Simulating plasma actuators in a channel flow configuration by utilizing the modified Suzen-Huang model

I.H. Ibrahim¹, M. Skote^{1,a}

¹School of Mechanical & Aerospace Engineering, Nanyang Technological University, 50 Nanyang Avenue, Singapore 639798. ^aCorresponding author, tel: +65 6790 4271, e-mail: mskote@ntu.edu.sg

Abstract

The present investigation is an attempt to simulate a channel flow driven by two plasma actuators placed on top of each other. The model utilizes a modified form of the Suzen-Huang plasma actuator which accounts for a 'dielectric shielding' boundary condition for the potential governing the electric field. In addition, the Fokker-Planck (drift-diffusion) characteristics were implemented on the potential governing the surface charge density. The model is able to correctly predict the maximum velocities for channel flow at larger channel heights. However, at lower channel heights, the model underestimates the maximum velocities. An analysis of the body force profile at the centreline region in the vicinity of the plasma actuators indicated that negative vertical body forces may have contributed to the discrepancies. Following this observation, a hypothetical model which does not account for vertical body force contributions on the fluid domain was simulated. While the results from this hypothetical model show marginally improvements to the maximum induced velocities at larger channel heights in relation to experimental data, the model still underpredicts the velocity magnitude at lower channel heights. This could point to the presence of interactions between the induced body force of the top and bottom actuators, specifically at lower channel heights, that have not been captured in the present model.

Keywords: channel flow; plasma actuator; numerical simulation

1. Introduction

The development of the Single Dielectric Barrier Discharge (SDBD) plasma actuator is motivated by applications in flow control. The ability to control the dynamics of fluid flow over a body allows engineers to incorporate designs that will drastically save fuel and energy. Recent research efforts with regards to flow control are being directed towards reducing drag, enhancing lift and augmenting the mixing of mass, momentum or energy. In order to achieve any of these, three issues have to be resolved; 1) transition from laminar to turbulent flow has to be delayed or advanced, 2) flow separation prevented or controlled and 3) turbulence levels have to be suppressed or enhanced.

Many methods have already been formulated to tackle the above mentioned aerodynamics issues but the catch is in discovering a control device or mechanism that is inexpensive to create as well as to operate and has greater savings than penalties involved. These methods are most effective when applied near the transition or separation points, in other words near the critical flow regimes where the instabilities magnify quickly. A review study of flow control by Gad-el-Hak [1] shows that energy wastage due to drag resistance results in losses amounting to billions of dollars in the aerospace industry.

A recent flow control review by Braun *et al.*[2] categorises three main forms of flow control actuators: Fluidic, Moving object/surface and Plasma. Fluidic actuators result in suction or an ejection of fluid near separation points. This is the main working mechanism of a synthetic jet, where a zero net mass flux is added to the aerodynamic system. The second group – Moving object or surface – alters wall effects of the flow, similarly reducing boundary layer separation and/or turbulence levels. Plasma actuators have evolved in the last two decades and reviews can be found in Corke *et al.*[3] and Moreau[4].



Figure 1: Features of a plasma induced SDBD.

The advantages of using plasma actuators for controlling airflow are that the actuators have no moving parts and are light. In addition, activating a plasma actuator results in realtime changes to the aerodynamic system. In the review by Cattefesta and Sheplak [5], flow control utilizing electric fields is described as an exciting topic due to two reasons; its multidisciplinary nature, and more importantly, potentially a long-term research into new sources of atomic energy that will eventually produce extremely high power.

Plasma actuators have been tested in various applications, including separation control on a delta wing [6] and variable-direction discharge [7]. In particular, the Single Dielectric Barrier Discharge (SDBD) has been researched widely by the group from Notre Dame [8-11]. The SDBD consists of one electrode exposed to the surrounding air and one electrode completely encapsulated by a dielectric material. This results in an asymmetric geometry as shown in Figure 1. An alternating current (AC) voltage is supplied to power the actuator. The asymmetric electrode design results in a body force that induces the flow in the direction from the exposed electrode toward the covered electrode.

The multitude of parameters involved in the design of the plasma actuator points to the use of CFD as a tool in the design process. As experimental methods pose a disadvantage because of high costs when designing actual geometries and data, computational models have evolved from the 1990s till today. Generally, all models require a coupling of the 'plasma' equations with the 'flow' equations. The Lorentz force is used to couple the resultant force produced by the charges to the source terms in the Navier-Stokes equations. Two forms of plasma actuator models exist: the microscopic [12-14] and macroscopic model [15-17]. The microscopic model captures the motion of the charge particles by coupling the drift-diffusion equation with the Maxwell equation. The macroscopic form captures the essential physics of the flow by modelling the electric potential characteristics as well as defining the charge density that forms the Lorentz body force.

