta t$, for every cell I_i. This is carried out according to

$$\Big(\mathbb{U}_i^{L,R}\Big)^{\text{New}} = \mathbb{U}_i^{L,R} + \frac{1}{2}\frac{\Delta t}{\Delta x}[\mathbb{F}(\mathbb{U}_i^L) - \mathbb{F}(\mathbb{U}_i^R)], \tag{10}$$

  where the above intercell fluxes are calculated at the boundary extrapolated values of each cell.

- The Riemann problem. The numerical fluxes employed in this step are computed using the following standard first-order centered (FORCE) flux where the solution of the Riemann problem associated with system (5) is fully numerical

$$\mathbb{F}_{i+\frac{1}{2}}^{\text{FORCE}} = \mathbb{F}_{i+\frac{1}{2}}\Big(\Big(\mathbb{U}_i^R\Big)^{\text{New}}, \Big(\mathbb{U}_{i+1}^L\Big)^{\text{New}}\Big), \tag{11}$$

where $\mathbb{U}_L = \left(\mathbb{U}_i^R\right)^{\text{New}}$ and $\mathbb{U}_R = \left(\mathbb{U}_{i+1}^L\right)^{\text{New}}$ and the FORCE intercell flux is defined as the average of fluxes obtained using Lax-Friedrichs (LF) and Richtmyer (RI) methods as

$$\mathbb{F}^{\text{FORCE}}_{i+\frac{1}{2}} = \frac{1}{2}\Big(\mathbb{F}^{\text{LF}}_{i+\frac{1}{2}} + \mathbb{F}^{\text{RI}}_{i+\frac{1}{2}}\Big),$$

with $\mathbb{F}^{\text{LF}}_{i+\frac{1}{2}}$ and $\mathbb{F}^{\text{RI}}_{i+\frac{1}{2}}$ are expressed as

$$\mathbb{F}^{\text{LF}}_{i+\frac{1}{2}} = \frac{1}{2}\Big[\mathbb{F}(\mathbb{U}_i^n) + \mathbb{F}(\mathbb{U}_{i+1}^n)\Big] + \frac{1}{2}\frac{\Delta x}{\Delta t}\Big[\mathbb{U}_i^n - \mathbb{U}_{i+1}^n\Big],$$

and

$$\mathbb{F}^{\text{RI}}_{i+\frac{1}{2}} = \mathbb{F}(\mathbb{U}_{i+\frac{1}{2}}), \quad \mathbb{U}^{\text{RI}}_{i+\frac{1}{2}} = \frac{1}{2}(\mathbb{U}_i^n + \mathbb{U}_{i+1}^n) + \frac{1}{2}\frac{\Delta t}{\Delta x}[\mathbb{F}(\mathbb{U}_i^n) - \mathbb{F}(\mathbb{U}_{i+1}^n)].$$

The SLIC scheme has been validated thoroughly using single-phase flow problems in, for example, gas dynamics [16]. Two-phase flow validation has been carried out for wave propagation in gas, liquid and solid phases using the two-fluid models and the current model, see for example [15, 19] and references therein.

IV. COMPUTATIONAL EXAMPLES

In order to test the model equations presented in the previous section, numerical computations have been performed by means of Godunov methods. More specifically, the accuracy and versatility of the proposed numerical model is investigated for three canonical two-phase flow test problems.
For test 1, the initial values are given as [11]:

$$\begin{aligned}(\alpha_2, \rho_2, u_2, \rho_1, u_1)_L &= (0.01, 50, -100, 1000, -100) \\ &\text{if} \quad x \le 0, \\ (\alpha_2, \rho_2, u_2, \rho_1, u_1)_R &= (0.01, 50, \quad 100, 1000, \quad 100) \\ &\text{if} \quad x > 0.\end{aligned}$$

For test 2, the initial values are given as [6, 15]:

$$\begin{aligned}(\alpha_2, \rho_2, u_2, \rho_1, u_1)_L &= (0.29, 10, 3800, 65, 1) \\ &\text{if} \quad x \le 0, \\ (\alpha_2, \rho_2, u_2, \rho_1, u_1)_R &= (\ 0.3, \ 1, \ \ 1000, 50, 1) \\ &\text{if} \quad x > 0.\end{aligned}$$

