Direct numerical study of air layer drag reduction phenomenon over a backward-facing step

By D. Kim and P. Moin

1. Motivation and objective

Skin-friction drag reduction has significant practical value for marine vehicles. It is known that skin-friction drag accounts for more than 60% of the total drag of a cargo ship. For several decades, various methods have been explored to reduce the skin-friction drag. For example, the use of longitudinal riblets, and injection of polymers and air bubbles have received considerable attention. Implementations of the latter two have encountered practical difficulties, and drag reduction by riblets has not been at appreciable levels (< 7%). Recently, it has been demonstrated that drag reduction by means of injection of an air layer between the water and the solid wall might be an attractive alternative for skin friction drag reduction. The present study aims to provide high-fidelity computational support for the recent experiments that have demonstrated air layer drag reduction and for the study of the stability of the air/water interface in high Reynolds number turbulent flow.

Drag reduction by injecting air can be categorized into bubble drag reduction (BDR) and air layer drag reduction (ALDR), depending on the shape and distribution of air bubbles. Early reporting of BDR was by Mccormick (1973). They observed drag reductions approaching 40% by generating bubbles near the leading edge of a body. To date, a number of experiments have been conducted to investigate BDR and to determine the parameters that influence skin-friction drag (Madavan & Deutsch 1985; Merkle 1992; Sanders & Winkel 2006). It has been observed from experiments that drag reduction increases with gas flow injection rate, but decreases rapidly with downstream distance as the bubbles move away from the surface. Recently, numerical simulations have been performed to predict the observed drag reduction (Felton & Loth 2002; Ferrante 2004).

In ALDR it has been reported that under certain conditions bubbles collapse into the air layer, which is formed between liquid and solid surface (Elbing et al. 2008). In the experiment of Elbing et al. (2008), a 12.9 m-long and 3.05 m-wide flat plate was used with water stream from 6.7 m/s to 13.3 m/s. A stable gas layer was formed over the entire test surface when the gas injection rate was higher than a critical value. At the critical gas injection rate, BDR undergoes transition into ALDR. The critical gas injection rate is dependent on the water free-stream velocity. The critical volumetric gas injection rate per unit span was 0.026 $m^2/s$ and the corresponding nominal thickness of the gas layer was about 4.2 mm for the water speed of 6.7 m/s. It was pointed out that the high gas injection rate and surface tension force stabilize the gas layer. The stable air layer reduces the drag more than 80%. In contrast to BDR, no decay in drag reduction was found with downstream distance. Because the expended power for gas injection diminishes the net energy savings, it is of interest to find optimal conditions under which maximum energy savings are produced.

In the present study, we perform direct numerical simulations in order to confirm the experimental results and examine the stability and mechanism of ALDR for different air
injection rates. We also investigate the stability of air layer theoretically by solving the Orr-Sommerfeld equations in both phases in order to find the stabilizing parameters and stability conditions for ALDR.

2. Governing Equations

The governing equations for incompressible, immiscible, two-phase flow are as follows:

\[ \nabla \cdot \mathbf{u} = 0, \]  
(2.1)

\[ \frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u} \mathbf{u}) = -\frac{\nabla p}{\rho} + \frac{1}{\rho} \nabla \cdot (\mu (\nabla \mathbf{u} + \nabla^T \mathbf{u})) + g + \frac{T}{\rho}, \]  
(2.2)

where \( \mathbf{u}, p, \mu, \) and \( g \) are the velocity, density, pressure, viscosity, and gravitational acceleration, respectively. \( T \) is the surface tension force, which acts only on the phase interface \( (x_f) \) and vanishes at other locations. It is described as follows:

\[ T = \sigma \kappa \delta(x - x_f)(n), \]  
(2.3)

where \( \sigma, \kappa, \delta, \) and \( \mathbf{n} \) are the surface tension coefficient, surface curvature, delta function, and surface normal vector, respectively. The temporal evolution of the phase interface can be predicted by solving the level-set equation:

\[ \frac{\partial G}{\partial t} + \mathbf{u} \cdot \nabla G = 0, \]  
(2.4)

where the iso-surface of \( G = 0 \) defines the location of the interface whereas \( G > 0 \) and \( G < 0 \) represent two different phase regions, respectively. In the computational domain, \( G \) is set to be a signed distance function to the interface such that

\[ |\nabla G| = 1. \]  
(2.5)

