Effect of local occurrence of non-equilibrium condensation on transonic flow field

Shigeru Matsuo*, A. B. M. Toufique Hasanb, Junji Nagao, Tokitada Hashimotoa,
Toshiaki Setoguchid, Heuy Dong Kim

aDepartment of Advanced Technology Fusion, Saga University, 1 Honjo-machi, Saga 840-8502, Japan
bDepartment of Mechanical Engineering, Bangladesh University of Engineering & Technology, Dhaka 1000, Bangladesh
cGraduate School of Science & Engineering, Saga University, 1 Honjo-machi, Saga-shi, Saga 840-8502, Japan
dInstitute of Ocean Energy, Saga University, 1 Honjo-machi, Saga-shi, Saga 840-8502, Japan
eSchool of Mechanical Engineering, Andong National University, 388 Song Cheon Dong, Andong 760-749, Korea

Abstract

The transonic flow over an airfoil is characterized by a shock wave standing on the suction surface. When non-equilibrium condensation occurs in a supersonic flow field, the flow is affected by latent heat released by it. According to previous works, it was found that the occurrence of non-equilibrium condensation just before the shock wave led to reduction of shock strength. In these studies, non-equilibrium condensation occurs across the passage of the nozzle and it causes the total pressure loss in the flow field. However, local occurrence of non-equilibrium condensation in the flow field may change the characteristics of total pressure loss compared with that by non-equilibrium condensation across the passage of the nozzle and there are few for researches of locally occurred non-equilibrium condensation in a transonic flow field. The purpose of this study is to clarify the effect of local occurrence of non-equilibrium condensation on transonic flow field with a circular bump. As a result, shock strength and total pressure loss decrease with a decrease of generation region of condensate droplets.

© 2013 The Authors. Published by Elsevier Ltd. Selection and/or peer-review under responsibility of the Bangladesh Society of Mechanical Engineers

Keywords: Compressible flow, Shock wave, Control, Transonic Flow, Non-equilibrium Condensation, Simulation

Nomenclature

- $C_p$: specific heat at constant pressure (J/kg·K)
- $E$, $F$: numerical flux
- $g$: condensate mass fraction
- $H$: source term of turbulence
- $H^*$: height of nozzle throat
- $I$: nucleation rate (1/(m³K))
- $k$: specific turbulence kinetic energy
- $L$: distance from throat (m)
- $M$: Mach number
- $p$: static pressure (Pa)
- $Q$: source term of condensation
- $R$, $S$: Source term of viscosity
- $R'$: radius of wall curvature (m)
- $S$: initial degree of supersaturation
- $U$: conservation mass term

* Corresponding author: Tel.: +81-952-28-8606; fax: +81-952-28-8687.
E-mail address: matsuo@me.saga-u.ac.jp
1. Introduction

The characteristics of transonic flow over an airfoil are determined by a shock wave standing on the suction surface. In this case, the shock wave/boundary layer interaction becomes complex because an adverse pressure gradient is imposed by the shock wave on the boundary layer. Several types of passive control techniques have been applied to shock wave/boundary layer interaction in the transonic flow. For instance, Bahi et al., 1983 and Raghunathan, 1988 described that a porous wall and cavity system, when it was applied at the foot of the shock wave, were known to be effective in alleviating undesirable adverse pressure gradient of the shock wave/boundary layer interaction. However, this control method essentially leads to large viscous losses caused by the porous walls, which can overcompensate the control benefit of the shock wave. Thus, the method can not be generalized as an effective method.

