Numerical assessment of the condensation shock mechanism in sheet to cloud cavitation transition

Diego Vaca-Revelo¹, Aswin Gnanaskandan ∗,2

Mechanical and Materials Engineering, Worcester Polytechnic Institute, United States of America

A R T I C L E I N F O

Keywords:
- Sheet to cloud cavitation
- Cavity auto-oscillation
- Condensation shock

A B S T R A C T

Sheet to cloud cavitation over a wedge is investigated using numerical simulations performed at a Reynolds number $Re = 200,000$ and cavitation numbers ranging from $\sigma = 1.44$ to $2.18$. The multiphase fluid is described using a homogeneous mixture model, and the governing equations are the compressible Navier–Stokes equations for the liquid/vapor mixture along with a transport equation for the vapor mass fraction. The solution methodology uses a predictor–corrector method on an unstructured grid wherein the predictor step uses a non-dissipative algorithm, and the corrector step uses characteristic-based filtering for shock and interface capturing. The numerical method is first validated by comparing it with the experimental measurements, showing good agreement. A systematic parametric investigation of the sheet to cloud cavitation phenomenon for different cavitation numbers is then carried out. The two primary mechanisms known to destabilize the sheet cavity viz. the re-entrant jet mechanism and the condensation shock mechanism are captured in the simulations. The role of condensation shocks as a mechanism to destabilize the sheet cavity has been demonstrated adequately in the past using both experimental and numerical studies. However, an in-depth knowledge of how condensation shocks form and the physical conditions that favor their formation has not been elucidated yet. Hence, this investigation focuses on analyzing the condensation shock mechanism with a main focus on elucidating its origin. The numerical results suggest that during the early stages of the condensation shock shedding cycle, a liquid re-entrant jet is formed at the cavity closure due to an adverse pressure gradient as in the well-investigated re-entrant jet mechanism. Then, the pressure waves, formed due to the collapse of cloud cavities downstream, impinge on the cavity closure. This causes the re-entrant jet to accelerate to supersonic speeds due to which a condensation shock is initiated inside the cavity. Then, the condensation shock travels inside the vapor cavity causing destabilization of the sheet cavity.

1. Introduction

Cavitation is a phase change phenomenon in which a liquid is converted to vapor when the pressure is reduced below the saturated vapor pressure of the liquid. Cavitation has both beneficial (Feng et al., 2015; Avvaru et al., 2018; Gnanaskandan et al., 2019; Maeda and Colonius, 2019) and detrimental effects (Karimi and Martin, 1986; Kim et al., 2014; Köksal et al., 2021). The vapor formation in cavitation is often followed by a growth of the vapor cavity and its violent collapse under high-pressure conditions. The physical consequences of this collapse include noise, vibration, surface erosion, and compromised structural integrity of the propeller. The detrimental impact of cavi-

https://doi.org/10.1016/j.ijmultiphaseflow.2023.104616

Received 23 June 2023; Received in revised form 22 August 2023; Accepted 14 September 2023
Available online 21 September 2023
0301-9322/© 2023 Elsevier Ltd. All rights reserved.
condensation shock, both of which are illustrated in Fig. 1. The liquid re-entrant jet mechanism has been well investigated by various experimental and computational studies (Kawanami et al., 1997; Arndt et al., 2000; Callenaere et al., 2001; Laberteaux and Cecicio, 2001; Leroux et al., 2004; Coutier-Delgosha et al., 2003; Ji et al., 2014; Gnanaskandan and Mahesh, 2016b). A sufficiently high adverse pressure gradient at the cavity closure is necessary for a re-entrant jet to develop, which travels upstream, splits the vapor cavity, and results in cloud formation and shedding. Kawanami et al. (1997) verified the role of the re-entrant jet on a hydrofoil and was able to stop the sheet to cloud transition by placing an obstacle preventing the jet development, thus conclusively proving the re-entrant jet as a major mechanism for the sheet to cloud cavitation transition. Recent studies have proposed Eulerian–Lagrangian multi-scale models to couple the macroscopic and microscopic structures that coexist in cavitation flows (Wang et al., 2021a, 2023; Ghahramani et al., 2021; Li et al., 2023). Wang et al. (2021a) have captured the re-entrant jet mechanism in their multi-scale numerical study, and observed that the liquid re-entrant jet significantly influences the bubble dynamics, bubble sizes, distributions and erosion effects.

At lower cavitation numbers, \( \sigma = (\rho - \rho_i)/0.5\rho_i u_{\infty}^2 \) (where \( \rho_i \) is the saturated vapor pressure), many authors (Arndt et al., 2000; Ganesh et al., 2016; Budich et al., 2018; Wu et al., 2019; Bhatt and Mahesh, 2020; Trummler et al., 2020; Vaca-Revelo and Gnanaskandan, 2022; Bhatt et al., 2023) have observed the presence of condensation shocks, also called bubbly shocks. The experiments of Ganesh et al. (2016) attributed the presence of a condensation shock as a mechanism for the transition from sheet to cloud cavitation. At sufficiently low cavitation numbers, once the sheet cavity reaches a critical length, a shock front is initiated inside the vapor cavity. This bubbly shock travels upstream and destabilizes the sheet cavity to form a cloud cavity (Ganesh et al., 2016). Budich et al. (2018) demonstrated the presence of a bubbly shock inside the sheet cavity using their numerical model and showed that the flow properties across these shocks obeyed the Rankine–Hugoniot jump conditions. They proposed that both pressure waves and the high adverse pressure gradient can trigger condensation shock regime, and high adverse pressure gradients at the cavity closure are particularly associated with the re-entrant jet formation. Wu et al. (2019) reported that the transition from sheet to cloud cavitation on a hydrofoil is influenced by the pressure waves formed due to the collapse of cavity clouds downstream. Bhatt and Mahesh (2020) simulated the same configuration as Ganesh et al. (2016) using Large Eddy Simulation (LES) to capture the condensation shock mechanism while pointing out differences in streamline curvature and pressure recovery between the re-entrant jet and condensation shock mechanisms. According to them, a higher streamline curvature leads to a higher adverse pressure gradient which supports the re-entrant jet mechanism, and lower pressure recovery benefits condensation shock formation. Wang et al. (2021b) numerically compared the re-entrant jet and the bubbly shock mechanisms on a hydrofoil using the LES approach with a dynamic cubic nonlinear subgrid-scale model, and reported that the shedding frequency is higher for the re-entrant jet-based cavity destabilization than the condensation shock-based destabilization. Additionally, they also applied the one-dimensional Rankine–Hugoniot jump conditions to verify the existence of condensation shock. Recently, Bhatt et al. (2023) experimentally studied the cavity dynamics on a hydrofoil and observed the previously reported near-surface liquid re-entrant jet and bubbly shock mechanisms. They quantified the probability to observe re-entrant jet and condensation shock shedding cycles at constant flow conditions and reported four flow regimes of transition from sheet to cloud cavitation. They found that it was more likely to find a re-entrant jet mechanism at high cavitation numbers and as \( \sigma \) was reduced, it was more probable to observe a condensation shock mechanism. Both mechanisms were found to occur simultaneously at intermediate cavitation numbers. Trummler et al. (2020) investigated the dynamics of condensation shocks and re-entrant jets in a constant throat area nozzle using LES and suggested qualitatively that the re-entrant transforms into a condensation shock as it moves upstream. However, no quantitative arguments were made in their study to corroborate this.