The interface normal vector \( \mathbf{n} \) and the interface curvature \( \kappa \) can be calculated from \( G \) as

\[ \mathbf{n} = -\frac{\nabla G}{|\nabla G|}, \]  
(2.6)

\[ \kappa = \nabla \cdot \mathbf{n}. \]  
(2.7)

Assuming \( \rho \) and \( \mu \) are constant within each fluid, the density and viscosity are defined as a function of \( G \):

\[ \rho = \psi \rho_1 + (1 - \psi)\rho_2, \]  
(2.8)

\[ \mu = \psi \mu_1 + (1 - \psi)\mu_2, \]  
(2.9)

\[ \psi = H(G), \]  
(2.10)

where the subscripts 1 and 2 denote properties in fluid 1 and 2, respectively, and \( H \) is
the Heaviside function. Here, the numerical Heaviside function developed by Pijl et al. (2005) is used to compute the volume fraction.

In the present study, the numerical algorithm based on the Refined Level Set Grid (RLSG) method (Herrmann 2008) is used to solve the level set equation (Eq. 2.4). The level-set equation is solved on a separate refined G-grid using a fifth-order WENO scheme (Jiang & Peng 2000) in conjunction with a local Lax-Friedrichs entropy correction (Shu & Osher 1989). Time integration is performed using a third-order TVD Runge-Kutta method. Reinitialization of the level-set is performed by solving partial differential equations using a fifth-order WENO scheme and a first-order pseudo-time integration (Sussman et al. 1994; Peng et al. 1999).

The fractional-step method is used to solve the unsteady three-dimensional Navier-Stokes and continuity equations for variable density and viscosity. Convection and diffusion terms which involve only derivatives in the wall-normal direction are treated implicitly, whereas all other terms are treated explicitly. Velocity components are staggered with respect to pressure and density. A second-order central difference scheme is used for spatial derivatives. A third order Runge-Kutta scheme is used for the explicit terms and a second order Crank-Nicolson scheme is used for implicit terms (Pierce & Moin 2001, 2004). The pressure gradient and surface tension forces are predicted by a balanced force algorithm, which guarantees a discrete balance between them (Francois et al. 2006; Herrmann 2008). The numerical details are explained in Kim & Moin (2009).

3. Numerical Results

3.1. Computational details

Because the Reynolds number in the experiment of Elbing et al. (2008) is too high for direct numerical simulation, DNS has been conducted with the same flow parameters and geometry as in a low Reynolds number experiment, which will be performed at the University of Michigan in the near future (Fig. 1). Turbulent water flows on the flat plate with a backward-facing step, and air is injected through the slot at the step. The slot height is 0.375$h$ in the vertical direction and 3$h$ in the spanwise direction, where $h$ is the step height and set to 12.7mm.

In the present study, the DNS simulation of water flow over a backward-facing step without air injection is performed first for the base flow and then the two cases at different air injection rates are computed to study the effect of air injection rate on the
stability of air layer. The inlet section length before the step is $3h$ and the post expansion length is $30h$. Total 2640 computational cells are selected for the streamwise direction ($x$-direction). Based on the inlet wall shear velocity, the grid spacing is $\Delta x^+ \simeq 12$ in the wall unit. In the vertical direction, the expansion ratio of 1.2 is chosen and the domain height is $6h$. A non-uniform grid spacing is used with fine grid spacing near the lower wall and the location of the step. The spacing at these two locations is $\Delta y^+ \simeq 0.1$ based on the inlet wall shear velocity. The total number of computational cells in the vertical direction is 265. In the spanwise direction, the domain length is $3h$ and the grid spacing is $\Delta z^+ \simeq 7$ with 380 cells. The total number of grid points is about 271 million. The free stream velocity of water outside the boundary layer is set to $u_\infty = 1.8\text{m/s}$ as in the experiment. The Reynolds and Weber numbers, based on the water properties and step height, are $Re_h = 22,800$ and $We_h = 560$, respectively. The turbulent inlet velocity profile was obtained from the inflow-generation method by Lund et al. (2003). For this particular profile, the boundary layer thickness is $\delta = 0.49h$ and the Reynolds number based on the momentum thickness is $Re_\theta \simeq 1100$ at the step.