For test 3, the initial values are given as [20]:

$$\begin{aligned}(\alpha_2, \rho_2, u_2, \rho_1, u_1)_L &= (0.9, 719.6856, 150, 1225.8912, 150) \\ &\text{if} \quad x \le 0, \\ (\alpha_2, \rho_2, u_2, \rho_1, u_1)_R &= (0.9, 719.6856, -u_{2_L}, 1225.8912, -u_{1_L}) \\ &\text{if} \quad x > 0.\end{aligned}$$

The following stiffened equation of state (EOS) is taken for both the gas and liquid phases

$$P_j = K_j\left(\frac{\rho_j}{\bar{\rho}_j}\right)^{\gamma_j} - P_\infty,$$

where $j = 2, 1$ stands for the gas and liquid phases, respectively, and γ_j, K_j, P_∞ and $\bar{\rho}_j$ are constant parameters to be specified for each test case. The three test problems are solved in the special domain $[-10, 10]$ using the total variation diminishing (TVD) slope limiter centre (SLIC) scheme. Simulation results are depicted in figures 1-4. Since analytical solutions do not exist, we produce a high-resolution numerical solution for system (1)-(3) to calculate the reference solution for each test problem. For these reference solutions we make use of the TVD SLIC scheme on a very fine mesh of 10000 cells. In all simulations, the CFL number taken as 0.9 along with transmissive boundary conditions using the SUPERBEE limiter within the SLIC scheme. Numerical results are compared with both the reference solutions and with the Lax-Friedrichs scheme, which is an explicit scheme that does not require any approximate Riemann solver. The reference solution is shown with a solid line while the symbols show the numerical solutions in all test problems.
Test 1 is a commonly used cavitation problem, see e.g. [11]. In nuclear reactor systems, cavitation phenomena may take place in conjunction with the starting or stopping of pumps or the closure of valves [21], or be induced by the flow in the coolant channels [22]. Numerical analysis of such phenomena are directly valuable for both safety and monitoring purposes. Test 1 is a modified version two-phase rarefaction problem of [11] that includes a tube filled initially with 1% dispersed gas among water at atmospheric pressure. Numerical results are displayed in figure 1 at $t = 1.826$ ms using the following EOS constant parameters for the gas and liquid phases, respectively, $\gamma_2 = 1.4$, $\bar{\rho}_2 = 100.0$ kg/m^3, $K_2 = 100.0$ Pa and $P_\infty = 0$ and γ_1 is the ratio of specific heat for the liquid, P_∞ and $\bar{\rho}_1$ are the reference pressure and density with values of 2.8, 10^7 Pa and 100.0 kg/m^3 and $K_1 = 700$ Pa. From figure 1 one can see that both system (1)-(3) and the TVD SLIC scheme can resolve the strong rarefaction waves well and similar to the results presented in [11] at $t = 1.826$ ms. In figure 1 also one can note the difference when comparing the first first-order numerical methods, first-order centered (FORCE) and Lax-Friedrichs methods with the TVD SLIC scheme. The agreement between the results provided by the three numerical methods with 100 mesh points and the reference solution is good enough and oscillation-free throughout the complete wave structure. The numerical results are in good agreement with the results of [11], obtained using different two-phase flow equations and numerical resolution. In particular, the relative velocity profile appears in figure 1, provided using the relative velocity equation (3), is in excellent agreement with the reference solution through the complete wave structure. This profile is missing in [11].
Next we consider a mixture shock tube problem, Test 2, which is an important application in nuclear safety analysis [23]. Results are shown in figure 2 at $t = 0.1$ ms. Shock wave propagation may, for example, occur in a nuclear reactor during a Loss of Coolant Accident (LOCA). This Riemann problem is an extended version of a test constructed in [6, 15]. The solution for this test case consists of a left rarefaction wave, a right travelling contact discontinuity and a right shock wave with the following EOS constant parameters, respectively, $\gamma_2 = 1.4$, $\bar{\rho}_2 = 1.0$ kg/m^3 and $K_2 = 1.0$