Although the origin of the re-entrant jet mechanism, its evolution, and the resultant cavity shedding have been rigorously documented and described, the condensation shock mechanism has not yet received the same amount of attention. Recent experimental studies (Ganesh et al., 2016; Wu et al., 2019; Bhatt et al., 2023) and numerical studies (Budich et al., 2018; Bhatt and Mahesh, 2020; Trummler et al., 2020) have focused mainly on demonstrating the existence of condensation shock and its role in cavity destabilization. However, the fundamental knowledge behind how the mechanism transitions from a re-entrant jet to a condensation shock has not yet been clearly elucidated. The origin of the condensation shock has been attributed to the impingement of pressure waves induced by the collapse of the cloud cavities downstream on the closure of the cavity (Bhatt and Mahesh, 2020; Wu et al., 2019; Wang et al., 2021b). However, in any of these studies, it was not clearly demonstrated whether condensation shocks initiate at cavity closure. Therefore, the main objective of this study is to shed light on the formation of condensation shocks and to determine the conditions that lead to their formation. This is achieved through a series of numerical simulations over a range of cavitation numbers, \( \sigma = 1.44, 1.62, 1.69, 1.76, 1.93, 2.08 \) and 2.18, covering both the re-entrant jet and condensation shock mechanism. Although it is widely accepted that these days that Large Eddy Simulations (LES) yield better predictions of the transition from sheet to cloud cavitation (Gnanaskandan and Mahesh, 2016a; Mahesh et al., 2015), we mainly use the Unsteady Reynolds Averaged Navier–Stokes equations (URANS) in this study, primarily to carry out a large number of parametric simulations, which would not be feasible with 3D LES that resolves the large scale eddies which are three dimensional in nature. However, it is important to mention that we use the LES approach at one cavitation number (\( \sigma = 1.39 \)) to verify our observations.

The paper is organized as follows. In Section 2, we present the governing equations and numerical methods used. Then the problem configuration, grid, and boundary conditions are discussed in Section 3. After a validation study of our model with experiments, the results of the simulated cases are analyzed and reported in detail in Section 4. A brief summary is then provided in Section 5.

2. Governing equations and numerical method

The multiphase medium is represented using a finite rate homogeneous mixture model (Gnanaskandan and Mahesh, 2015b) that assumes mechanical and thermal equilibrium between the liquid and vapor phases. The governing equations are the compressible Navier–Stokes
equations for the liquid/vapor mixture along with a transport equation for the vapor mass fraction. The governing equations are:

\[ \frac{\partial \rho}{\partial t} = -\frac{\partial}{\partial x_k}(\rho u_k), \]
\[ \frac{\partial \rho u_i}{\partial t} = -\frac{\partial}{\partial x_k}(\rho u_i u_k + \rho \delta_{ik} - \sigma_{ik}), \]
\[ \frac{\partial \rho e_i}{\partial t} = -\frac{\partial}{\partial x_k} \left( \rho e_i u_k - Q_k \right) - \rho \frac{\partial \delta_{ik}}{\partial x_k} + \sigma_{ik} \frac{\partial u_i}{\partial x_k}, \]
\[ \frac{\partial \gamma y}{\partial t} = -\frac{\partial}{\partial x_k} \left( \rho \gamma y u_k - \delta_{ik} \right) + S_k - S_y, \]

where \( \rho, u_i, e_i, \) and \( p \) are density, velocity, internal energy, and pressure of the mixture respectively and \( Y \) is the vapor mass fraction. The mixture density in a homogeneous mixture model is given by:

\[ \rho = \rho_l(1 - \alpha) + \rho_v \alpha, \]

where \( \rho_l \) and \( \rho_v \) are the density of the liquid and vapor, respectively. \( \alpha \) is the vapor volume fraction and is related to the vapor mass fraction by:

\[ \rho_l(1 - \alpha) = \rho(1 - Y) \quad \text{and} \quad \rho_v \alpha = \rho Y. \]

The mixture equation of state is given by Seo and Lele (2009)

\[ p = \frac{Y \rho R_e T}{\gamma} + \frac{1 - Y}{\gamma} \right) \frac{P}{\rho + p}, \]

where \( R_e = 461.6 \) J/KgK, \( K_t = 2684.075 \) J/KgK, and \( P_e = 786.33 \) MPa are constants associated with the equation of state of vapor and liquid.