In order to overcome the demerits above several techniques were proposed. Firstly, the passive control using the porous wall with a cavity and vortex generator was applied to the shock wave/boundary layer interaction. It was shown by Saida et al., 2002 that this method was effective to the reduction of the wave drag and suppression of the boundary layer growth. Second, Raghunathan et al., 1999, O’Rourke et al., 2001 reported that the passive control using the porous wall with a cavity and vortex control jets upstream of porous wall might be effective to control of the shock position and pressure gradient. Ogawa et al., 2006 described that there are the optimum flow deflection for maximum reduction of total pressure losses in the flow with shock wave theoretically when the flow direction are changed by control device. Furthermore, possibilities for the control of flow fields due to non-equilibrium condensation have been shown so far (Wegener et al., 1958, Matsuo et al., 1985, Schnerr, 1986, Setoguchi et al., 1997, Matsuo et al., 1997). Furthermore, the development of boundary layer was reduced behind the shock wave. In these flow fields, non-equilibrium condensation occurs across the passage of the nozzle and it causes the total pressure loss in the flow field.

However, local occurrence of non-equilibrium condensation in the flow field may change the characteristics of total pressure loss compared with that by non-equilibrium condensation across the passage of flow field. Nagao, et al., 2012 described that locally occurred non-equilibrium condensation in supersonic nozzle reduces the total pressure loss in comparison with that occurred across the passage of the nozzle. However, there are few for researches of locally occurred non-equilibrium condensation in a transonic flow field.

The purpose of this study is to clarify the effect of locally occurred non-equilibrium condensation on the shock strength and total pressure loss on a transonic internal flow field with a circular bump.

2. Experimental Setup and Procedure

To validate simulated results, static pressures on the wall were measured and the flow was visualized by schlieren optical method. Figure 1(a) shows the experimental apparatus. The apparatus is consisted of reservoir tank, air drier, test section, valves and vacuum tank. Figure 1(b) shows the details of test section. Height of the nozzle is 60 mm. A radius of circular arc is 100 mm. The height of throat is 56 mm. Dry air ($S_e=0.18$) is used as a working gas. Total pressure and temperature of reservoir tank are 102 kPa and 296 K, respectively.
3. Computational Conditions

Figure 2 shows a computational domain of the transonic flow field and boundary condition. The nozzle has a height of \( H = 60 \) mm at the inlet and exit, and a radius of circular arc is \( R = 100 \) mm. The height of nozzle throat \( H^* \) is 56 mm. The region upstream of the nozzle was separated into dry air and moist air regions by a plate. Thickness of the plate is 0 mm.

Table 1 shows initial conditions used in the present calculation. Total pressure \( p_0 \) and temperature \( T_0 \) at stagnation point are 102 kPa and 298 K, respectively. The working gases of upper and lower sides of the plate are dry and moist airs, respectively. The plate position \( h/H \) is 0 (Case 1), 1.0 (Case 2), 0.125 (Case 3). The initial degree of supersaturation \( S_0 \) of moist air is 0.8. The number of grids is 450×119.

The adiabatic no-slip wall was used as boundary condition. The boundary condition of inlet and exit were fixed at initial condition and out flow condition, respectively. Condensate mass fraction \( g \) was set at \( g = 0 \) on the wall.

4. CFD Analysis

The governing equations, i.e., the unsteady 2D compressible Navier-Stokes equations that were combined with continuity, energy, nucleation rate, a droplet growth and diffusion equations (Bird et al., 1983, Hirschfelder, 1954) were written in

\[
\frac{\partial U}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} = \frac{1}{Re} \left( \frac{\partial R}{\partial x} + \frac{\partial S}{\partial y} \right) + H + Q
\]  (1)

