Numerical Simulation of Compressible Two-Phase Condensing Flows

H. Bagheri Esfe, M. J. Kermani† and M. Saffar Avval

Department of Mechanical Engineering, Amirkabir University of Technology (Tehran Polytechnic), 15875- 4413Tehran, Iran.

†Corresponding Author Email:mkermani@aut.ac.ir

(Received December 3, 2014; accepted March 16, 2015)

ABSTRACT

In the present paper, the hybrid AUSM-van Leer scheme is extended to solve the governing equations of two-phase condensing flows. The method of moments with the classical homogeneous nucleation theory is used to model the non-equilibrium condensation phenomenon. Firstly, the hybrid method is validated using two test cases (i.e. Laval nozzle and rotor-tip cascade) and the results are compared with the MacCormack method. Then the hybrid method is used to solve two other problems (i.e. wavy channel and VKI stage). Based on the numerical results of the paper, the hybrid AUSM-van Leer scheme is an accurate method to simulate two-phase transonic flows with nucleation. If the super cooling degree reaches to its maximum value, the non-equilibrium condensation begins and wetness fraction increases suddenly. Also across a shock the wetness fraction decreases due to evaporation of the droplets.

Keywords: Non-equilibrium condensation; Nucleation; Two-phase flow; Hybrid method.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a_f$</td>
<td>speed of sound</td>
</tr>
<tr>
<td>$e_i$</td>
<td>total internal energy per unit volume</td>
</tr>
<tr>
<td>$F_k$</td>
<td>horizontal inviscid flux vector</td>
</tr>
<tr>
<td>$G_k$</td>
<td>vertical inviscid flux vector</td>
</tr>
<tr>
<td>$H$</td>
<td>total enthalpy</td>
</tr>
<tr>
<td>$h_{lg}$</td>
<td>latent heat of evaporation</td>
</tr>
<tr>
<td>$J$</td>
<td>jacobian</td>
</tr>
<tr>
<td>$J_{nuc}$</td>
<td>nucleation rate of droplets</td>
</tr>
<tr>
<td>$k_B$</td>
<td>Boltzmann constant ($=1.3807\times10^{-23}$ J/K)</td>
</tr>
<tr>
<td>$M$</td>
<td>Mach number</td>
</tr>
<tr>
<td>$m_v$</td>
<td>mass of one molecule of water</td>
</tr>
<tr>
<td>$N$</td>
<td>total number of droplets per unit mass of mixture</td>
</tr>
<tr>
<td>$P$</td>
<td>pressure</td>
</tr>
<tr>
<td>$P_s$</td>
<td>saturation pressure</td>
</tr>
<tr>
<td>$Q$</td>
<td>conservative vector</td>
</tr>
<tr>
<td>$r$</td>
<td>average radius of droplets</td>
</tr>
<tr>
<td>$r_c$</td>
<td>critical radius of droplets</td>
</tr>
<tr>
<td>$R_v$</td>
<td>vapor constant ($=461.4$ J/kg.K)</td>
</tr>
<tr>
<td>$S$</td>
<td>source term</td>
</tr>
<tr>
<td>$T$</td>
<td>temperature</td>
</tr>
<tr>
<td>$T_s$</td>
<td>saturation temperature</td>
</tr>
<tr>
<td>$u,v$</td>
<td>velocity components</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>specific heat ratio ($=1.324$ J/kg.K)</td>
</tr>
<tr>
<td>$\rho$</td>
<td>mixture density</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>surface tension</td>
</tr>
<tr>
<td>$\phi$</td>
<td>dissipation coefficient</td>
</tr>
<tr>
<td>$\chi$</td>
<td>wetness fraction</td>
</tr>
<tr>
<td>$\xi,\eta$</td>
<td>curvilinear coordinates</td>
</tr>
</tbody>
</table>

1. INTRODUCTION

Condensation is a common phenomenon in many industrial thermo-fluid problems. Examples include flows in vapor nozzles, rocket motors, steam turbines, low temperature fuel cells, and many more. In such problems, flows contain vapor as one of the species that condenses due to the removal of latent heat from the vapor species.