The equation to evaluate internal energy is given by

\[ \rho e + \rho C_{in} T + \rho(1 - Y) \frac{P}{\rho + p} R_e T, \]

\[ C_{in} = (1 - Y) C_{lv} + Y C_{dv}. \]

\( C_{lv} \) and \( C_{dv} \) are the specific heats at constant volume for liquid and vapor respectively. The viscous stress \( \sigma_{ij} \) and heat flux \( Q_i \) are given by

\[ \sigma_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right), \]
\[ Q_i = k \frac{\partial y}{\partial x_i}, \]

where \( \mu \) is the mixture viscosity and \( k \) is the mixture thermal conductivity. The source terms for evaporation of liquid \( \sigma_{lv} \) and condensation of vapor \( \sigma_{vl} \) are given by Saito et al. (2007)

\[ \sigma_{lv} = C_s \frac{a^2(1 - \alpha)^2 \rho_l \max(\rho_l - \rho_v, 0)}{\sqrt{T_a R_e T}}, \]
\[ \sigma_{vl} = C_s \frac{a^2(1 - \alpha)^2 \max(\rho_v - \rho_l, 0)}{\sqrt{T_a R_e T}}, \]

where \( \rho_l \) is the vapor pressure, \( C_s \) and \( C_v \) are empirical constants whose value is 0.1. The vapor pressure can be calculated by the following empirical expression (Sugawara, 1932)

\[ p_v = p_v \exp \left\{ \left( \frac{1 - T_a}{T} \right) \left( a + (b - c T_a)(T - d T_a)^2 \right) \right\}, \]
\[ a = 7.21, \quad b = 1.152 \times 10^{-5}, \quad c = -4.787 \times 10^{-7}, \quad d = 483.16. \]

The model used in this study does not consider dissolved non-condensable gases. Past experimental studies (Mäkiharju et al., 2017) on a similar wedge configuration have shown that the effect of non-condensable gases from the free steam on the formation of condensation shock waves is negligible. However, the artificial injection of non-condensable gas near the apex or at the mid-cavity caused the reduction of vapor formation because of the pressure increase in the suction area. Since this study addresses cavity destabilization in the absence of controlled injection, the effect of non-condensable gases can be considered negligible based on the existing experimental study. Additionally, Brandao et al. (2020) numerically studied flow over a circular cylinder considering non-condensable dissolved gas and observed that the non-condensable gases do not affect the condensation shock wave formation, but only the pressure rise across it.

To perform LES, Eqs. (1) are first Favre filtered spatially:

\[ \frac{\partial \tilde{\rho}}{\partial t} = -\frac{\partial}{\partial x_k} \left( \tilde{\rho} \tilde{u}_k \right), \]
\[ \frac{\partial \tilde{\sigma}_{ij}}{\partial t} = -\frac{\partial}{\partial x_k} \left( \tilde{\rho} u_i \tilde{u}_k - \delta_{ik} \right), \]
\[ \frac{\partial \tilde{\gamma} Y}{\partial t} = -\frac{\partial}{\partial x_k} \left( \tilde{\rho} Y u_k - \delta_{ik} \right) + S_k - S_y, \]

The tilde quantities are Favre averaged quantities and \( \tau_{kk}, q_k \) and \( \tau_i \) are subgrid scale (SGS) stresses, SGS heat flux and SGS scalar flux. These terms are modeled using the Dynamic Smagorinsky model (DSM);

\[ \tau_{ij} = \frac{\delta}{\tau} \tau_{kk} - 2 C_S \left| \tilde{S} \right| \tilde{S}_{ij}, \]
\[ \tau_{kk} = 2 C_S \left| \tilde{S} \right| \tilde{S}, \]

where \( \tilde{S} = \sqrt{\tilde{S}_{ij} \tilde{S}_{ij}} \) and \( \tilde{S}_{ij} = \tilde{S}_{ij} - 1/3 \tilde{S}_{kk} \delta_{ij} \). The model coefficients \( C_s, \tau_k, Pr_T \) and \( Sc_T \) are determined using the Germano identity. For example,

\[ C_s \tilde{S} = \frac{1}{2} \left( \tilde{M}^*_s \tilde{M}^*_s \right), \]
\[ \tilde{M}^*_s = \frac{\left( \tilde{\rho} u_i - \tilde{\rho} \tilde{u}_i \right)}{\tilde{\rho}} - \tilde{\rho} \tilde{u}_i \tilde{u}_i, \]

where \( \tilde{\rho} \) denotes spatial average over neighboring control volumes and the caret denotes test filtering. Test filtering is defined by the linear interpolation from face values of a control volume, which is again the interpolation from two adjacent cell center values (Park and Mahesh, 2007).

For URANS, we use the Spalart–Allmaras (Spalart and Garbaruk, 2020) model given by

\[ \frac{\partial \rho \tilde{e}}{\partial t} + \frac{\partial (\rho \tilde{e} u_k)}{\partial x_k} = \frac{\tilde{e}}{\sigma} \frac{\partial}{\partial x_k} \tilde{e} \left( \frac{\partial \tilde{e}}{\partial x_k} \right)^2. \]

where \( \nu_T = \nu_{l1} f_{l1}, f_{l1} = \chi^2 / (\chi^2 + c_{w1}) \) and \( \chi = \nu_{l1} / \nu_s. \) \( S \) is the strain rate. The model is closed with the following coefficients and wall functions:

\[ \tilde{S} + \frac{\nu}{k} f_{l1} = \frac{1}{\chi}, \]
\[ f_{l1} = \frac{1}{c_{w1}^2}, \]
\[ f_w = \frac{1 + c_{w1}^2}{2}, \]
\[ r = \frac{\nu}{k} f_{l1}, \]
\[ \tau_k = 0.135, \]
\[ c_{w1} = 0.622, \]
\[ c_{w2} = 0.3, \]
\[ c_{w3} = 2, \]
\[ c_{w4} = 7.1, \]