3.2. Turbulent water flow without air injection

In order to verify our numerical method and provide the base flow for ALDR, turbulent water flow over a backward-facing step without air injection is simulated. Table 1 shows the mean reattachment length of the present computation with other experimental results. The mean reattachment length is determined by the location of zero wall-shear stress. As shown in table 1, the present computational result is consistent with the experimental data. The computed mean skin friction coefficient normalized by the inlet velocity is plotted with the experimental data for comparison in Fig. 2. Good agreement is also achieved between computational and experimental results. The mean streamwise velocity profiles at four representative points behind the step are shown in Fig. 3. The backward-facing step geometry produces a strong shear layer near the step, and a large recirculation region is formed underneath of the shear layer as shown in Fig. 3. The mean reattachment length is measured as $X_r = 6.45$ in the simulation.
Figure 3. Mean streamwise velocity profiles at four different positions in the streamwise direction. ——, mean velocity profile with air injection, \( q_{air} = 0.0114 m^2/s \); – – –, mean velocity profile without air injection.

Table 1. Reattachment lengths (\( X_r \)) of the present study and experimental data (Jovic & Driver 1995).

<table>
<thead>
<tr>
<th>Case</th>
<th>( Re_h )</th>
<th>Expansion ratio</th>
<th>( X_r/h )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Jovic &amp; Driver</td>
<td>6,800</td>
<td>1.2</td>
<td>5.35</td>
</tr>
<tr>
<td>Jovic &amp; Driver</td>
<td>10,400</td>
<td>1.09</td>
<td>6.35</td>
</tr>
<tr>
<td>Present study</td>
<td>22,860</td>
<td>1.2</td>
<td>6.45</td>
</tr>
<tr>
<td>Jovic &amp; Driver</td>
<td>25,500</td>
<td>1.2</td>
<td>6.70</td>
</tr>
<tr>
<td>Jovic &amp; Driver</td>
<td>37,200</td>
<td>1.2</td>
<td>6.84</td>
</tr>
</tbody>
</table>

3.3. High air injection flow rate

As shown in the experiments (Elbing et al. 2008), the stability of the air layer is found to be enhanced as the air injection rate is increased. The air injection rate is one of the most important parameters in ALDR because it is easily controlled in a practical sense. In order to confirm ALDR phenomenon, the air injection rate is set to \( q_{air} = 0.0114 m^2/s \), which is above the critical air injection rate predicted from stability analysis (see section 4). The mean air velocity inside the slot is 2.39 m/s, which is higher than the free-stream velocity of water. However, the flow inside the slot is laminar and the Reynolds number based on the slot height and air properties is \( Re_{air} = 758 \). Owing to the air injection, the mean velocity profiles are changed as shown in Fig. 3. The velocity profile near the injector has sharp peak (\( x/X_r = 0.075 \)) and then it is converged to the stable profile in the downstream region (\( x/X_r = 3.0 \)), which looks like a two-phase Couette flow. Figure 4 shows the snapshot of the phase interface. Near the injector where the recirculation occurs, the water flow contacts the air layer with high velocity difference leading to the Kelvin-Helmholtz instability, as in the mixing layer. In the downstream region, however, the growth of the unstable wave is suppressed and further nonlinear development or breakup does not occur. As the flow reaches the far downstream region (\( x/X_r > 3.0 \)), the velocity profile turns into a two-phase couette flow like profile. In this region, the interface is stable and the amplitude of disturbances decreases. Numerical result is consistent with the linear stability analysis, which shows the negative growth-rate of the disturbances...
Figure 4. Snapshot of stable air-water interface for $q_{\text{air}} = 0.0114m^2/s$.

Figure 5. Mean skin-friction coefficients with air injection at $q_{\text{air}} = 0.0114m^2/s$.