The goal of this investigation is to study the effects of two actuators placed on top of each other in a channel configuration by utilizing a modified form of the Suzen and Huang (S-H) model [16]. This modified S-H model has been utilized in the linear actuator [18] and the Linear Plasma Synthetic Jet Actuator (L-PSJA) model [19]. The present study represents a continuation of the previous studies by examining the feasibility of implementing the modified S-H model in channel flow configuration. The original and modified models are described in section 2.1 and 2.2 respectively. Simulations are conducted in a multiphysics

software, COMSOL Multiphysics [20] and details of the software will be mentioned in section 3. Results will be presented in section 4. We compare of downstream velocity profiles with experimental data by Debiasi *et al.* [21] in section 4.1. The resultant body forces and induced velocities near the vicinities of the plasma actuators are presented in section 4.2. In addition, the downstream velocity profiles at various stations are shown in section 4.3. Based on the analysis of the results in the previous sections, we introduce a hypothetical body force to account for the difference between simulation and experimental results in section 4.4, before ending with the discussions and conclusions in section 5.

2. Model description and computational methodology

The geometries of our investigations consist of actuator configurations that are placed on top of each other with varying channel heights, as shown in Figure 2. The dimensions of our geometries are similar to that investigated experimentally by Debiasi *et al.* [21]. The lengths of the exposed and encapsulated electrodes are 8 mm and 15 mm respectively. The channel heights are varied at 0.004 m, 0.006 m, 0.008 m, 0.01 m, 0.02 m and 0.03 m. As used in previous S-H models [16, 22-25], the electrode heights and dielectric thickness in our simulations are in the order of 0.1 mm.



Figure 2: Features of the SDBD in a channel flow configuration. The configuration is not drawn to scale.

2.1. The S-H model

The plasma actuator model designed by Suzen and Huang [16] is based on the splitting the total electric potential term (Φ) into two parts: one being influenced by the external electric field (ϕ), and the other potential affected by the surface charge density (ϕ). This technique had been applied to turbine blades [16, 22-24] to achieve reduction in flow separation. The equations governing the plasma actuator are:

$$\nabla . (\varepsilon_r \nabla \phi) = 0$$
(1)
$$\nabla . (\varepsilon_r \nabla \phi) = \rho_c / \varepsilon_0$$
(2)

where ε_0 is the vacuum permittivity, ε_r is the relative permittivity, and ρ_c is the surface charge density.

The surface charge density potential (φ) can be eliminated by utilizing the Debye length (λ_D) which relates φ to ρ_c through:

$$\rho_c/\varepsilon_0 = (-1/\lambda_D^2)\varphi \tag{3}$$

which combined with (2) yields:

$$\nabla \cdot (\varepsilon_r \nabla \rho_c) = (\rho_c / \lambda_D^2) \tag{4}$$

Finally, the Lorentz body force which is used in the Navier-Stokes equations is obtained through:

$$F_b = \rho_c(-\nabla\phi) \tag{5}$$

The two equations defining the electric field potential (ϕ) and the surface charge density (ρ_c) can be solved initially before the Navier-Stokes equation as these equations do not contain a time derivative term. This is done by converting the terms into a non-

dimensionalised form. The variables ϕ and ρ_c are non-dimensionalised by their values at the upper and lower electrodes respectively. The term ϕ_o is set as a boundary condition and refers to the applied AC voltage at upper electrode:

$$\phi_0(\tau) = \phi_{max} \times f(\tau) \tag{6}$$

where ϕ_{max} (V) refers to the maximum amplitude of the AC voltage supplied.

The term $\rho_{c,o}$ is set as a boundary condition at the lower electrode as:

$$\rho_{c,o}(\tau) = \rho_{c,max} \times f(\tau) \tag{7}$$

where $\rho_{c,max}$ (C/m³) refers to the maximum surface charge density of the AC voltage supplied.