Additionally, we use the eddy viscosity modification suggested by Coutier-Delgosha et al. (2003), who observed that the eddy viscosity
evaluated from standard RANS models can be excessive and prevent cloud formation. Hence, the modified eddy viscosity is given by
\[ \nu_T = \nu_F \{ \rho_F + (\rho_1 - \rho_F)(1 - a)^{10} \}. \] (14)

The turbulent thermal conductivity and turbulent scalar diffusivity are computed assuming a turbulent Prandtl number \( (Pr_T) \) of 0.9 and a turbulent Schmidt number \( (Sc_T) \) of 0.7. The turbulent scalar equation is then modified as
\[ \frac{\partial \phi}{\partial t} = \nabla \cdot \left( \Gamma \nabla \phi \right) + S_\phi + \frac{\partial}{\partial x_k} \left( \Gamma \frac{\partial \phi}{\partial x_k} \right). \] (15)

The simulations use the algorithm developed by Gnanaskandan and Mahesh (2015b) for cavitating flows on unstructured grids. The algorithm makes use of a novel predictor–corrector approach to handle shocks and material interfaces in the flow with minimal dissipation. In the predictor step, the governing equations are discretized using a symmetric non-dissipative scheme, where the fluxes at a cell face are computed assuming a turbulent Prandtl number \( (Pr_T) \) of 0.9 and a turbulent Schmidt number \( (Sc_T) \) of 0.7. The turbulent scalar equation is then modified as
\[ \frac{\partial \phi}{\partial t} = \nabla \cdot \left( \Gamma \nabla \phi \right) + S_\phi + \frac{\partial}{\partial x_k} \left( \Gamma \frac{\partial \phi}{\partial x_k} \right). \] (15)

The simulations use the algorithm developed by Gnanaskandan and Mahesh (2015b) for cavitating flows on unstructured grids. The algorithm makes use of a novel predictor–corrector approach to handle shocks and material interfaces in the flow with minimal dissipation. In the predictor step, the governing equations are discretized using a symmetric non-dissipative scheme, where the fluxes at a cell face are given by
\[ \phi_{i+1} = \phi_{i} + \frac{\Delta t}{\Delta x} \left[ \gamma_{i+1} \phi_{i} - \gamma_{i} \phi_{i-1} \right], \] (17)

where \( \gamma_{i} \) denotes the \( i \)th component of the right hand side of the governing equations, and the superscript \( n \) denotes the \( n \)th time step. The final solution \( \phi_{i+1} \) at \( t + \Delta t \) is obtained from a corrector scheme
\[ \phi_{i+1} = \phi_{i} + \Delta t \sum_{faces} (F_j^e \cdot \hat{A}^f), \] (18)

where \( F_j^e \) is the filter numerical flux of the following form:
\[ F_j^e = \frac{1}{2} R_j^e \Phi_j^e. \] (19)

Here \( R_j^e \) is the right eigenvector vector at the face computed using the Roe-average of the variables from left and right control volumes. The expression for the \( l \)th component of \( \Phi^f \), \( \Phi^e \) is given by
\[ \phi_{i+1}^l = k \theta_{j}^l \Phi_{j}^e, \] (20)

where \( k \) is an adjustable parameter and \( \theta_{j}^l \) is the Harten’s switch function given by
\[ \theta_{j}^l = \sqrt{0.5 \left( \hat{\theta}_{j+1}^l + \hat{\theta}_{j-1}^l \right)}, \] (21)

where \( \hat{\theta}_{j+1}^l \) denotes the \( j+1 \)th element of the Jacobian matrix. \( \hat{\theta}_{j+1}^l \) is an element of the Jacobian matrix. Park and Mahesh (2007) and Gnanaskandan and Mahesh (2015b) proposed a modification to Harten’s switch to accurately represent under-resolved turbulence for single phase and multi phase flow mixtures respectively by multiplying \( \theta_{j}^l \) with \( \theta_{j}^* \) given by
\[ \theta_{j}^* = \frac{1}{2} \left( \theta_{j+1}^* + \theta_{j-1}^* \right) + \left( a_{j+1} - a_{j-1} \right). \] (22)
compute the operating cavitation number (see Fig. 2 for the locations on another plane (E2) at 3. It is expected that the major vapor production will take place along the length of the wedge, so the grid is made very fine in this region. For the 2D computational domain, the minimum grid spacing near the wedge is 0.001h x 0.001h in the normal and streamwise directions, respectively (see Fig. 3). The wall-normal spacing stretches to 0.005h at a height of 0.5h from the apex of the wedge and further to about 0.01h at a height of h from the apex. In the streamwise direction, the grid is stretched to 0.02h at 3.5h from the apex and further to 0.01h at the end of the wedge. This grid configuration was reached by carefully conducting a grid sensitivity study and balancing between accuracy and cost. The sufficiency of this grid resolution is demonstrated by the satisfactory agreement between the experimental and numerical results. On the other hand for the 3D computational domain, the minimum grid spacing near the wedge is 0.005h x 0.005h x 0.025h in the normal, streamwise and spanwise directions, respectively (see Fig. 4). The wall-normal spacing stretches to 0.02h at a height of 0.5h. In the streamwise direction, the grid is stretched to 0.05h at a distance of 3.5h. A non-dimensional time step of 1 x 10^-6 u∞/h is used for the simulations; this corresponds to a dimensional time of 3.2 nanoseconds. To numerically simulate one shedding cycle, it took approximately 1 day with the 2D unsteady RANS method using 480 processors. For the unsteady RANS approach using 128 processors, and 2 weeks with the 3D LES method using 480 processors. A justifiable starting point for this study is to reach two of the study cases Shedding mechanism Inlet velocity e L/h T/h St