where

\[
U = \begin{bmatrix}
\rho \\
\rho u \\
\rho v \\
\rho E \\
\rho k \\
\rho \omega \\
\rho \theta \\
\rho D_1 \\
\rho D_2 \\
\rho D_3 \\
\rho l
end{bmatrix},
E = \begin{bmatrix}
\rho u \\
\rho u^2 + \frac{\xi_x \tau_{xx} + \xi_y \tau_{xy}}{} \\
\rho v \xi_y + \frac{\xi_y \tau_{yy}}{} \\
\rho E + \rho k \\
\rho \omega \\
\rho \theta \\
\rho D_1 \\
\rho D_2 \\
\rho D_3 \\
\rho l
end{bmatrix},
F = \begin{bmatrix}
\rho v \xi_x + \xi_y \tau_{xy} \\
\xi_x \tau_{xx} + \xi_y \tau_{xy} \\
\xi_y \tau_{xx} + \xi_y \tau_{xy} \\
\eta_x \tau_{xx} + \eta_y \tau_{xy} \\
\eta_x \tau_{xx} + \eta_y \tau_{xy} \\
\eta_x \tau_{xx} + \eta_y \tau_{xy} \\
\eta_x \tau_{xx} + \eta_y \tau_{xy} \\
\eta_x \tau_{xx} + \eta_y \tau_{xy} \\
\eta_x \tau_{xx} + \eta_y \tau_{xy} \\
\eta_x \tau_{xx} + \eta_y \tau_{xy}
end{bmatrix},
H = \begin{bmatrix}
P_x - \beta \rho \omega \\
0 \\
0 \\
0 \\
0 \\
0 \\
0 \\
0 \\
0 \\
0 \\
0
end{bmatrix},
Q = \begin{bmatrix}
0 \\
0 \\
0 \\
0 \\
0 \\
0 \\
0 \\
0 \\
0 \\
0 \\
0
end{bmatrix}
\]  (2)

In Eq.(1), \( U \) is conservative vector, \( E \) and \( F \) are inviscid flux vector and \( R \) and \( S \) are viscous flux vectors. \( H \) and \( Q \) are the source terms corresponding to turbulence and condensation, respectively. \( \tau_{xx}, \tau_{xy}, \tau_{yx} \) and \( \tau_{yy} \) are components of viscous shear stress.

In Eq.(2),
\[ \tau_{e_1, \xi} = \xi_x \left( q_x + u \tau_{xx} + v \tau_{xy} \right) + \xi_y \left( q_y + u \tau_{yx} + v \tau_{yy} \right), \quad \tau_{e_2, \eta} = \eta_x \left( q_x + u \tau_{xx} + v \tau_{xy} \right) + \eta_y \left( q_y + u \tau_{yx} + v \tau_{yy} \right) \]  \hfill (3)

\[ \tau_{e_1, \xi} = \xi_x \left( \rho \Delta_1 \frac{\partial c_1}{\partial x} \right) + \xi_y \left( \rho \Delta_1 \frac{\partial c_1}{\partial y} \right), \quad \tau_{e_1, \eta} = \eta_x \left( \rho \Delta_1 \frac{\partial c_1}{\partial x} \right) + \eta_y \left( \rho \Delta_1 \frac{\partial c_1}{\partial y} \right) \]  \hfill (4)

\[ q_{sx} = \frac{1}{\gamma - 1} \left( \frac{\mu}{\Pr} + \frac{\mu_t}{\Pr_t} \right) \frac{\partial T}{\partial x} + \rho \left( h_1 \Delta_1 \frac{\partial c_1}{\partial x} + h_2 \Delta_2 \frac{\partial c_2}{\partial x} \right) \]  \hfill (5)

\[ f_k = \mu + \sigma^* \frac{\rho k}{\omega}, \quad f_\omega = \mu + \sigma \frac{\rho k}{\omega}, \quad f_{\text{H}} = \omega \frac{\partial \rho}{\partial k} - \beta \rho \omega^2 + \sigma_k \rho \left( \frac{\partial k}{\partial x} + \frac{\partial k}{\partial y} \right) + \frac{\partial k}{\partial y} \]  \hfill (6)

\[ P_k = 2 \mu_k \left( \frac{\partial u}{\partial x} \right)^2 + \frac{1}{2} \left( \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right) \left( \frac{\partial v}{\partial y} \right)^2 \], \quad \mu_k = \frac{\rho k}{\omega}, \quad \omega = \max \left\{ \frac{2 S_{1,j} \bar{S}_{1,j}}{\beta^*} \right\} \]  \hfill (7)

\[ S_{1,j} = S_{i,j} - \frac{1}{3} \frac{\partial u_k}{\partial x_k}, \quad S_{1,j} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \]  \hfill (8)