The function $f(\tau)$ for the AC voltage source appearing in eqns (6) and (7) is:

$$f(\tau) = \sin(\frac{\pi}{2}\tau) \tag{8}$$

where τ refers to a non-dimensionalised time quantity which is related to the frequency ω as:

$$\tau = \omega t \tag{9}$$

where ω (Hz) refers to the frequency of the AC voltage supply and is equal to 4.5 kHz in experiments [25]. We also relate the frequency to the characteristic time, t_c as:

$$\omega = \frac{1}{t_c} \tag{10}$$

where $t_c = 2.22 \times 10^{-4}$ s. Note that this characteristic time is the time taken to complete one sinusoidal cycle and has the same order of 10^{-4} s as other studies mentioned in the review by Corke *et al.*[3]. The simulations are run in the non-dimensionalised time quantity, τ , from a value of 0 to 1. A time step of 0.1 is chosen to adequately observe the evolution of the four variables of interests: electric field, surface charge density, body force and induced velocity.

The resulting non-dimensionalised quantities are:

$$\phi^* = \phi/\phi_0(\tau) \tag{11}$$

$$\rho_c = \rho_c / \rho_{c,0}(\tau) \tag{12}$$

$$F_b^* = F_b / (\rho_{c,max} \times \phi_{max})$$
(13)

The non-dimensionalized equations of (1) and (4) are:

$$\nabla . \left(\varepsilon_r \nabla \phi^* \right) = 0 \tag{14}$$
$$\nabla . \left(\varepsilon_r \nabla \rho_c^* \right) = \rho_c^* / \lambda_D^2$$

(15)

The boundary condition for the upper electrode for ϕ^* is set to unity so that once ϕ^* is determined, the dimensional value ϕ can be obtained at any given time by multiplying the distribution with the corresponding value $\phi(\tau)$ given by (6). Similarly, the boundary condition for the lower electrode for ρ_c^* is set to unity. This allows the dimensional value ρ_c to be obtained by multiplying the non-dimensionalised distribution ρ_c^* with the corresponding value $\rho_{c,o}(t)$ given in (7).

The maximum amplitude of the AC voltage supplied, ϕ_{max} , is set as 12 kV. Equations (14) and (15) are the governing equations used to model the non-dimensionalised body force in COMSOL and are solved separately from the Navier Stokes equations. The dimensionalised body force can be obtained from (13) and is inserted into the Navier-Stokes computations.

Finally, equations (14) and (15), and their boundary conditions are shown together with the geometries in Figure 3 and Figure 4 respectively. Note only the bottom actuator is shown here, as the boundary conditions are mirror imaged to the top electrode. The terms GE and BC refer to governing equation and boundary conditions respectively. Next we introduce the modification to the boundary condition.



Figure 3: Governing equation (14) for the S-H model. The illustration is not drawn in scale with simulation.



Figure 4: Governing equation (15) for the S-H model. The illustration is not drawn in scale with simulation.

2.2. Boundary condition modifications

In the present study we utilize the S-H model with the Fokker-Planck boundary condition as introduced by Ibrahim and Skote [18]. The S-H model was chosen against other physics-based models due to its ability to simulate the electric field and surface charge density variables separately. These two variables constitute the Lorentz body force which is inserted into the Navier-Stokes equation. The 'dielectric shielding' and Fokker-Planck boundary conditions were included to introduce drift and diffusion characteristics at the vicinity of the encapsulated electrode. These characteristics are based from the results seen in microscopic models. These modifications improved the linear actuator model results by about 50% when considering the maximum induced velocity value [18]. In the Linear- Plasma Synthetic Jet Actuator (L-PSJA) study, centreline velocity results were also closer to experimental values, when compared to the default S-H model [19].

To implement the modifications, two new BCs are introduced for the Kapton surface above the encapsulated electrode. BC3 for equation (14) (shown in Figure 3) and BC5 for equation (15) (shown in Figure 4) are replaced and described below.

2.2.1. Dielectric shielding

The 'dielectric shielding' condition for the non-dimensionalised electric potential shown as BC3N in Figure 5 describes a thin layer of thickness λ_D and relative bulk permittivity $\varepsilon_{r,m}$. The terms ∇_n and ∇_t describe the normal and tangential derivatives of the non-dimensionalised electric potential variable. The condition equates the normal and tangential derivatives of the dimensionless electric potential, to produce a thin layer across the boundary that shields the electric field formed by the two electrodes. This results in a spread of the electric potential and electric field magnitude across the boundary.



Figure 5: Governing equations for the ϕ^* equation in S-H model. The modified boundary condition is placed at the lower boundary as shown.