Commence of wide studies in two-phase vapor-liquid flows, backs to the 1970s when Mcdonald (1962) presented the theory of homogeneous nucleation. He studied this phenomenon in two thermodynamic and kinetic aspects. In the next years some experimental and numerical results related to two-phase condensing flows were presented. Moore et al. (1973) performed one of the earliest experiments related to condensation in nozzles. Then some experiments were performed on a cascade of turbine to study the phenomena associated with spontaneous condensation by

In 1984, Young presented a theoretical investigation of choking in steady, one dimensional, non-equilibrium wet steam in the nozzles (Young, 1984). Then he presented a method to solve the governing equations of wet steam flow in two and quasi three dimensional turbine cascades (Young, 1992). The mixture conservation equations were solved in an Eulerian reference frame and droplet phase was computed by integrating the relevant equations along true streamlines in a Lagrangian reference frame. White and Young (1993) presented a time marching method to predict unsteady phenomena in condensing steam flows. In 2000, White developed a numerical method for the prediction of condensing steam flow within compressible boundary layers (White, 2000). Then White and Hounslo (2000) presented a new method for modeling droplet size distributions within condensing steam flows. Also White (2003) presented a comparison of modeling methods for polydisperse wet-steam flow.

In 2002, Gerber used the classical theory of nucleation and proposed a new numerical model (Eulerian-Lagrangian) to solve the two-phase compressible flows in steam nozzles and turbine blades (Gerber, 2002). Then, Gerber and Kermani (2004) presented a pressure based Eulerian–Eulerian multi-phase model for non-equilibrium condensation in transonic steam flow. In 2007, Gerber and Mousavi investigated the effectiveness of the Quadrature Method of Moments (QMOM) in representing droplet size distributions present in the low-pressure steam turbine stages (Gerber and Mousavi 2007).

Then Halama et al. (2010) used an in-house code for simulating two-phase condensing steam by addition of Giles’s matching algorithm using Lax-Wendroff method.

Single- and two- fluid models for steam condensing flow modeling were presented by Dykas and Wroblewski (2011). In 2012, they presented the computational results of the wet steam flow through the Laval nozzles for low and high inlet pressures (Dykas and Wroblewski 2012). Also, an effective method of determination of water vapor properties was presented in the case of expansion in the nozzle at high pressures. Recently Hamidi and Kermani (2013) investigated numerical solution of a compressible two-phase two-component moist-air flow with and without shock waves. They used the equilibrium thermodynamic model to study the condensation in a one dimensional nozzle.

In the past, several upwind schemes have been developed and successfully used for the calculation of many problems. Prominent representatives of this class of algorithms are schemes based on the flux vector splitting and flux difference splitting concepts. Classical flux vector splitting methods are simple and very robust upwind techniques but they exaggerate diffusive effects which take place in shear and boundary layers. On the other hand, schemes based on flux difference splitting are very accurate for viscous calculations, but at the cost of increased computational expense. Moreover, they lack robustness for flows with strong expansions into regions of low pressure and low density.

The Advection Upstream Splitting Method (AUSM) retains the robustness and efficiency of the flux vector splitting schemes but it achieves the high accuracy attributed to schemes based on the flux difference splitting concept. The computational effort for the flux evaluation is only linearly proportional to the number of unknowns, as in the case of central differencing. Furthermore, the scheme can be easily extended to real gas calculations. In other words, the special merits of AUSM compared to other upwind schemes are the low computational complexity and the low numerical diffusion. The application to various relevant flow problems, however, has shown that the AUSM method has several deficiencies. It locally produces pressure oscillations in the vicinity of shocks. Furthermore, the scheme has a poor damping behavior for low Mach numbers which leads to spurious oscillations in the solution and affects the ability of scheme to capture flows aligned with the coordinate grids. In order to improve the shock resolution capability and the damping behavior of AUSM, in particular, a hybrid method was introduced which switches from AUSM to the van Leer scheme at shock waves (Radespiel and Kroll 1995). This ensures the well-known sharp and clean shock capturing capability of the van Leer scheme and the high resolution of slip lines and contact discontinuities through AUSM. This hybrid method has been used to simulate some single-phase flows so far (e.g. inviscid calculation of NACA 0012 airfoil, 2D supersonic flow over a circular blunt body, flow over a compression ramp, etc) (Radespiel and Kroll 1995; Halder et al. 2011).