3.4. Low air injection flow rate

When the air injection rate is decreased to $q_{\text{air}} = 0.00357m^2/s$, the air layer becomes unstable and undergoes breakup as shown in Fig. 6. The Kelvin-Helmholtz type instability occurs near the injector and makes water penetrate into the air layer (Fig. 7a). The water penetration makes the water spot in the wall where the solid wall contacts with water flow (Fig. 7b). This water spot becomes larger and larger as it moves downstream region with this profile (section 4). Because the stable air layer is located between water and solid wall, the skin-friction is dramatically reduced. Figure 5 shows the mean skin-friction coefficient along the streamwise direction. Compared to the case of water flow without air injection (Fig. 2), the total drag reduction rate is more than 99%. Thus, it is confirmed that ALDR phenomenon occurs at this air injection rate.
Figure 6. Snapshot of unstable air-water interface for $q_{air} = 0.00357m^2/s$.

(Fig. 7c) and finally breaks down producing air sheets and bubbles. The generation and breakup of the water spots are clearly seen in Fig. 8, which shows the bottom view of the layer.

Because the air layer loses its stability and no longer covers the entire solid wall, the skin-friction coefficient increases in contrast to that in the case of the high air-flow injection. Figure 9 shows the time averaged skin-friction coefficients of three cases for comparison. After losing the stability of the air-water interface, the skin-friction is dramatically increased due to the water spots. The total drag reduction rate on the entire flat plate is 99% in the case of high air injection where the stability of air layer is maintained on the entire plate. In the case of low air-flow injection, however, the drag reduction rate is decreased to 49%. This case belongs to the transitional regime between ALDR and BDR as reported in Elbing et al. (2008). The critical air flow rate, which is the minimum air flow rate to enable the ALDR, can be calculated from the stability theory by solving the Orr-Sommerfeld equation (section 4). The predicted critical air flow rate is $q_{theory} = 0.0052m^2/s$ which is between 0.00357 and 0.0114$m^2/s$ used in the aforementioned simulations. The result shows that the stability of the air layer predicted by numerical simulations is consistent with the stability theory.

4. Stability Analysis

The linear stability of air layer has been investigated by solving the Orr-Sommerfeld equations on a Couette-Poiseuille flow configuration modeling the far downstream region from the air injector (Fig. 3). Figure 10 shows a Couette-Poiseuille flow configuration consisting of two immiscible, incompressible gas and liquid between two infinite-long horizontal plates. The density, molecular viscosity, and layer thickness for the two fluids are denoted as $\rho_g$, $\mu_g$, and $\delta_g$ where subscripts g and l indicate gas and liquid, respectively. The top plate is at rest and the bottom plate is moving with a fixed velocity ($u_\infty$, 0), which corresponds to the water free-stream velocity. The density ratio, viscosity ratio, Reynolds number, Weber number, and Froude number are defined as $r = \rho_g/\rho_l$, $m = \mu_g/\mu_l$, $Re = \rho_l u_\infty \delta_g/\mu_l$, $We = \rho_l u_\infty^2 \delta_g/\tau$, $Fr = (\rho_l u_\infty^2/g(\rho_l - \rho_g)\delta_g)^{1/2}$, where $\tau$ is
the surface tension force and \( g \) is the gravity force. The reference length and time scales are \( \delta_g \) and \( \delta_g/u_\infty \), respectively. The base flow satisfying the Navier-Stokes equations is:

\[
U_g(\hat{y})/u_\infty = -G\hat{y}^2 + a\hat{y} + b \\
U_l(\hat{y})/u_\infty = ma\hat{y} + b, \tag{4.1}
\]