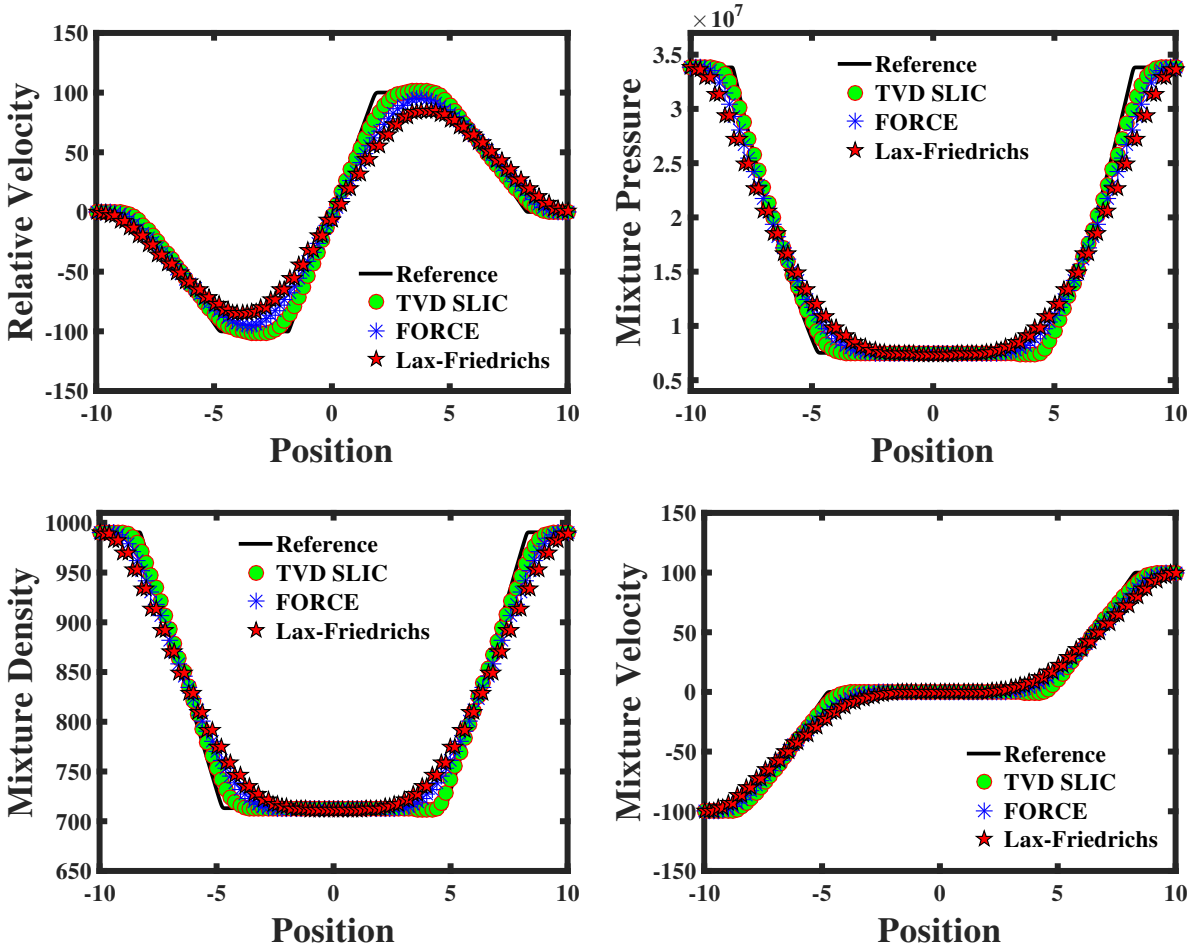

Fig. 1. Test 1. Cavitation problem in a gas-liquid mixture at time $t = 1.826$ ms. Three different numerical methods, namely the TVD SLIC, FORCE and Lax-Friedrichs methods are compared with the reference solution results. Coarse meshes, symbols, are provided on 100 cells and very fine meshes of 10000 cells for the solid lines. The relative velocity profile was not displayed in [11].