Only few methods can simulate correctly two-phase condensing flows in a complicated geometry such as a turbine stage, because these flows experience spontaneous nucleation with a sharp discontinuity at the point which non-equilibrium condensation begins (Wilson point). In the present paper, the hybrid AUSM-van Leer scheme is extended to solve the governing equations of condensing two-phase flows in four different problems. Firstly, this method is validated using two test cases (i.e. Laval nozzle and rotor-tip cascade) and the results are compared with the MacCormack method. Then the hybrid method is used to solve two other problems (i.e. wavy channel and VK1 stage).

2. GOVERNING EQUATIONS

Assumptions using in the present study are as follows: slip velocity between droplets and vapor is
ignored due to infinitesimal radius of the droplets, condensation is homogeneous and condensing steam flow is assumed to be adiabatic and inviscid.

Two dimensional governing equations consist of Euler equations for the mixture and two additional equations for the liquid phase. These equations are shown in generalized coordinates as (Halama et al. 2010):

\[
\frac{1}{J} \frac{\partial Q}{\partial t} + \frac{\partial F_k}{\partial \xi} + \frac{\partial G_k}{\partial \eta} = S
\]

where \(Q, S, F_k\) and \(G_k\) denote the Jacobian of transformation, conservative vector, source term, horizontal and vertical flux vectors, respectively. These vectors are computed as follows:

\[
Q = \begin{bmatrix}
\rho \\
\rho u \\
\rho v \\
\rho e \\
\rho n \\
\rho N
\end{bmatrix} \quad F_k = \frac{1}{J} \begin{bmatrix}
\rho u \\
\rho u U + \xi_{\rho} P \\
\rho v V + \eta_{\rho} P \\
\rho U \\
\rho F_k U + \xi_{\rho} P U \\
\rho F_k V + \eta_{\rho} P U
\end{bmatrix} \quad G_k = \frac{1}{J} \begin{bmatrix}
\rho \rho v + \eta_{\rho} P \\
\rho \rho v + \xi_{\rho} P \\
\rho \rho v + \eta_{\rho} P \\
\rho \rho v + \xi_{\rho} P
\end{bmatrix} \quad S = \begin{bmatrix}
0 \\
0 \\
0 \\
0 \\
\frac{4}{3} \pi \rho \phi (\rho n) \frac{J_{\text{muc}}}{J_{\text{muc}}} + 3 \rho N v^2 \frac{dr}{dt}
\end{bmatrix}
\]

where \(P, e, c, \text{and} N\) are pressure, total energy per unit volume, wetness fraction and total number of droplets per unit mass of mixture, respectively. Also, \(\rho\) denotes the mixture density and \(u, v\) are velocity components for both vapor and liquid.

Based on the classical homogeneous nucleation theory, the number \(J_{\text{muc}}\) of new condensed droplets per unit volume and per second is computed as (Kermani and Gerber 2003):

\[
J_{\text{muc}} = \frac{J_{\text{cl}}}{1 + \phi} = \frac{2(\gamma - 1)}{\gamma + 1} \frac{h_{\text{fg}}}{R_v} \left( \frac{h_{\text{fg}}}{R_v} \frac{T_v}{T_v} \right) \left( \frac{h_{\text{fg}}}{R_v} \frac{T_v}{T_v} - 1 \right)
\]

\[
J_{\text{cl}} = \frac{2(\gamma - 1)}{\gamma + 1} \frac{h_{\text{fg}}}{R_v} \frac{T_v}{T_v} \left( \frac{h_{\text{fg}}}{R_v} \frac{T_v}{T_v} - 1 \right)
\]

\[
\phi = \frac{2\sigma}{\pi m_v \rho \phi} \exp \left( \frac{4\pi m_v^2 \sigma}{3 k_B T_v} \right)
\]

\[
r_{c} = \frac{2\sigma}{\rho \phi R_v T_v \ln(P / P_s)}
\]

where \(\phi, h_{\text{fg}}, \gamma\) are correction parameter, latent heat of evaporation and specific heat ratio of the vapor, respectively, \(m_v\) is molecular weight of the vapor and \(\tau\) is surface tension. In Eq. (3), \(k_B, r_c, P_s\) denote the Boltzmann constant, critical radius and saturation pressure, respectively.