2.2.2. Fokker-Planck characteristics

For the equation governing the non-dimensionalised surface charge density, the original boundary condition (BC5 in Figure 4) used in the S-H model resulted in an instantaneous surface charge density growth along the boundary of the lower electrode that propagates in the normal direction (upwards) of the dielectric surface. This propagation direction is different compared to the results obtained by charge transport models, where propagation towards the right-side of the exposed electrode was obtained [13, 26]. This motion of the surface charge corresponds to the physics of the plasma actuator, in which streamers originate from the exposed electrode and travels along the dielectric surface [13]. These streamers dissipate and propagate from the exposed electrode and have not been shown in previous S-H models.



Figure 6: A graphical representation of the one-dimensional Fokker-Planck equation, with increasing time (t_1) to (t_5) .

The dissipation and propagation characteristics can be modelled by the solution to the one-dimensional Fokker-Planck equation, which is written as:

$$\frac{\partial f}{\partial t} = -D_1 \frac{\partial f}{\partial x} + D_2 \frac{\partial^2 f}{\partial x^2}$$
(16)

here D_1 and D_2 corresponds to drift and diffusion constants.

Figure 6 shows a one-dimensional representation of a normal distribution function exhibiting characteristics of equation (16). The value of the constants D_1 and D_2 are set to unity. The Dirichlet boundary conditions are inserted at both ends of the one-dimensional analysis. The figure shows that the function is initially (at t_1) at a maximum, dissipating and propagating in time. The dissipation is manifested by the decreasing peaks of each successive time step. Similarly, propagation can be observed by the increasing x-coordinates of the dashed lines, which indicate the horizontal locations of the peaks.



Figure 7: Governing equations for the ρ_c^* equation in S-H model. The modified boundary condition is placed at the lower boundary as shown.

We imposed the boundary modifications as mentioned in Ibrahim and Skote [18] at the interface above the encapsulated electrode as shown in Figure 7. To replicate Fokker-Planck characteristics in our new boundary condition, two boundary conditions are inserted. They form a new boundary condition which is obtained by multiplying BC8 and BC9. These boundary conditions are written as:

BC8:
$$\exp\left[-\left(\frac{x-x_1}{x_2}\right)^2/2\sigma^2\right]$$
 (17)
BC9: $\exp(-(x-x_1)-\tau) \quad x > x_1$ (18)

where x_1 is the x-coordinates of the left leading edges of the encapsulated electrode, which is at 0 m. The terms x_2 and σ are determined by the width of the normal distribution function and are taken as 2 x 10⁻³ m and 0.3 respectively as used previously by Ibrahim and Skote [18]. In addition, since the surface charge density only exists at the dielectric surface above the encapsulated electrode, BC7 is changed to BC7N as shown in Figure 7. The original BC7 would result in unphysical surface charge density growth in the dielectric domain, above the encapsulated electrode.

We used a Debye length of 10^{-5} m and maximum surface charge density of 70×10^{-4} C/m³ in our simulations. These parameters have been shown in previous studies [18, 27] as tuning parameters and were chosen based on preliminary findings to produce the best match with experimental results. The peak voltage based on the exposed electrodes was also set at 12 kV to mimic experimental conditions. The simulation was solved for the non-dimensionalised time quantity τ , from 0 to 1, similar to the previous studies [18, 19, 27].

3. Computational method and Grid independence

Since the problem was electrostatic and fluid dynamic in nature, we have used a software capable of handling the multiphysics features, COMSOL 4.2 [20]. The system used for the COMSOL simulations was a Dell workstation model E5520 with dual quad-core processors running at 2.26GHz. The workstation had 24GB of installed memory (RAM) and used Windows 7 Professional as its operating system.

The simulations were solved using the finite element computational package, COMSOL MultiphysicsTM 4.2a [20]. Since the study of plasma actuators is multiphysics in nature, the electrostatics application in COMSOL was used to model the potentials in the S-H model. The fluid domain is governed by the incompressible Navier-Stokes application, which is the ruling application mode for the investigation. The solver settings were based on the ruling application model. The application mode also used Lagrange p2-p1 elements to stabilize the pressure. Thus 2nd-order Lagrange elements modelled the velocity components while linear elements modelled the pressure.

The step converges if the following condition was met:

$$\left(\frac{1}{N}\Sigma(\frac{|E_i|}{A_i + R|U_i|})^2\right)^{1/2} < 1$$
(19)

where A_i is the absolute tolerance for degree of freedom *i*, *R* is the relative tolerance and *N* is the number of degrees of freedom.