Pa, $P_\infty = 0$ Pa, and γ_1, P_∞, K_1 and $\bar{\rho}_1$ are 2.8, 1 Pa, 1 Pa and 1.0 kg/m^3. One can see the complete wave structure is resolved very well by the velocity non-equilibrium mixture model and the TVD SLIC scheme. In particular, the model in hand significantly compute the relative motion between the gas and liquid flows without any added stability terms to the governing equations. The relative velocity profile appearing in figure 2 changes only discontinuously across the contact discontinuity. Further, the performance of the numerical methods and model equations are very satisfactory, all discontinuities are sharp and oscillations-free. For this test problem, the numerical results agree well with the reference solutions. Among the three presented methods the TVD SLIC scheme is shown to be more accurate leading to sharper and better profiles than the results obtained with the lower order methods for gas-liquid mixture two-phase flow.

The final test problem consists of a symmetric colliding shock waves and a trivial contact discontinuity proposed in [20] for a currently used two-fluid mode of two compressible fluids. The numerical results are shown in figure 3 at $t = 0.0004$ ms with $\gamma_2 = 1.4$, $\bar{\rho}_2 = 1.0$ kg/m^3, $P_\infty = 0$ Pa and $K_2 = 1 \times 10^5$ Pa, and γ_1, P_∞ and $\bar{\rho}_1$ are 7.15, 0 Pa and 1000.0 kg/m^3 and $K_1 = 3.03975 \times 10^8$. The shock-wave test case represents a well-established way to investigate the inclination to produce oscillatory solutions for numerical schemes employed in two-phase flow modelling [24, 25]. It is often used to test numerical formulations for duct-like flow, such as two-phase flow inside a pipeline. This flow situation is of high relevance for nuclear reactor applications where the spatial discretization, in general, usually relies on the assumption that the flow is essentially one-dimensional. Such a one-dimensional treatment of the two-phase mixture is typically justified by the fact that the magnitude of the secondary flows existing in a fuel assembly sub-channel is order of magnitude smaller than the magnitude of the axial flow [4]. Thus, non-oscillatory solutions in shock-wave test cases are regarded as appropriate indicators of robust enough behaviour for application to more realistic nuclear core simulations. The numerical solutions