<table>
<thead>
<tr>
<th>Study case</th>
<th>Shedding mechanism</th>
<th>Inlet velocity</th>
<th>e</th>
<th>L/h</th>
<th>T/h</th>
<th>St</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Shock</td>
<td>7.98 m/s</td>
<td>1.44</td>
<td>8.23</td>
<td>0.99</td>
<td>0.055</td>
</tr>
<tr>
<td>B</td>
<td>Shock</td>
<td>7.97 m/s</td>
<td>1.62</td>
<td>5.39</td>
<td>0.68</td>
<td>0.073</td>
</tr>
<tr>
<td>C</td>
<td>Shock &amp; Jet</td>
<td>7.97 m/s</td>
<td>1.69</td>
<td>4.58</td>
<td>0.59</td>
<td>0.123</td>
</tr>
<tr>
<td>D</td>
<td>Shock &amp; Jet</td>
<td>7.97 m/s</td>
<td>1.76</td>
<td>3.70</td>
<td>0.54</td>
<td>0.161</td>
</tr>
<tr>
<td>E</td>
<td>Jet</td>
<td>7.91 m/s</td>
<td>2.08</td>
<td>2.39</td>
<td>0.34</td>
<td>0.184</td>
</tr>
<tr>
<td>F</td>
<td>Jet</td>
<td>7.89 m/s</td>
<td>2.18</td>
<td>1.91</td>
<td>0.30</td>
<td>0.220</td>
</tr>
</tbody>
</table>

Table 1: Details of the simulations conducted using the unsteady RANS method.

It is estimated from the power spectral density computed using the fast Fourier transform of the pressure history obtained at specific points inside the partial cavity and in the wake of the cavity. For example, Fig. 6-a shows the time evolution of pressure at x/h = 4.5 at station y/h = 5. Fig. 6-b shows its corresponding spectrum in the frequency domain, and the dominant frequency is exhibited at St = 0.055. An identical methodology is used for St computation for the other cases and the results are shown in Table 1. With a reduction in e, the shedding frequency reduces, and the length and thickness of the cavity increase; in other words, the period needed to complete a shedding cycle increases. For cases A and B, 17 and 13 shedding cycles are simulated respectively; all of them show condensation shock mechanism. For cases E and F, 11 and 16 cycles are simulated with all of them exhibiting purely re-entrant jet mechanism. For case C, 10 destabilization cycles are simulated, 8 of which are due to the condensation shock mechanism; for case D, 13 cycles are simulated, and we observe the condensation shock mechanism in 6 of them.
4.1. Comparison to experiment

Validation of the unsteady RANS results is first carried out by comparison with the experiments reported by Ganesh et al. (2016) at \(\sigma_{\text{exp}} = 1.73\) and \(\sigma_{\text{exp}} = 2.06\). At \(\sigma_{\text{exp}} = 1.73\), the condensation shock mechanism was reported, and at \(\sigma_{\text{exp}} = 2.06\), the re-entrant jet mechanism was observed. The simulated time-averaged values of the void fraction at \(\sigma = 1.69\) shown in Fig. 7-a correspond to ten shedding cycles and have approximately 11,500 samples. Vertical profiles extracted at three different streamwise locations \(x/h = 0.7\), \(x/h = 1.4\), and \(x/h = 2.8\) are compared with the experiments, which are shown in Figs. 7-b to 7-d. Note that the values of mean void fraction predicted by the simulation show close agreement with the experimental values, thus providing confidence in the capability of the unsteady RANS simulations in the condensation shock regime.

Additional validation is also carried out in the re-entrant jet regime \(\sigma = 2.08\) and the results are shown in Fig. 8. The predicted time-averaged void fraction profiles at \(\sigma = 2.08\) are extracted at stations \(x/h = 0.5\), \(x/h = 1.0\), and \(x/h = 2.0\) and compared with the experimental measurements. The thickness and length of the simulated cavity agree well with the experiments. We use the root mean square deviation (RMSD) to quantitatively compare the experimental and simulated profiles. The RMSD is evaluated using equation (24), where \(N\) is the number of data points, \(e_{\text{exp}}\), and \(e_{\text{num}}\) are the experimental and numerically estimated volume fraction of vapor, respectively. The RMSD of vapor volume fraction ranges from 0.02 to 0.06 at \(\sigma = 1.69\) and from 0.02 to 0.08 at \(\sigma = 2.08\).

\[
\text{RMSD} = \sqrt{\frac{\sum_{i=1}^{N} (a_{\text{exp},i} - a_{\text{num},i})^2}{N}}
\]

The two-dimensional assumption of the computational domain will induce an error in the validation because the experiment considers the effects of the streamwise vorticity due to the three-dimensional nature of the experimental setup; however, the streamwise vorticity is not directly resolved in our two-dimensional simulations. Fig. 9 shows a comparison of the dynamic pressure evolution on the wedge surface at a location \(x/h = 1.76\) between experiments and simulation at \(\sigma = 2.08\). The intrinsic cyclic behavior of the time series and their fluctuation range exhibit similarity between the curves. The fast Fourier transform (FFT) of the simulated values shows a dominant frequency at approximately 56 Hz which is in agreement with the experimental dominant frequency of approximately 50 Hz. The differences in the magnitude of pressure peaks arise from the fact that our simulations are 2D and it is well known that a 3D spherical collapse is stronger than a 2D cylindrical collapse (Bhatt et al., 2015). Having gained confidence that the numerical method predicts both the re-entrant jet and condensation shock mechanisms well, we then proceed to examine the condensation shock cases in detail.