The density of gas mixture is calculated by the sum of density of vapor (\( \rho_1 \)) and dry air (\( \rho_2 \)):

\[ \rho = \rho_1 + \rho_2 \]  \hfill (9)

The mass fraction can be given as

\[ c_i = \frac{\rho_i}{\rho} \]  \hfill (10)

In equations (4) and (5) \( \Delta_1 \) and \( \Delta_2 \) are effective diffusivities. The closure coefficients are,

\[ \beta = 0.0708, \quad \alpha = \frac{13}{25}, \quad \beta^* = \frac{9}{100}, \quad \sigma = \frac{1}{2}, \quad \sigma^* = \frac{3}{5}, \quad \Pr_t = \frac{7}{8} \]  \hfill (11)

\[ \sigma_\eta = \begin{cases} 0, & \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \leq 0 \\
\sigma_\eta, \quad \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \geq 0 & \left\{ \sigma_\eta = \frac{1}{8} \right\} \end{cases} \]  \hfill (12)

The governing equation systems that are non-dimensionalized with reference values at the reservoir condition were mapped from the physical plane into a computational plane of general transform. To close the governing equations, \( k-\omega \) model (Wilcox, 2008) was employed in computations. A third-order TVD finite difference scheme with MUSCL (Yee, 1989) was used to discretize the spatial derivatives, and second-order central difference scheme for the viscous term, and a second-order fractional step method (Strang, 1976) was employed for time integration.
Fig. 1. Experimental apparatus and details of test section

Fig. 2. Computational domain and boundary conditions

Table 1. Computational conditions

<table>
<thead>
<tr>
<th>$h/H$</th>
<th>Initial degree of supersaturation</th>
<th>$p_0$ [kPa]</th>
<th>$T_0$ [K]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>0</td>
<td>$S_{0US}$</td>
<td>$S_{0LS}$</td>
</tr>
<tr>
<td>Case 2</td>
<td>1.0</td>
<td>0</td>
<td>0.8</td>
</tr>
<tr>
<td>Case 3</td>
<td>0.125</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Fig. 3. Comparison of static pressures distributions between the experimental and simulated flow fields ($S_0 = 0$)
5. Result and discussions

Figure 3 shows comparison of static pressure distributions on lower wall obtained by experiment and simulation in case of dry air. Values of initial degree of supersaturation for experiment and simulation are $S_0=0$ and $0.18$, respectively. As is evident from this figure, results of the number of grids of $450 \times 119$ and $470 \times 129$ are good agreement well with experimental result. Therefore, the number of grids of $450 \times 119$ was adopted in order to prevent the consumption of computation time in the present study.

Figure 4 shows comparison between experimental and simulated results in cases of dry air (Experiment : $S_0=0.18$ (Fig.4(a)), Simulation : $S_0=0$ (Fig.4(b))) and moist air ($S_0=0.5$ (Experiment : Fig.4(c), Simulation : 4(d)), respectively. Flow direction is left to right. As seen from these figures, shock wave is observed on the circular arc bump. In the case of dry air (Figures 3(a)
and (b)), shock wave is clearly visible compared with that of moist air (Figures 3(c) and (d)). Further, simulated results agree well with experimental results.

Figures 5(a) and (b) show time-averaged contour maps of static pressure \( p/p_0 \) (solid line) and condensate mass fraction \( g \) for Cases 2 and 3, respectively. Initial degree of supersaturation of moist air is \( S_0=0.8 \). Regions of moist air for Cases 2 (Fig. 5(a)) and 3 (Fig. 5(b)) are \( h/H=1.0 \) and 0.125, respectively. As seen from these figures, condensate droplets are observed and condensate mass fraction begins to increase along the bump wall side upstream of the shock wave and distributes over downstream region. For Case 3 (Fig. 5(b)), condensate droplets expand in a narrow region compared with that of Case 2 (Fig. 5(a)).