where \( a = (-1 + G)/(1 + mn) \), \( b = 1 + mna \), \( n = \delta_g/\delta_l \), \( m = \mu_g/\mu_l \) and \( \hat{y} = y/\delta_g \). \( U_g \) and \( U_l \) are the base velocity profiles in the gas and liquid layers, respectively. A non-dimensional pressure gradient \( G \) is imposed along the \( x \)-direction only in the gas layer. The air injection rate is defined as
Two-dimensional disturbances of the form \( \exp(i\alpha(\mathbf{x} - ct)) \) to the velocity, pressure, and interface position are imposed, where \( \alpha \) is the non-dimensional wave number of the disturbance and \( c \) is the complex wave speed. The non-dimensional growth rate, \( \sigma \), is the real part of \(-i\alpha c\). The sign of the growth rate determines the stability of the system. Substitution of disturbances into the linearized Navier-Stokes equations results in Orr-Sommerfeld equations for each layer as described in Joseph & Renardy (1993). A Chebyshev collocation method is employed to solve the Orr-Sommerfeld equation. All the parameters are chosen based on the simulation above.

Figure 11 shows growth rates of disturbances as a function of the wave number at three different liquid free-stream velocities with the fixed air layer thickness. As seen in Fig. 11, the overall instability of the phase interface is enhanced for all wave numbers as the liquid free-stream velocity increases. Surface tension force is one of the important parameters affecting the instability of phase interface. Growth rates of disturbances computed at three different Weber numbers are shown in Fig. 12. Increasing the surface tension force is found to enhance the interface stability especially at high wave numbers. This is expected because the curvature of a high wave number disturbance is larger than that of a low.
Figure 11. Growth rates as a function of wave number for different free stream velocities at $\delta_g=5\text{mm}$, $\delta_l+\delta_g=19\text{mm}$, $G=0$, $We=260$ and $Fr=65$. ––, $u_\infty=1.8\text{m/s}$; –––, $u_\infty=0.9\text{m/s}$; ····, $u_\infty=4.0\text{m/s}$.

Figure 12. Growth rates as a function of wave number for different Weber numbers at $\delta_g=5\text{mm}$, $\delta_l+\delta_g=19\text{mm}$, $G=0$, $u_\infty=1.8\text{m/s}$ and $Fr=65$. ––, $We=260$; –––, $We=26$; ····, $We=2600$.

As shown in Fig. 13, a higher value of the inverse Froude number (the ratio of gravity force to inertia) is more beneficial to the air-layer stability. This is because the air layer is located above the heavier water layer in the present flow configuration. The air injection rate $q_{air}$, which is defined in Eq. 4.3, is a practical parameter to control the air layer stability as shown in Elbing et al. (2008). Figure 14 shows the effect of air injection flow rate on the growth rate. All other parameters are matched with those of the numerical simulation (section 3). The growth rate of an interfacial disturbance is found to decrease as the air flow rate $q$ is increased, which is consistent with the numerical and experimental observations (Elbing et al. 2008). The
5. Conclusions and future work

In the present study, the stability of the air-layer is investigated by theoretical and numerical approaches. DNS of ALDR with two different air injection rates have been performed to understand the role of air injection rate on the stability of air layer. From theoretical critical air injection rate is predicted as $q_{\text{critic}} = 0.0052m^2/s$, which is located between $q_{\text{air}} = 0.00357m^2/s$ and $0.011m^2/s$ used in the numerical simulations, showing good consistency.
the numerical simulations, it is observed that the air layer with higher air injection rate was stabilized on the entire domain whereas lower air injection rate induces the instability of air layer, which is consistent with the experimental observation (Elbing et al. 2008). The breakup of the air layer makes the water spots on the solid wall which results in much larger skin-friction. Validation of DNS results against experimental measurements will be done after the experiment is completed.

Linear viscous stability analysis was also performed in a two-dimensional two-phase Couette-Poiseuille flow configuration that mimics the far downstream region from an air injector. Effects of flow parameters such as surface tension, Reynolds number, the boundary layer thickness and air flow rates on the stability of the air-liquid interface are identified by solving the Orr-Sommerfeld equation for two-phase flow. Although the linear instability theory cannot predict the nonlinear mechanisms, we found that the critical air flow rates predicted from the linear stability analysis agree favorably with the numerical simulations, indicating that stability analysis can be used as a practical guide for ALDR.

Acknowledgments

The work presented in this paper was supported by the Office of Naval Research.

REFERENCES


DNS of air layer drag reduction over a backward-facing step