### 3. NUMERICAL SOLUTION

#### 3.1. Temporal Discretization

Using a forward Euler scheme for the time derivative, Eq. (1) is written in a semi-discrete form:

\[
\frac{1}{J} \frac{Q^{n+1} - Q^n}{\Delta t} + \left( \frac{\partial F_k^n}{\partial \xi} + \frac{\partial G_k^n}{\partial \eta} \right) = S^n
\]

The value of \(Q^n\) is obtained from Eq. (4), then all the primitive variables \((\rho, u, v, e, x, N)\) at the new time step will be determined. Since this equation is explicit in time, stability of the solution is governed by the CFL condition (Anderson 1995).

#### 3.2. Spatial Discretization

Extrapolation of the primitive variables such as pressure, velocity and temperature from the cell centers to the cell faces, is performed by the MUSCL strategy (Van Leer 1979). In this part, the hybrid AUSM-van Leer scheme is used to calculate the flux vectors. The underlying idea of the approach is based on the observation that the flux vectors (Eq. (2)) consist of two physically distinct parts, namely the convective and the pressure parts (Radespiel and Kroll 1995). The horizontal flux at the east face of the control volume is defined as

\[
F_k^j = \frac{1}{2} \int_{-\frac{1}{2}}^{\frac{1}{2}} \frac{M}{J} \left[ \begin{array}{c}
\rho u \\
\rho u u + \xi_{\rho} P \\
\rho v v + \eta_{\rho} P \\
\rho v u + \xi_{\rho} P
\end{array} \right]
\]

where

\[
\phi = \frac{2(\gamma - 1)}{\gamma + 1} \frac{h_{\text{fg}}}{R_v} \frac{T_v}{T_v} \left( \frac{h_{\text{fg}}}{R_v} \frac{T_v}{T_v} - 1 \right)
\]

\[
\phi = \frac{2(\gamma - 1)}{\gamma + 1} \frac{h_{\text{fg}}}{R_v} \frac{T_v}{T_v} \left( \frac{h_{\text{fg}}}{R_v} \frac{T_v}{T_v} - 1 \right)
\]

The first term on the right-hand side of the above equation represents a Mach number-weighted average of the left and right states. The second term has a dissipative character. It is scaled by the scalar value \(\phi^i_{t_{1/2}^j}\), which is termed as dissipation coefficient.

The advection Mach number \(M_{t_{1/2}^j}\) is obtained from the relation

\[
M_{t_{1/2}^j} = M^+ + M^-
\]
where the split Mach numbers are defined as

$$
M^+ = \begin{cases}
\frac{1}{4}(M^L + 1)^2 & \text{for } M^L \geq 1 \\
0 & \text{for } M^L \leq -1
\end{cases}
$$

and

$$
M^- = \begin{cases}
0 & \text{for } M^R \geq 1 \\
\frac{1}{4}(M^R - 1)^2 & \text{for } M^R \leq -1
\end{cases}
$$

The Mach numbers $M^L$ and $M^R$ are evaluated using the left and right states, respectively, i.e.

$$
M^L = \left( \frac{u}{a_f} \right)_L, \quad M^R = \left( \frac{u}{a_f} \right)_R
$$

where $a_f$ is the speed of sound and is evaluated from the following equation (Traupel 1971):

$$
\frac{\partial \rho}{\partial P} = (1 - \chi) \frac{R}{P} \frac{dP}{dT}, \quad \frac{\partial \rho}{\partial T} = (\chi - 1) \frac{TR}{P^2}
$$

where

$$
B(T) = b_0 + \frac{b_1}{b_2 + T}
$$

and

$$
\begin{align*}
b_0 &= 9.915 \times 10^3 \\
b_1 &= 2.21 \\
b_2 &= 3.0304 \times 10^5 \\
b_3 &= 3.986 \times 10^4
\end{align*}
$$

The pressure at the east face of the control volume is obtained from the splitting:

$$
P^{+}_{i+\frac{1}{2},j} = P^+ + P^-
$$

with the split pressures given by

$$
P^+ = \begin{cases}
P^L & \text{for } M^L \geq 1 \\
\frac{1}{4} P^L (M^L + 1)^2 (2 - M^L) & \text{for } M^L < 1 \\
0 & \text{for } M^L \leq -1
\end{cases}
$$

and

$$
P^- = \begin{cases}
0 & \text{for } M^R \geq 1 \\
\frac{1}{4} P^R (M^R - 1)^2 (2 + M^R) & \text{for } M^R < 1 \\
P^R & \text{for } M^R \leq -1
\end{cases}
$$