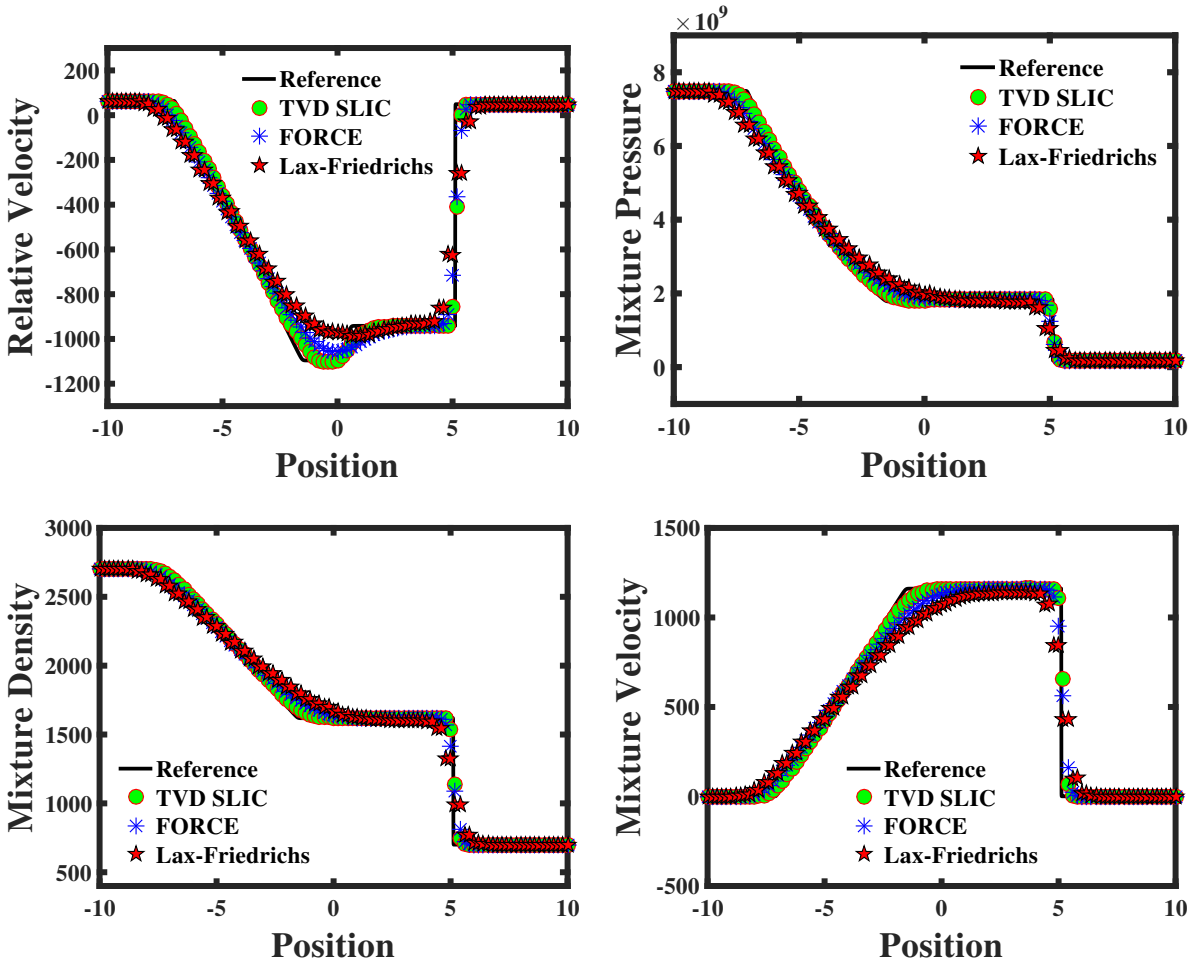

Fig. 2. Test 2. Gas-liquid two-phase shock tube problem at time t = 0.1 ms. Three different numerical methods using a coarse mesh of 100 cells are compared with the reference solution on a very fine mesh of 10000 cells.

provided by the three different methods is compared with the reference solution in this figure. Overall, one observe that all the shock waves are accurately captured and located in the correct position. Also, one can note that the mixture flow density, mixture velocity and mixture pressure remain primarily undisturbed whereas the relative velocity between the two phase systems jump discontinuously across the middle wave as observed in the previous test problems. Further, it is noted that a visible unphysical kink or dip in the mixture density and mixture pressure across the contact discontinuity in this test problem but not the previous test problems. This unphysical kink also becomes more noticeable in the TVD SLIC scheme even with the reference solutions on a very fine mesh.

To get insight of this behaviour, we repeat the same test problem using Lax-Friedrichs scheme with a different number of cells (a very coarse mesh of 50 cells, a coarse mesh of 500 cells, a fine mesh of 5000 cells and a very fine mesh of 10000 cells) using as shown in figure 4. The unphysical kink or dip over the contact discontinuity for the mixture density and mixture pressure is visible regardless of whether the mesh is coarse or fine. Overall, there is an excellent agreement between the Lax-Friedrichs scheme fine mesh with the reference solution. It is worth noting that this unphysical kink is not related to the hyperbolic nature of the mixture equations nor the conservative formulation. Such a behaviour may be related to spurious mixture density perturbation which is a well-known anomalies of hyperbolic systems [16]. However, considerable research should be carried out to further investigate how such unphysical kinks can appear in gas-liquid mixture collision and its association with thermal-hydraulics two-phase flows.