Figures 6(a), (b) and (c) show the time-averaged contour maps of static pressure \( p/p_0 \) (solid line) and Mach number \( M \) for Cases 1, 2 and 3, respectively. As seen from these figures, shock wave is shown in the flow field. For Cases 2 (Fig. 6(b)) and 3 (Fig. 6(c)), the height of adiabatic shock wave seems to be small compared with that for Case 1 (Fig. 6(a)). It is considered for the reason that Mach number just before shock wave reduces due to the condensation.

Figure 7 shows time-averaged distributions of condensate mass fraction \( g \) on line A for Cases 2 and 3, respectively. The abscissa is the distance \( x/H^* \) from throat, and the ordinate is condensate mass fraction \( g \). As seen from this figure,
condensate mass fraction begins to increase before the shock wave \((x/H^*=0.16)\). In the case of Case 3, condensate mass fraction increases rapidly and maximum value is large in comparison with that for Case 2.

Figure 8 shows time-averaged distributions of static pressure \(p/p_0\) on line A for all cases. The abscissa is the distance \(x/H^*\) from throat, and the ordinate is static pressure \(p/p_0\). As seen from this figure, static pressure begins to increase after the onset of condensation. In cases of Cases 2 and 3, the position of shock wave is located upstream compared with that of Case 4. This is due to reduction of Mach number before the shock wave by non-equilibrium condensation. Further, the position of shock wave for Case 2 is located upstream compared with that for Case 3.

Figure 9 shows time-averaged distributions of displacement thickness \(\delta'/H^*\) for all cases. The abscissa is the distance \(x/H^*\) from throat, and the ordinate is displacement thickness \(\delta'/H^*\). As is evident from this figure, displacement thickness behind the shock wave for Cases 2 and 3 are small compared with that for Case 1 (no condensation).

Table 2 shows ratio of \(p_2/p_1\) as indicated in Fig. 6 (shock strength \(\phi=p_2/p_1\)) and change in \(\phi\) based on shock strength \(\phi_{\text{Case1}}\) in case without condensation (Case 1) for all cases. As seen from this table, shock strengths for Cases 2 and 3 are reduced by 3.5 % and 2.7 %, respectively.

Table 3 shows integrated total pressure loss \(\beta\) from Line B to Line C as shown in Figs. 3 and 4 and change in \(\beta\) based on integrated total pressure loss \(\beta_{\text{ad}}\) in case without condensation (Case 1). Integrated total pressure loss is calculated from following equation.

\[
\beta = \int_{\text{Upper wall}}^{\text{Lower wall}} \left(1 - \frac{p_0}{p_0^1}\right) dy 
\]

In Eq. (15), \(p_0^1\) and \(p_0\) are local and stagnation total pressure, respectively. As is evident from this table, integrated total pressure losses for Cases 2 and 3 are small compared with that for Case 1 and integrated total pressure loss for Case 3 is the smallest. This reason is considered that the region of non-equilibrium condensation for Case 3 is small compared with that for Case 2.

6. Conclusions

A numerical study has been made to investigate the effect of locally occurred non-equilibrium condensation on a transonic flow field with circular arc bump. The results obtained are summarized as follow:

(1) Adiabatic shock wave moved upstream by locally occurred non-equilibrium condensation.

(2) Displacement of adiabatic shock wave position decreased with a decrease of occurrence region of condensate droplets.

(3) Displacement thickness with local occurrence of non-equilibrium condensation became thin downstream compared with that of no condensation.

(4) Shock strength in case with local occurrence of non-equilibrium condensation was small compared with that of no condensation.

(5) Total pressure loss of the flow field decreased with a decrease of occurrence region of condensate droplets.

References