The estimated local error is typically of the same order of magnitude as the true global error [20]. The method used to solve the time-dependent problem was the backward differentiation formula (BDF) which is a robust method that is commonly used for a wide range of problems. The algorithm for this method can be further studied in Hindmarsh *et al.* [28].

The Navier-Stokes equation in COMSOL used three artificial diffusion techniques: the Galerkin Least-Squares (GLS), crosswind and isotropic diffusion [20]. These techniques added terms to the transport equation to stabilize the solution. The GLS technique is a form of streamline diffusion that adds higher-ordered superviscosity terms and provides extra stability. Crosswind diffusion addresses the sharp gradients that may occur during the simulation. The terms add diffusion orthogonal to the streamline direction and are mostly consistent and do not alter the equation. The isotropic diffusion technique introduces an artificial diffusion coefficient to the Peclet number so that it never exceeds unity.

We have performed a Grid Convergence Index (GCI) study as described in Celik et al [29] on the linear plasma actuator by Ibrahim and Skote [18]. The results of the verification analysis conducted shows that the choice of grid settings in the order of 10^{-4} m was adequate in producing numerically acceptable results in the grid convergence analysis. To verify the grid independence for the present geometry, a simulation with a grid consisting of twice the amount of elements was performed. The coarse grid was defined as a setting where regions near the electrodes had a maximum grid size of 10^{-4} m. Fine grid setting had a maximum size of 5×10^{-5} m. The two settings are shown in Figure 8.



Figure 8: Zoomed image of the electrodes (left). Coarse grid (right, top) and Fine grid (right, bottom) settings.

Figure 9 shows the comparison of velocity profiles for the two grid settings. The maximum velocity is obtained both numerically and experimentally at 15mm from the exposed. The experimental data had a thicker boundary layer compared to the simulations results. The main factor in contention is, however, the magnitude of the maximum velocities. Both the coarse and fine grid settings resulted in a maximum of about 4.5 m/s, which was similar to that obtained experimentally.



Figure 9: Velocity profiles at 15mm downstream of the exposed electrodes for a linear actuator model ; Comparison between the two grid settings and experimental results.

The results of the verification analysis show that the coarse grid settings are adequate in producing numerically acceptable results. Similar grid settings are used in the investigations, where the channel heights are varied. Next we proceed with analysing the results obtained.

4. Results

We conducted simulations of increasing channel heights (h) as shown in Figure 2. The heights chosen were 0.004 m, 0.006 m, 0.008 m, 0.01 m, 0.02 m and 0.03 m. First, we compare simulation and experimental velocity profiles 63 mm downstream of the exposed electrodes. Next, we analyse the body force and induced velocity surface plots, at the vicinity of the plasma actuators, with decreasing channel heights. We then investigate the evolution of velocity profiles downstream of the encapsulated electrodes. In the subsequent section the

body force is examined in greater detail. Finally, we propose a hypothetical body force to account for the difference in the simulation and experimental velocity plots.

4.1. Comparing simulation and experimental velocity profiles

The velocity profiles were compared at 63 mm downstream where the flow is fully developed along the channel. The results are shown in Figure 10.



Figure 10: Velocity profiles of different channel heights, 63 mm away from the exposed electrode by Debiasi [21](a) and COMSOL (b).

For the two widest channels (heights 0.03m and 0.02m) the velocity profiles have two distinct peaks near the two walls, and the maximum value, as well as the shape, compares well between the simulations and experiments.

Debiasi *et al.*[21] observed that the flow downstream have a similar shape as a Poiseuille flow for small channel heights. This can also be seen in the simulation by COMSOL for channel heights below 0.006 m. Twin peaks are seen in larger channel heights, merging to form a single peak, or characteristically, a Poiseuille flow structure. Increasing channel heights from 0.008 m to 0.03 m results in decreasing maximum velocities in the COMSOL simulation, while the opposite trend were observed in the experiments. The maximum velocities for channel height 0.02 m and 0.03 m are in agreement between COMSOL and experimental results. Both recorded a velocity of about 1.8 m/s and 2.1 m/s at 0.02m and 0.03m respectively.

However, the simulation overestimates the velocity as compared to experiment result for the channel height of 0.008 m by about 0.7 m/s, or 27%. The difference for COMSOL and experimental results at 0.004 m decreased to about 0.5 m/s, or 14%. The results obtained by COMSOL showed that the maximum velocity at a channel height of 0.004 m was smaller than that at 0.006 m, while experimental data recorded a larger velocity at 0.004 m. To further investigate the differences in the velocity profiles for our simulation and experimental results, we analyse the body force and induced velocity in the vicinity of the plasma actuators.