V. CONCLUDING REMARKS

A hyperbolic mixture model in conservative form is presented. The model formulation accounts for the non-equilibrium behaviour between the two phase systems and guarantees that the relative velocity is expressed by a separate equation. It is found that the model equations are suitable for Godunov-type methods without the need for a posteriori numerical treatments. Further, the model equations

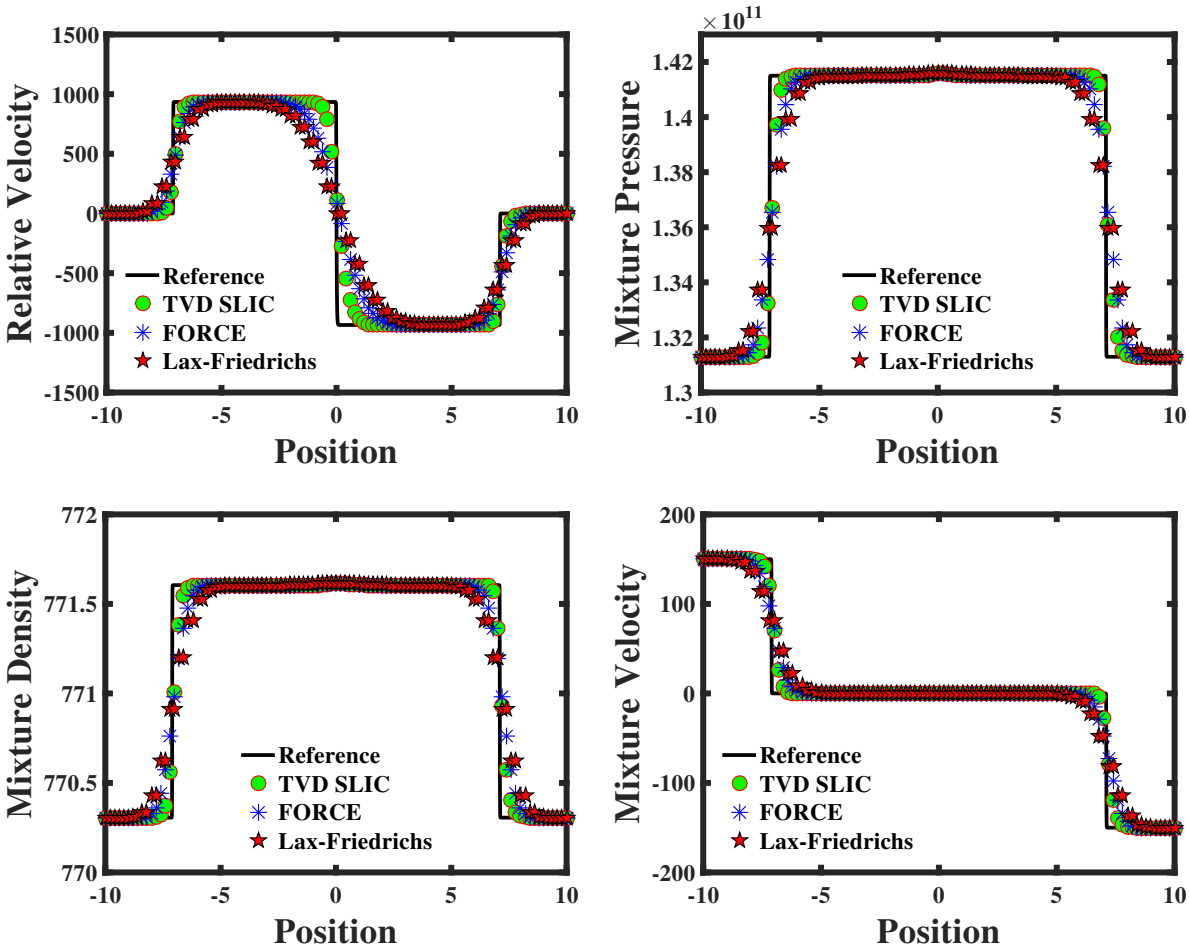

Fig. 3. Test 3. Solution of the mixture model for the one-dimensional colliding shock waves problem at time t = 0.0004 ms.

are able to handle flows such as those encountered in Loss of Coolant Accident (LOCA) and in other specific situations arising in the framework of nuclear reactor systems. Compared to other models existing in the literature, the model in hand does not require any specific simplifications as indicated by the numerical results. Further, the model and the methods have been shown to be computationally efficient and robust, the main advantage being that the model equations are solved using mixture formulations. The robustness of the simulation results indicates that the mathematical model and the associated numerical method may be extended in various ways to address even more complex phenomena such as the stability of dispersed bubbly gas-liquid flows in nuclear reactors. A natural next step would for example be to extend the current work to engineering simulations of nuclear reactors thermal-hydraulics and multiphase flows research.