4.2. Time evolution of the cavity during the condensation shock destabilization cycle

Our numerical results confirm that the attached sheet cavity forms at the apex wedge and grows to a critical length, after which it sheds into a cloud cavity. A series of snapshots that capture the vapor shed due to bubbly shock is shown in Fig. 10. We define a normalized \(r^*\) given by

\[
r^* = \frac{t - t_i}{t_j - t_i}.
\]

where \(t_s\) is the instant when the maximum cavity length is reached, and \(t_i\) is the instant when the sheet cavity pinches off into a cloud cavity. In the first snapshot at \(r^* = 0.15\), the instantaneous void fraction values show high levels of vapor across the entire cavity, which follows the observations of Ganesh et al. (2016). Then, a shock-like front is seen to move upstream until it reaches the wedge apex (see Figs. 10-b to 10-e), and finally the attached sheet cavity transitions into a vapor cloud that is convected downstream (see Figs. 10-f). Note that as the shock front moves upstream, the void fraction ahead of the front is higher than
Fig. 9. Time evolution of dynamic pressure on the wedge surface at \( s/h = 1.76 \); (a) experimental data reported by Ganesh et al. (2016) and (b) numerically simulated values at \( \sigma = 2.08 \).

Fig. 10. Instantaneous void fraction contours (\( \sigma = 1.44 \)) showing the time evolution of cavity destabilization and shedding due to the condensation shock mechanism at \( t^* = (a) 0.15, (b) 0.30, (c) 0.50, (d) 0.81, (e) 1.08 \) and \( (f) 1.38 \). (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

Fig. 11. Evolution of the instantaneous volume fraction of vapor taken on a line parallel to the wedge surface at a normal distance \( n/h = 0.16 \) at \( \sigma = 1.44 \) showing approximately 3 shedding cycles. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

Fig. 12. Schematic of Rankine–Hugoniot jump condition computation across a 2D shock wave.

The values behind it because the front condenses the vapor as it moves upstream. The time evolution of the cycle can also be plotted using a \( \text{s-t} \) diagram obtained from void fraction data on a line parallel to the wedge surface and stacking the solution for multiple time instances as shown in Fig. 11. The triangular region indicates one cycle of cavity shedding. The cavity growth, collapse, and the shock front within the cycle are indicated in the figure by arrows. To verify whether what we observe in Figs. 10 and 11 is a condensation shock or not, we take two points immediately ahead and behind the front and apply the Rankine–Hugoniot jump conditions.

4.2.1. Rankine–Hugoniot jump conditions

In this study, the 2D Rankine–Hugoniot jump conditions are considered due to the two-dimensional nature of the shock. Fig. 12 shows a schematic of a 2D shock \( \xi \) that moves with a velocity \( \vec{\xi} \). The points L and R at which properties are extracted must be taken immediately ahead of and behind the shock front. Krehl (2015) noted that across a 2D shock, the pressure, density, and the component of velocity normal to the shock can be related using the 1D Rankine–Hugoniot equations, and additionally the tangential velocity is conserved across the shock. The Rankine–Hugoniot jump conditions assume a planar single shock propagating within a homogeneous medium, which travels as a steady plane wave. The two-dimensional Rankine–Hugoniot jump conditions
in a frame of reference moving with the shock are given by

\[ \rho_L \cdot \frac{\partial \varepsilon_{n,L}}{\partial t} = \rho_R \cdot \frac{\partial \varepsilon_{n,R}}{\partial t}, \]
\[ \rho_L \cdot \frac{\partial^2 \varepsilon_{n,L}}{\partial t^2} + \rho_L = \rho_R \cdot \frac{\partial^2 \varepsilon_{n,R}}{\partial t^2} + \rho_R, \]
\[ \varepsilon_{n,L} = \varepsilon_R, \]
\[ e_L \cdot \frac{\partial \varepsilon_{n,L}}{\partial t} = e_R \cdot \frac{\partial \varepsilon_{n,R}}{\partial t} + \frac{\varepsilon_{R}^2}{2}, \]
\[ \rho_L \cdot \frac{\partial \varepsilon_{n,L}}{\partial t} = \rho_R \cdot \frac{\partial \varepsilon_{n,R}}{\partial t} + \frac{\varepsilon_{R}^2}{2}, \]

where \( \varepsilon_R = v_i - S \), \( \tau = (u, v) \), and \( \tau_r = (u, v) \) Here \( \tau \) denotes the unit vector normal to the shock curve \( \xi \), \( v_i \) is the velocity along the axis normal to the shock and \( v_j \) is the tangential component. The shock velocity is given by \( S = \tau_r \).

Using the equation of state (4) and the relations expressed in Eq. (26), the shock speed \( S \) can be evaluated as (Brandao et al., 2020)

\[ S = \frac{\sqrt{\rho_L (p_L - p_l)^{\frac{\gamma L}{\gamma} + 1}}}{\sqrt{\rho_R (p_R - p_l)^{\frac{\gamma R}{\gamma} + 1}}}. \] (27)

\( \gamma \) is given by

\[ \frac{1}{\gamma - 1} = C_{\text{int}} + (1 - Y)K_{r}P_{r} \]
\[ \frac{1}{\gamma - 1} = \frac{Y R_{r} (p + P_{r}) + K_{r}(1 - Y)P_{r}}{R_{r} (p + P_{r}) + K_{r}(1 - Y)P_{r}}, \] (28)

where \( C_{\text{int}} = (1 - Y)C_{i} + Y C_{g} \). Here \( C_{i} \) and \( C_{g} \) are the specific heats at constant volume for liquid and gas respectively. \( P_{r}, R_{r}, \) and \( K_{r} \) are constants associated with the equation of state whose values are given in Eq. (4). The Mach number (\( M_a \)) of the shock wave is given by \( M_a = S/a \), where \( a \) denotes the speed of sound and is given by:

\[ a^2 = \frac{C_{\text{int}}}{C_0 - \frac{C_{\text{int}}}{C_{\text{int}}}}, \]
\[ C_0 = 1 - (1 - Y)p_{r} T_{r} \frac{P_{r}}{p + P_{r}}, \]
\[ C_{\text{int}} = K_{r} Y + K_{r}(1 - Y) \frac{P_{r}}{p + P_{r}}, \]
\[ C_{\text{int}} = Y C_{g} + (1 - Y)C_{p}. \]