4.2. Non-dimensionalised body force and induced velocity near the plasma actuators

Figure 11 shows the non-dimensionalised body force and induce velocity surface plots, near the vicinity of the actuators, for two channel heights of 0.03 m and 0.004 m. The surface plots for the non-dimensionalised body force (centre column) have a maximum of 3500, while the plots for the velocity diagrams (right column) have a maximum of 3.5 m/s.



Figure 11: Non-dimensionalised body force and induce velocity near the plasma actuators

Note that the plots for the non-dimensionalised body forces (left column) and induced velocity (right column) are symmetric with respect to the channel centreline. This is due to the similar voltage settings used in the top and bottom exposed electrodes, resulting in the same profile and magnitude of each variable. Two distinct induced flows can be seen at the channel height of 0.03 m. As the channel height decreases, the distinction between the two flows become less pronounce. At 0.004 m, the merging of the induced flow can be seen downstream from the lower electrodes. An illustration of the evolution of the velocity plots further downstream will be shown later in Figure 12.

As with the induced velocity data, the non-dimensionalised body force plots (on the left column of Figure 11) show a merging of the top and bottom body forces with decreasing channel height. At 0.004 m, this has an adverse effect on the core flow, seen in the centre of the channel. At channel height of 0.03 m, the core flow has a velocity lower than 2 m/s, as seen from the coloured surface plots. The core flow then increases to a peak as seen at the

channel height of 0.008 m as seen in Figure 10. Finally, at 0.004 m, the overlap between the top and bottom the non-dimensionalised body forces become more prominent, resulting in a decrease in core flow velocity.

4.3. Downstream velocity profiles

To further illustrate the effects of decreasing channel heights, several velocity profiles downstream the encapsulated electrode, are shown in Figure 12. Two channel heights (0.03 m and 0.004 m) are shown. The stations at which the velocity profiles are extracted were between 20 mm and 50 mm downstream, with a spacing of 5 mm between each station. The number in the figure legend denotes the stations. The results show that at larger channel heights, the two distinct flows, which had initiated near the vicinity of the plasma actuators, remain similar throughout the stations. This can be seen in the twin peaks at channel heights of 0.03 m. At the channel height of 0.004 m, the fluid begins to merge at the station furthest away from the plasma actuator (station 7, shown as the line with the star marker in Figure 12). The formation of the Poiseuille flow that was described in Debiasi *et al.* [21] can be seen. The two peaks of the induced fluid merges to form a single peak with an increasing core of the velocity profile.

4.4. Body force analysis

The results shown in Figures 11 and 12 suggest that the discrepancies in the velocity plots of Figure 10 are due to the increased overlap or interaction of body forces at lower channel heights in the simulations (0.006 m and 0.004 m). To test this hypothesis, we analyze the body force interactions by extracting body force plots along the centreline of the channel at the vicinity above the encapsulated electrodes, as shown with the centreline cut in Figure 13.

The non-dimensionalised horizontal body force profiles (left side of Figure 14) show decreasing magnitudes with increasing channel height for the full range (0.004 m to 0.03 m). This is in line with the observations of decreasing velocities obtained for channel heights of 0.008 m to 0.03 m in Figure 10. The velocity magnitude in Figure 10 showed an increasing trend from 0.004 m up to 0.008 m, which does not correspond to similar reversed trend in the

body force magnitude. To fully understand the impact of the body force on induced velocity profile, we investigate the vertical components.



Figure 12: Downstream velocity profiles for channel heights of (a) 0.03 m and (b) 0.004 m.



Figure 13: Centreline cut for the body force profiles.

The non-dimensionalised vertical body force (shown in the right side of side of Figure 14) shows similar (as the horizontal component) decreasing magnitude trends with increasing channel height. However, there are negative vertical body forces present at x = 0.002 m to 0.015 m. The negative forces indicate that they are directing towards the encapsulated electrodes, suggesting inhibition of horizontal channel flow motion. These forces are only prominent at heights of 0.004 m to 0.008 m, where they measured up to -30. For channel heights of 0.02 m and 0.03 m, the vertical body force is small and the profiles overlap in the figure. It is assumed that this vertical body force present in the vicinity of the channel wall contributes to the lower than expected velocities mentioned earlier in Figure 10.