ACKNOWLEDGMENTS

The authors gratefully acknowledge support from the German Jordanian University.

REFERENCES

1. L. Zou, H. Zhao and H. Zhang, "Numerical Implementation, Verification and Validation of Two-Phase Flow Four-equation Drift Flux Model with Jacobian-Free Newton-Krylov Method," *Annals of Nuclear Energy*, **87**, 707-719 (2016).
2. Bahman Zohuri and Nima Fathi, "Thermal-Hydraulic Analysis of Nuclear Reactors," (1st ed.), Springer (2015).
3. Ishii M and Hibiki T., "Thermo-fluid Dynamics of Two-Phase Flow," (2nd ed.), Springer (2011).
4. C. Demazière, "Multi-physics Modelling of Nuclear Reactors: Current Practices in a Nutshell," *International Journal of Nuclear Energy Science and Technology*, **7**, 288-318 (2013).
5. G. Yadigaroglu, "Computational Fluid Dynamics for Nuclear Applications: From CFD to Multi-Scale CMFD," *Nuclear Engineering and Design*, **235**, 153-164 (2005).
6. S. Evje and T. Flatten, "Hybrid Flux-Splitting Schemes for a Common Two-Fluid Model," *J. Comput. Phys.*, **192**, 175-210 (2003).
7. H. Städtke, "Gasdynamic Aspects of Two-Phase Flow:

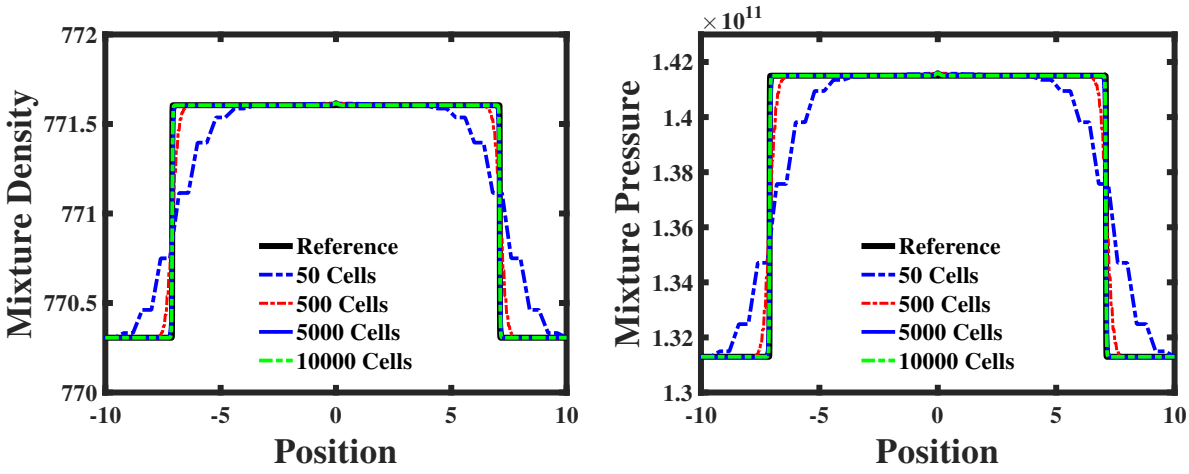

Fig. 4. Mesh refinement study for Test 3: colliding shock waves. Results for the mixture density and mixture pressure: calculated Lax-Friedrichs scheme (line symbols) and reference (solid line) at t = 0.0004 ms

Hyperbolicity, Wave Propagation Phenomena, and Related Numerical Methods," (1st ed.), Weinheim: Wiley-VCH (2006).