The dissipation coefficient in Eq. (5) is defined as sum of the dissipation terms using the van Leer and AUSM schemes:

$$
\phi^+_{i+\frac{1}{2},j} = (1 - \omega) \phi^+_V + \omega \phi^A_{i+\frac{1}{2},j}
$$

where $\omega$ is the weighting factor and $\phi^+_V$ is defined as

$$
\phi^+_{i+\frac{1}{2},j} = \begin{cases}
M^L_{i+\frac{1}{2},j} & \text{for } M^L_{i+\frac{1}{2},j} \geq 1 \\
\frac{1}{2} (M^R_{i+\frac{1}{2},j} - 1)^2 & \text{for } 0 \leq M^L_{i+\frac{1}{2},j} < 1 \\
\frac{1}{2} (M^L_{i+\frac{1}{2},j} + 1)^2 & \text{for } -1 \leq M^L_{i+\frac{1}{2},j} < 0
\end{cases}
$$

When the advection Mach number $M^L_{i+\frac{1}{2},j}$ tends to zero, the dissipation term in Eq. (5) will approach zero, too. Thus, there will be some disturbances which cannot be damped by the scheme. In order to solve this problem, it is proposed that the scaling of the dissipation term of the AUSM method be modified as follows

$$
\phi^+_{i+\frac{1}{2},j} = \begin{cases}
M^L_{i+\frac{1}{2},j} & \text{for } M^L_{i+\frac{1}{2},j} \geq \delta \\
\frac{1}{2} (M^R_{i+\frac{1}{2},j} - 1)^2 & \text{for } 0 \leq M^L_{i+\frac{1}{2},j} \leq 1 \\
\frac{1}{2} (M^L_{i+\frac{1}{2},j} + 1)^2 & \text{for } -1 \leq M^L_{i+\frac{1}{2},j} \leq 0
\end{cases}
$$

where $\delta$ is a small value ($0 \leq \delta \leq 5$). Hence there will always be a sufficient amount of numerical dissipation. The weighting factor in Eq. (14) is defined as

$$
\omega = \min (v_{i,j}, v_{i+1,j})
$$

$$
v_{i+1,j} = \max (1 - \alpha, \frac{P_{i+1,j} - 2P_{i,j} + P_{i-1,j}}{P_{i+1,j} + 2P_{i,j} + P_{i-1,j}})
$$

$$
\alpha = 5
$$

Thus according to Eqs. (14), (17), it can be seen that the hybrid method switches to the van Leer scheme at shocks and high gradient regions.

### 4. RESULTS

The main validation test cases in this paper are the “Laval nozzle and rotor-tip cascade”. Accuracy of the hybrid method is compared with the MacCormack method (MacCormack 1969) which is a widely used scheme to solve transonic
compressible flows. Then the hybrid method is used to solve two other problems, i.e., wavy channel and VKI stage.

While there is no available experimental data for the “wavy channel and VKI stage”, these test cases are brought here just to demonstrate the capability of the present method. Moreover, these results can also be used for comparison purposes in future works.

Herein, the VKI stage is the most complicated problem among the presented ones (because of the presence of a relative motion between the stator and rotor). Furthermore, the wavy channel problem is studied here so as to examine the ability of the scheme to capture the key phenomena of sequential condensation and evaporation.

### 4.1 Laval Nozzle

To validate the hybrid method, firstly flow passing through the nozzle A from Moore et al. (1973) at low pressure conditions (below 1 bar) is considered. Figs. 1 and 2 show, respectively, distribution of pressure ratio and wetness fraction along the centerline of the nozzle for the conditions \( P_{T0} = 354.6 \text{K} \), \( P_{T0} = 25 \text{kPa} \) and supersonic outflow. As is apparent, the hybrid AUSM-van Leer scheme has more accurate results in comparison to the MacCormack method to predict two-phase condensing flow in the nozzle. Because the MacCormack method uses a central difference discretization, whereas the hybrid method is an upwind-biased scheme.

![Fig. 1. Distribution of pressure ratio along the centerline of the nozzle Moore A for the grid 100×20; Comparison between the results of hybrid and MacCormack methods.](image)

As can be seen in Fig. 2, there are some fluctuations in the distribution of wetness fraction along the centerline of the nozzle using the MacCormack method. Although the MacCormack method is a suitable scheme to solve one-phase compressible flows, but it is not accurate enough for two-phase condensing flows.