To test this assumption, a simulation of a new body force was conducted. This hypothetical body force does not account for the any vertical components, that is, there are no vertical body force contributions to the fluid domain.



Figure 14: Horizontal (a) and vertical (b) body force profiles along the centreline of the channel.

Figure 15 shows the velocity profiles at 63 mm downstream of the exposed electrode when applying the hypothetical body force. The shapes of the velocity profiles are generally

similar to that obtained by the previous simulation (right, Figure 10). However there is a marginal increase in velocities obtained in the channel height of 0.004 m. This is shown more clearly in Figure 16, where the peak velocity of each profile is plotted versus channel height. Both of the simulations (with and without contribution from the vertical body force component) are compared with the experimental data.

The induced velocity at the channel height of 0.004 m, for the hypothetical body force that neglects the vertical influence, is 2.93 m/s. The previous simulation obtained a smaller velocity magnitude of 2.88 m/s. The simulation with the hypothetical body force also recorded a smaller maximum induced velocity of 3.08 m/s at the channel height of 0.008 m/s. The simulation results at larger channel heights (0.02 m and 0.03 m) were marginally larger compared to that obtained by the previous simulation. These results were closer to that obtained experimentally by Debiasi *et al.* [21].

However, as can be observed in Figure 16, the incorrect predictions of the velocity magnitudes at lower channel heights have not been resolved by the hypothetical model. In particular, the maximum velocity obtained at 0.006 m was still higher than that compared to at 0.004 m. This feature was not observed experimentally, as seen in Figure 16. This indicates that there are other contributing factors (than absence of the vertical body force component) that have not been captured in the modified S-H model. The effects of horizontal body forces could be more pronounced as a result of these interactions specifically at lower channel heights to produce larger velocities downstream. Future investigations of implementing plasma actuators in a channel flow could account for these features.



Figure 15: Velocity profiles of the simulation without any vertical body force contribution.



Figure 16: Comparisons between experimental and simulation results.

5. Conclusion

This investigation is an attempt to simulate a channel flow model, initiated by plasma actuators placed on top of each other. The model utilizes a modified form of the Suzen-Huang plasma actuator, which accounts for a 'dielectric shielding' condition for the potential governing the electric field. The Fokker-Planck (drift-diffusion) character was implemented on the potential governing the surface charge density. Experimental data for comparison with our simulations was obtained Debiasi *et al.* [21].

The model was able to correctly predict the maximum flow velocities at channel heights of 0.02 m and 0.03 m. The results also indicate that at larger channel heights, the two distinct peaks created by each plasma actuator remain prominent further downstream. However at lower channel heights, the model underestimates the maximum velocities. At lower channel heights, the peaks merge to a single core flow that is characteristic of a Poiseuille flow. This feature was also seen experimentally, but with larger velocity magnitudes.

Further studies to ascertain the difference in the downstream velocity magnitudes were conducted. An analysis of the induced body force and velocity plots at the vicinities of the plasma actuator indicate that merging body forces at lower channel heights may have contributed to the discrepancies obtained in the simulations. Further investigations to the body force profile, at the centreline region near the vicinity of the plasma actuators, showed that negative vertical body forces from the merging of the top and bottom actuators may have contributed to the discrepancies. Following this observation, a hypothetical model which does not account for vertical body force components was utilized. At larger channel heights, the results showed marginally improvements to the maximum induced velocities in relation to experimental data. However, the model still underpredicts the velocity magnitude at a channel height of 0.004 m. This could point to microscopic interactions that have not been captured in the present model, and a source of motivation for future investigations of plasma actuators in a channel flow configuration.

References

[1] Gad-el-Hak M. Flow Control: Passive, Active, and Reactive Flow Management. 1 ed. Virginia Commonwealth University: Cambridge University Press; 2000.

[2] Braun EM, Lu FK, Wilson DR. Experimental research in aerodynamic control with electric and electromagnetic fields. Progress in Aerospace Sciences. 2009;45:30-49.

[3] Corke TC, Enloe CL, Wilkinson SP. Dielectric barrier discharge plasma actuators for flow control Annual Review of Fluid Mechanics. 2010;42:505-29.

[4] Moreau E. Airflow control by non-thermal plasma actuators. J Phys D: Appl Phys. 2007;40:605-36.

[5] Cattafesta LN, Sheplak M. Actuators for Active Flow Control. Annual Review of Fluid Mechanics. 2011;43:247-72.

[6] Zhang PF, Wang JJ, Feng LH, Wang GB. Experimental study of plasma flow control on highly swept delta wing. AIAA Journal. 2010;48:249-52.