8. H. Ström, S. Sasic, k. Jareteg and C. Demaziere, "Behaviour and Stability of the Two-Fluid Model for Fine-Scale Simulations of Bubbly Flow in Nuclear Reactors," *Int. J. Chem. React. Eng.*, **13**, 449-459 (2015).
9. D. Bestion, "Applicability of Two-Phase CFD to Nuclear Reactor Thermalhydraulics and Elaboration of Best Practice Guidelines," *Nuclear Engineering and Design*, **253**, 11-321 (2012).
10. T. Vazquez-Gonzalez, A. Llor and Christophe Fochesato, "Ransom test Results From Various Two-Fluid Schemes: is enforcing hyperbolicity a thermodynamically consistent option?," *International Journal of Multiphase Flow*, **81**, 104-112 (2016).
11. G.-S. Yeom and K.-S. Chang, Numerical Simulation of Two-Fluid Two-Phase Flows by HLL Scheme Using an Approximate Jacobian Matrix, *Numerical Heat Transfer, Part B*, **49**, 155-177 (2006).
12. Eric Goncalvès and Dia Zeidan, "Numerical Simulation of Unsteady Cavitation in Liquid Hydrogen Flows," *International Journal of Engineering Systems Modelling and Simulation*, **9**, 41-51 (2017).
13. I. Peshkov and E. Romenski, "A Hyperbolic Model for Viscous Newtonian Flows," *Contin. Mech. Thermodyn.*, **28**, 85-104 (2016).
14. E. Romenski, A.D. Resnyansky and E.F. Toro, "Conservative Hyperbolic Formulation for Compressible Two-Phase Flow with Different Phase Pressures and Temperatures," *Quart. Appl. Math.*, **65**, 259-279 (2007).
15. D. Zeidan, "Assessment of Mixture Two-Phase Flow Equations for Volcanic Flows using Godunov-type Methods," *Applied Mathematics and Computation*, **272**, 707-719 (2016).
16. E.F. Toro, "Riemann Solvers and Numerical Methods for Fluid Dynamics: A Practical Introduction," (3rd ed.), Springer (2009).
17. S.M. Monahan, V.S. Vitankar and R.O. Fox, "CFD Predictions for Flow-Regime Transitions in Bubble Columns," *AIChE Journal*, **51**, 1897-1923 (2005).
18. K. Jareteg, P. Vinai, S. Sasic and C. Demazière, "Coupled Fine-Mesh Neutronics and Thermal-Hydraulics - Modeling and Implementation for PWR Fuel Assemblies," *Annals of Nuclear Energy*, **84**, 244-257 (2015).
19. D. Zeidan, "The Riemann Problem for a Hyperbolic Model of Two-Phase Flow in Conservative Form," *International Journal of Computational Fluid Dynamics*, **25**, 299-318 (2011).
20. C.E. Castro and E.F. Toro, "A Riemann Solver and Upwind Methods for a Two-Phase Flow Model in Non-Conservative Form," *International Journal for Numerical Methods in Fluids*, **50**, 275-307 (2006).
21. M. Souli, R. Messahel, B. Cohen and N. Aquelet, "Numerical Investigation of Phase Change and Cavitation Effects in Nuclear Power Plant Pipes ," 13th International LS-DYNA Users Conference, Detroit, USA, 1-10 (2014).
22. R.F. Saxe and L.W. Lau, "Cavitation Noise in Nuclear Reactors," *Nuclear Engineering and Design*, **8**, 229-240 (1968).
23. S.C. Sutradhar, "Shock Wave Propagation Through the Pressure Tubes of a CANDU-type Nuclear Reactor," PhD thesis, McMaster University, Canada (1985).
24. M. Baudin, C. Berthon, F. Coquel, R. Masson and Q. H. Tran, "A Relaxation Method for Two-Phase Flow Models with Hydrodynamic Closure Laws," *Numerische Mathematik*, **99**, 411-440 (2005).
25. S. T. Munkejord, S. Evje, T. Flatten, "The Multi-stage Centred-Scheme Approach Applied to a Drift-Flux Two-Phase Flow Model," *International Journal for Numerical Methods in Fluids*, **52**, 679-705 (2006).