![Fig. 2. Distribution of wetness fraction along the centerline of the nozzle Moore A for the grid 100×20; Comparison between the results of hybrid and MacCormack methods.](image)

### 4.2 Rotor-Tip Cascade

To validate the hybrid method in a more complex situation, the experimental data reported by Bakhtar and Ebrahimi (1995) for the cascade flow have been used. The boundary conditions for the current numerical simulation have been chosen identical to those in the experimental test such as, the inflow direction is -38° with respect to horizontal axis, the inflow stagnation pressure and temperature are set to 99.9 kPa and 360.8 K, respectively.

With regard to Fig. 1, the position of the condensation shock is captured well using the hybrid method. Condensation shock is a rapid release of heat from a portion of vapor that is going to condense toward the core vapor. Hence the core vapor experiences a rapid temperature rise behaving like a shock. Just on this shock, condensation occurs and wetness fraction increases. This phenomenon occurs in the supersonic region.

Fig. 3 shows distribution of nucleation rate and supercooling degree \((T_s - T)\) along the centerline of nozzle Moore A. It can be seen that the maximum value of supercooling degree exists at the nucleation zone \((x=0.035\text{m})\), also after this zone the flow returns to the equilibrium state.

Fig. 4 illustrates Mach number and entropy contours in the nozzle Moore A. With regard to this figure, it can be seen that the flow becomes supersonic in the divergent part of the nozzle. Also after the nucleation zone, the entropy increases due to thermodynamic losses. The prevailing feature in a non-equilibrium flow is that the temperature of the phases differs. This difference in temperature is the source of irreversible heat transfer between phases. Subsequently, the entropy operation rate throughout the flow field becomes relatively substantial and is usually referred to as thermodynamic losses.
Fig. 3. Distribution of (a) nucleation rate (b) supercooling degree, along the centerline of the nozzle Moore A; \( T_0 = 354.6 \, K \), \( (P_0)_m = 25 \, kPa \) and supersonic outflow.

Fig. 4. (a) Mach number, (b) Entropy Contours in the nozzle Moore A; \( T_0 = 354.6 \, K \), \( (P_0)_m = 25 \, kPa \) and supersonic outflow.

To show the grid convergence for hybrid and MacCormack methods, distribution of wetness fraction (\( \chi \)) along curve A-B has been plotted in figure 6. As shown the grid size 498×65 is appropriate for both hybrid and MacCormack methods. Also, the distribution of \( \chi \) reveals severe fluctuations in the MacCormack method which clearly shows the superiority of the present hybrid scheme.

Fig. 5. Curve A-B in the computational domain of the cascade.

Fig. 6. Grid independency test; Distribution of wetness fraction (\( \chi \)) along the curve A-B for different grid sizes; (a) hybrid method, (b) MacCormack method.

Comparison of the pressure ratio on the suction and
pressure surfaces between numerical and experimental results is shown in Fig. 7. With regard to this figure, it can be seen that the result of the hybrid method is close to the experimental data and the condensation shock on the suction surface is captured well, whereas the MacCormack method doesn’t have enough accuracy. Also there are some numerical fluctuations in the result of the MacCormack method.

Fig. 8 illustrates Mach number and wetness fraction contours in the passage for different outlet pressures. The outflow regime is supersonic for \( P_b \) (back pressure) = 30, 45 kPa and subsonic for \( P_b = 60 \) kPa.

As shown in Fig. 8 (a), an oblique shock is formed at the trailing edge of the blade to match the outlet pressure in the supersonic outflow cases. It can be seen in Fig. 8(b) that the wetness fraction increases with the outlet pressure reduction. For example, the outlet wetness fraction increases by 223% when the outlet pressure decreases from 60 kPa to 30 kPa. Also the wetness fraction decreases across the oblique shock due to increases in the vapor temperature.

4.3 Wavy Channel

The third test case in the current paper is the flow passing through a wavy channel. The geometry and computational domain of the channel is shown in Fig. 9. Size of the grid is specified as 321 × 101 after grid-independency test.

Different contours related to condensing two-phase flow in the channel is illustrated in Fig. 10. As shown in Fig. 10 (a), supersonic pockets are formed at maximum height of the profiles on the lower wall (points a, c). With regard to Fig. 10 (b), temperature of the vapor reaches to its minimum value at the first bump (point a), thus the supercooling degree becomes maximum and non-equilibrium condensation starts from this region. Due to increase of temperature during the path a-b, the wetness fraction decreases. Afterwards during the path b-c, the wetness fraction increases due to temperature reduction and reaches to its maximum value at point c.