The study of the early stages of the condensation shock cycle will give us a better understanding of how the shock front originates in the partial cavity. In the past, condensation shock or bubbly shock was described to be initiated at the cavity closure (Bhatt and Mahesh, 2020) by the pressure waves that impinge on it. These pressure waves are thought to originate from a downstream cloud collapse. However, our results suggest that at the early stages of the condensation shock cycle, no bubbly shock is present. Fig. 13 shows snapshots of this early stage for \( \sigma = 1.44 \) and \( \sigma = 1.62 \). Figs. 13-a and 13-b correspond to the instances right after the maximum length of the sheet cavity is reached, \( t^* = 0.02 \) and 0.04. Then, Figs. 13-a2 and 13-b2, \( t^* = 0.15 \) and 0.13, show a liquid re-entrant jet close to the wedge wall that is forced back inside the cavity. This re-entrant jet is caused due to the adverse pressure gradient that exists at the cavity closure. In Figs. 13-a3 and 13-b3, we can observe an intermediate state between the initial re-entrant jet and the condensation shock initiation at approximately \( t^* = 0.30 \). Our observations suggest that this re-entrant jet is accelerated into the cavity due to the pressure waves impinging on the cavity closure. This is illustrated in Fig. 14 that shows the pressure waves in the computational domain at \( \sigma = 1.44 \) and \( \sigma = 1.62 \) at \( t^* = 0 \) and 0.15 or 0.13, respectively. The pressure waves seem to impinge on the cavity closure and accelerate the liquid re-entrant jet upstream. To quantify this acceleration and contrast it with a purely re-entrant jet case, Fig. 15 shows the velocity and pressure profiles parallel to the wedge above the surface at \( n/A = 0.04 \). for the \( \sigma = 1.44 \) and 2.08 cases at three instants: (i) before the collapse-induced pressure wave hits the cavity closure (a1 and a2), (ii) the moment when pressure waves reach the cavity closure (b1 and b2) and (iii) a subsequent instant (c1 and c2). The vertical blue lines indicate the leading edge of the re-entrant jet. The pressure values inside the cavities are relatively low, and as we pass the interface and enter the re-entrant jet regions the pressure increases significantly. In Figs. 15-a1 and b1, we observe two pressure jumps downstream in the liquid region; the first one is attributed to the stagnation pressure at
the cavity closure and the second one is the incoming collapse-induced pressure wave. In the liquid jet region of both cases, close to the interface, the low negative velocity values demonstrate the minor effect of the stagnation pressure. In Figs. 15-a2 and b2, the pressure wave has reached the cavity closure, and the acceleration of the re-entrant jet becomes evident in Figs. 15-a3 and b3. The liquid jet undergoes a more noticeable acceleration in the condensation shock case than in the re-entrant jet case reaching high negative velocity values of up to \( \frac{u}{u_\infty} = -0.7 \). Our results suggest that the collapse induced pressure wave is necessary to significantly accelerate the liquid jet and push it upstream. The jet front is accelerated and reaches higher negative velocity values when the pressure wave intensity is stronger.

This indicates that the liquid jet is accelerated upon pressure wave impingement. For a cavity destabilization that occurs due to condensation shock, it is hypothesized that this acceleration causes the re-entrant jet to reach supersonic speed which initiates the formation of a condensation shock wave. However, for re-entrant jet cycles the jet speed would not reach supersonic values. To illustrate this, we track the evolution of the re-entrant jet over time by measuring its velocity and computing the corresponding Mach number to ascertain if it attains supersonic speed. The jet speed at different instances of time is measured at several points on the re-entrant jet interface and the average speed (from the different points) at each time instance is shown in Fig. 16a for different cavitation numbers. The Mach number is

\[
\sigma = 1.44 \quad \sigma = 1.62
\]

Fig. 13. Instantaneous void fraction contours during early stage of condensation shock cycles. At \( \sigma = 1.44: t^* = (a1) 0.02, (a2) 0.15, (a3) 0.30 \) and \( \sigma = 1.62: t^* = (b1) 0.04, (b2) 0.13 \) and (b3) 0.30. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

Fig. 14. Instantaneous contours lines of pressure showing pressure waves in the domain at different instants of time and cavitation numbers. At \( \sigma = 1.44: t^* = (a1) 0, (a2) 0.15, \) and \( \sigma = 1.62: t^* = (b1) 0, (b2) 0.13 \). (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)
Fig. 15. Pressure and velocity profiles parallel to the wedge wall, \( n/h = 0.04 \) away from the surface at \( \sigma = 1.44 \) (condensation shock regime) and 2.08 (re-entrant jet regime). (a1) and (b1): instant before the collapse-induced pressure wave hits the cavity closure, (a2) and (b2): instant when the wave impinges on the closure, and (a3) and (b3): a subsequent time instant. Vertical blue line: location of the liquid jet front. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

Fig. 16. Evolution of (a) the re-entrant jet speed inside the cavity and (b) the Mach number corresponding to the jet speed. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

then calculated with the corresponding jet speed, \( Ma = V_{jet}/u_s \), and its evolution is shown in Fig. 16b, which shows that supersonic speeds are attained for cases \( \sigma = 1.44 \) and \( \sigma = 1.62 \) for which the transition from sheet to cloud cavitation occurs due to the bubbly shock mechanism. Only subsonic values are observed for \( \sigma = 2.08 \) and \( \sigma = 2.18 \) where the transition occurs due re-entrant jet mechanisms only. At \( \sigma = 1.69 \), the transition can occur due to either mechanism and it is observed that for a condensation shock cycle, supersonic speeds are attained while for the re-entrant jet cycle, subsonic speeds are maintained. The same behavior is observed for the other intermediate cavitation number \( \sigma = 1.76 \).

Hence, for the bubbly shock cycles, we observe that the re-entrant jet reaches a supersonic speed which causes the condensation shock formation. Subsequently, the Mach number drops again to subsonic speed once the shock is formed. Therefore, our results suggest that the supersonic re-entrant jet at the early stages of the condensation shock mechanism leads to the shock front formation.