[7] Sosa R, Arnaud E, Memin E, Artana G. Study of the flow induced by a sliding discharge.IEEE Transactions on Dielectrics and Electrical Insulation. 2009;16:305-11.

[8] Orlov DM. Modelling and Simulation Of Single Dielectric Barrier Discharge Plasma Actuators [Dissertation]. Notre Dame, Indiana: University of Notre Dame; 2006.

[9] Orlov DM, Apker T, He C, Othman H, Corke TC. Modeling and Experiment of Leading Edge Separation Using SDBD Plasma Actuators. AIAA Paper No 2007-0877. 2007.

[10] Corke TC, Post ML, Orlov DM. Single Dielectric Barrier Discharge Plasma Enhanced Aerodynamics: Concepts, Optimization and Applications J Propul Power. 2008;24:935-45.

[11] Corke TC, Post ML, Orlov DM. Single Dielectric Barrier Discharge Plasma Enhanced Aerodynamics: Physics, Modeling and Applications (Review Article). Experimental Fluids. 2009:1-26. [12] Massines F, Rabehi A, Decomps P, Gadri RB, SeÌgur P, Mayoux C. Experimental and theoretical study of a glow discharge at atmospheric pressure controlled by dielectric barrier. J Appl Phys. 1998;83:2950-7.

[13] Boeuf JP, Pitchford LC. Electrohydrodynamic force and aerodynamic flow acceleration in surface dielectric barrier discharge. J Appl Phys. 2005;97:1-10.

[14] Rouffet J-B. Plasma Actuator Influence on Air Flow. In: COMSOL, editor. COMSOLConference 2008. Hannover: COMSOL; 2008. p. 1-4.

[15] Roth JR, Sherman DM, Wilkinson SP. Boundary layer flow control with a one atmosphere uniform glow discharge surface plasma. AIAA Paper 98-0328. 1998.

[16] Suzen YB, Huang PG, Jacob JD, Ashpis DE. Numerical simulations of plasma based flow control applications. AIAA Paper No 2005-4633. 2005.

[17] Orlov DM, Corke TC, Patel MP. Electric Circuit Model for the Aerodynamic Plasma Actuator. AIAA Paper No 2006-1206. 2006.

[18] Ibrahim IH, Skote M. Boundary condition modifications of the Suzen-Huang plasma actuator model. International Journal of Flow Control. 2011;3:111-31.

[19] Ibrahim IH, Skote M. Simulations of the linear plasma synthetic jet actuator utilizing a modified Suzen-Huang model. Phys Fluids. 2012;24 (113602).

[20] COMSOL. COMSOL Multiphysics Modeling Guide. COMSOL 42 2011.

[21] Debiasi M, Jiun-Ming L. Experimental Study of a DBD-Plasma Driven Channel Flow AIAA Paper 2011-954. 2011.

[22] Suzen YB, Huang PG. Simulations of flow separation control using plasma actuators. AIAA Paper No 2006-877. 2006.

[23] Suzen YB, Huang PG, Ashpis DE. Numerical simulations of flow separation control in low-pressure turbines using plasma actuators. AIAA Paper No 2006-937. 2007. [24] Reasor Jr DA, LeBeau Jr RP, Suzen YB. Unstructured grid simulations of plasma actuator models. AIAA Paper No 2007-3973. 2007.

[25] Santhanakrishnan A, Reasor Jr DA, LeBeau Jr RP. Characterization of linear plasma synthetic jet actuators in an initially quiescent medium. Phys Fluids. 2009;21.

[26] Mamunuru M, Guo S, Poon D, Burman D, Simon T, Ernie D, et al. Separation control using plasma actuator: Simulation of plasma actuator. AIAA Paper No 2009-4186. 2009.

[27] Ibrahim IH, Skote M. Effects of the scalar parameters in the Suzen-Huang model on plasma actuator characteristics. International Journal of Numerical Methods for Heat and Fluid Flow. 2013;23(6): 1076-1103.

[28] Hindmarsh AC, Brown PN, Grant KE, Lee SL, Serban R, Shumaker DE, et al. SUNDIALS: Suite of nonlinear and differential/algebraic equation solvers. ACM Transactions on Mathematical Software. 2005;31:363-96.

[29] Celik IB, Ghia U, Roache PJ, Freitas CJ, Coleman H, Raad PE. Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications. Journal of Fluids Engineering. 2008;130:078001-4.