Mach number and wetness fraction contours in the channel related to \( P_b = 30 \) kPa are depicted in Fig. 11. As shown in this figure, if the outlet pressure decreases from 50 kPa to 30 kPa, a shock is formed on the second bump (point d) to match the outlet pressure. Also the wetness fraction increases in the computational domain due to higher value of supercooling degree in comparison to the previous case. Moreover, the wetness fraction decreases across the shock due to evaporation of the droplets.
Fig. 10. Condensing flow passing through the wavy channel; (a) Mach number contours (b) Temperature contours (c) Wetness fraction contours with the conditions.

\[ T_0 = 367 \text{ K}, (P_0)_a = 70 \text{ kPa}, P_b = 50 \text{ kPa} \]

Fig. 11. Condensing flow passing through the wavy channel; (a) Mach number contours (b) Wetness fraction contours with the conditions.

\[ T_0 = 367 \text{ K}, (P_0)_a = 70 \text{ kPa}, P_b = 30 \text{ kPa} \]

4.4 VKI Stage

The complete stage under consideration consists of the VKI stator and rotor blades. Pitch length of the rotor and stator is 71mm. Also, the angle between inlet flow and horizontal direction is -30°. Angular velocity of the rotor is 3000 rpm. Moreover, inflow stagnation pressure and temperature are set to 41.7 kPa and 357.5 K, respectively. The grid size of the stator and rotor is specified as 451×911 after grid-independency test.

Fig. 12 illustrates the contours related to two-phase condensing flow in the stage for \( P_b = 24.25 \text{ kPa} \). As shown in this figure, the flow in the stage is subsonic and condensation starts after the trailing edge of the rotor. The average value of wetness fraction at exit plane of the computational domain is 1.3%.

Fig. 13 illustrates the contours in the stage for supersonic outflow with \( P_b = 14.55 \text{ kPa} \). As pointed earlier, non-equilibrium condensation starts from the point with the maximum supercooling degree (minimum temperature). This point has been specified in Fig. 13 (a).

As shown in Fig. 13 (b), the flow in the throat of the rotor passage has been choked and an oblique shock has been formed in the trailing edge of the rotor to match the outlet pressure.

With regard to Fig. 13 (c), the wetness fraction decreases across the oblique shock due to increase of the vapor temperature. Also the average value of...
wetness fraction at exit plane of the computational domain is 3.3%. Moreover, due to higher rate of flow expansion, condensation rate and wetness fraction in the vicinity of the suction surface is more than the pressure surface.

Fig. 13. Rotor/stator interaction in condensing steam flow through VKI stage, Flow data: \( (P_0)_{\text{stator}} = 41.7 \text{kPa}, (T_0)_{\text{stator}} = 357.5 \text{K} \) \( (P_0)_{\text{rotor}} = 14.55 \text{kPa} \), (a) Temperature contours, (b) Mach number contours, (c) Wetness fraction contours.

5. CONCLUSION

The hybrid AUSM-van Leer scheme was extended to solve the governing equations of two-phase flows with nucleation. This scheme was subsequently applied to low-pressure nucleating steam flow in the Laval nozzle and the rotor-tip cascade. Reasonable agreement with experimental data was obtained. The physical features of flows with homogeneous nucleation, e.g. condensation shock, were captured well. Then the hybrid method was used to solve the wavy channel and VKI stage problems. Based on the results of the paper, following conclusions can be drawn:

- With the use of presented in-house CFD code, one may predict the condensing steam flows with generally good degree of accuracy.
- Although the MacCormack method is a widely used scheme to solve one-phase compressible flows, but it is not accurate enough for two-phase condensing flows.
- Maximum value of supercooling degree exists at the nucleation zone and after this zone the flow returns to the equilibrium state.
- Entropy of the flow field increases during the non-equilibrium condensation due to the thermodynamic losses.
- Across the shock the liquid phase rapidly evaporates due to increase of the temperature.
- The suction surface of the blade has more expansion rate in comparison to the pressure surface. Thus condensation rate and wetness fraction near the suction surface are more than those of the pressure surface.

REFERENCES