4.4. Large Eddy simulation of condensation shock cycle

To verify if the observations from URANS simulations are indeed universal, one case at cavitation number \( \sigma = 1.39 \), in the condensation shock regime, is simulated using the LES approach. The rationale behind choosing a cavitation number of 1.39 is to ensure that it is low enough to trigger a condensation shock, in order to verify our hypothesis in the 3D simulations as well. The evolution of the partial cavity during a bubbly shock shedding cycle is shown in Fig. 17. Fig. 17-a1 shows the isocontours of void fraction (for \( a = 0.2 \), the
void fraction contours in the cavity at $z/h = 3$ and pressure contours in the plane $z/h = 0$. Fig. 17-a2 shows the void fraction contours on a plane parallel to the wedge at $n/h = 0.04$, and Fig. 17-a3 shows the void fraction contours at $z/h = 1.5$. The above plots are shown at six different time instants, $t^* : 0.17, 0.43, 0.57, 0.77, 1.06,$ and $1.26$ in a condensation shock cycle. The figures elucidate a classical cavity...
destabilization due to condensation shock similar to what was observed in URANS simulations, with additional spanwise disturbances along the cavity interface. It is evident in Fig. 17-d3 that a re-entrant jet is formed at the cavity closure, which then leads to a bubbly shock. The condensation shock is visible on snapshots 17-c3 and then becomes more evident in Fig. 17-d3. The spanwise evolution of the shock front is shown in Figs. 17-c2 and d2, the spanwise shape of the shock becomes more two-dimensional as it gets closer to the wedge apex. Figs. 17-e and f show the vapor cloud pinched off from the attached cavity. Finally, the cloud cavity is convected downstream and a subsequent shedding cycle starts. To compute Rankine–Hugoniot conditions, five spanwise planes are extracted at $z/h = 0.0, 0.6, 1.2, 1.8$, and $2.4$ at the instants presented in Fig. 17 before the attached cavity transitions into a cloud cavity. The Rankine–Hugoniot jump conditions are then computed across the shock/interface on these planes to verify the presence of a shock front. Since the shock front could be located at different streamwise locations in different spanwise planes, an average and variance of the Rankine–Hugoniot errors are plotted in Fig. 18. At the early stages of the condensation shock, $t^* = 0.17$ and $0.43$, errors higher than $30\%$ are shown; however, the Rankine–Hugoniot jump conditions are satisfied with acceptable errors at $t^* = 0.57$ and $0.77$. At $t^* = 0.57$, the errors are lower than $20\%$, and at $t^* = 0.77$, they are lower than $10\%$. The trends of these results are similar to the 2D unsteady RANS cases. Furthermore, it is also verified that the re-entrant jet front is accelerated to supersonic speeds due to pressure wave impingement on the cavity closure as shown in Fig. 19. The interface speed is extracted from the planes $z/h = 0.0, 0.6, 1.2, 1.8$, and $2.4$ at the presented instants. The mean and variance of these speeds are used to compute the Mach number and shown in Fig. 19.

4.5. Modified description of the bubbly shock mechanism

Our analysis suggests that a re-entrant jet is initially formed, and is accelerated mainly due to the pressure waves generated due to cloud collapse downstream, impinging on the cavity closure. The jet then accelerates to supersonic speeds and causes the formation of a condensation shock inside the cavity. Then, the shock travels upstream, and once it reaches the apex of the wedge, the sheet cavity is pinched into one or more cloud cavities. Our findings of the cavity destabilization and shedding in the wedge configuration due to condensation shock are summarized in Fig. 20. The proposed description is different from the one proposed by Bhatt and Mahesh (2020). According to them, the condensation shock mechanism is initiated due to a bubbly shock wave forming at the cavity closure. However, our results suggest the presence of a re-entrant liquid jet at the wedge wall that subsequently accelerates to supersonic speeds which initiates a shock front inside the cavity.

5. Summary

This paper is an effort to elucidate a better understanding of the sheet to cloud transition mechanisms, with special emphasis on the condensation shock mechanism. The partial cavity dynamics are investigated in a sharp wedge configuration using unsteady RANS and LES approaches. The numerical methodology is first validated by comparison with the experiments of Ganesh et al. (2016) showing good agreement in both the bubbly shock and re-entrant jet regimes.

Simulations are then performed at different cavitation numbers varying from $1.44$ to $2.18$. The re-entrant jet mechanism is observed at higher cavitation numbers while the condensation shock mechanism is found to be responsible for cavity destabilization at lower cavitation numbers. Both mechanisms are found to occur at intermediate cavitation numbers. For the condensation shock cases, the 2D Rankine–Hugoniot jump conditions across the condensation shock reveal a supersonic shock front moving upstream in the vapor cavity.

Our observations suggest that condensation shock is developed through a particular process during the early stages of the bubbly shock mechanism. We observe that initially a near-wall liquid re-entrant jet is formed at the cavity closure, then, the jet is accelerated due to the
pressure waves from the clouds collapsing downstream. The liquid jet then accelerates to supersonic speeds and triggers the formation of a condensation shock in the vapor cavity. Subsequently, the shock front moves upstream, covering the entire cavity thickness and condensing the vapor upstream. Once it reaches the apex of the wedge, the attached cavity transitions into a cloud cavity. The validated numerical simulations have thus been used to elucidate a better understanding of condensation shock formation.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Data availability

Data will be made available on request.

Acknowledgments

We gratefully acknowledge the Fulbright Scholarship awarded to DVR and the start-up funds provided by Worcester Polytechnic Institute (WPI), United States of America to AG. Computing time provided by Academic Research and Computing (ARC), WPI is also acknowledged. Additionally, this work used computing time at the Texas Advanced Computing Center (TACC) through allocation PHY230017 from the Advanced Cyberinfrastructure Coordination Ecosystem: Services & Support (ACCESS) program.

References
