Available online at: http://brsj.cepsbasra.edu.iq

ISSN -1817 -2695

Numerical Investigation of Fluid Boundary layer over Flat Surface

Amir Hussein Ali*

University of Basrah, College of Education for Pure Science, Department of Physics.

***Corresponding author:**

E-mail address: amirss_3000@yahoo.com

Abstract

In present study numerical investigation of two kinds of fluids namely air and water flow on flat surface have been performed to investigate the boundary layer. Computational fluid dynamics has been used as a numerical technique to solve continuity equation and momentum equation. The air has been used in this study due to its wide importance in aerodynamics applications. In addition water has been used for comparison. A 1 m length flat surface has been used as a base for boundary layer fluid flow. The altitude of the computational domain which includes the free stream fluid flow was 0.1 m. The fluid velocity at the leading edge of the surface was 10 m/s for both air and water. Flow on smooth surface has been considered. Viscous fluid flow was used and hence no-slip condition has been adopted for both air and water. The Reynolds number (Re) was based upon the distance x from the leading edge as a characteristic length. The boundary layer thickness (δ) has been calculated. In addition wall shear stress (τ) has been studied for both air and water. Also some other parameters have been calculated such as displacement thickness (δ^*), momentum thickness (θ) and the coefficient of drag (C_D). The results showed that the Reynolds number of water is bigger than the Reynolds number of air based on the conditions of the current study. The boundary layer of air is laminar from leading edge until 0.8 m. Beyond 0.8 m up to the surface end, the boundary layer is in the transition region. Also the boundary layer of water transited to turbulent region at 0.2 m while the air boundary layer is still laminar. As expected, the momentum thickness (θ) showed similar behavior as boundary layer thickness (δ).The displacement thickness of air is higher than the displacement thickness of water.The results also showed that the air drag coefficient has less value than water drag coefficient from approximately 0.2 m up to the downstream. The results showed that water flow affected the flat surface by bigger wall shear stress than air.

Keywords: Computational fluid dynamics, Boundary layer**,** Fluid flow.

1. Introduction

When a real free stream fluid flows past over a solid flat surface, then the velocity magnitude of the fluid at the solid surface must be the same as that of the solid surface. Hence in case of the solid surface is stationary, the velocity magnitude of the fluid at the solid surface must be zero because of no slip condition. The velocity increases from zero at the flat surface to the velocity of the mainstream. This region in the vicinity of the solid flat surface which starts at leading edge of the solid surface is generally a narrow region and called boundary layer. Due to the viscous condition and the large shear stress, the velocity gradient is large in the boundary layer region. Therefore the boundary layer thickness increases with the distance x along the solid surface. The development of boundary layer has been illustrated in figure 1. Near the leading edge of the solid surface, the flow is laminar and the boundary layer is said to be laminar boundary layer. As the thickness of the boundary layer increases in the downstream direction, the laminar boundary layer becomes unstable. Hence the motion of the fluid is disturbed and transferred from laminar to turbulent boundary layer through transition region. In literature several papers interested with turbulent boundary layer [1-3]. The velocity profile in the laminar boundary layer can be approximated by a parabola as shown in figure 2.

The shearing stress τ at the wall is related to the velocity gradient at the wall by:

$$\tau = \mu\left(\frac{\partial u}{\partial y}\right) \qquad (1)$$

Where: μ is the dynamic viscosity of the fluid.

The Reynolds number Re in this case is based upon the distance *x* from the leading edge as the characteristic length:

$$Re = \frac{U\rho x}{\mu} = \frac{Ux}{\nu} \qquad (2)$$

Where: ρ is the density of the fluid, U is the mainstream velocity and $\nu = \frac{\mu}{\rho}$ is the kinematic viscosity.

It can be noticed that in recent years the boundary layer theory received increasing attention [4].

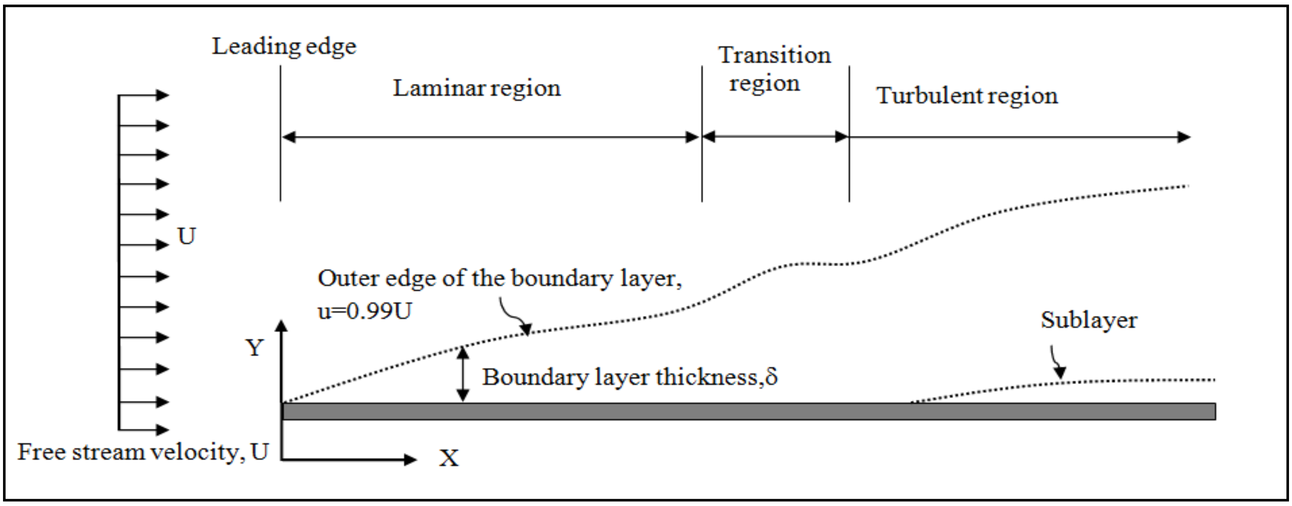

Fig.1. Development of the boundary layer along a solid flat surface.

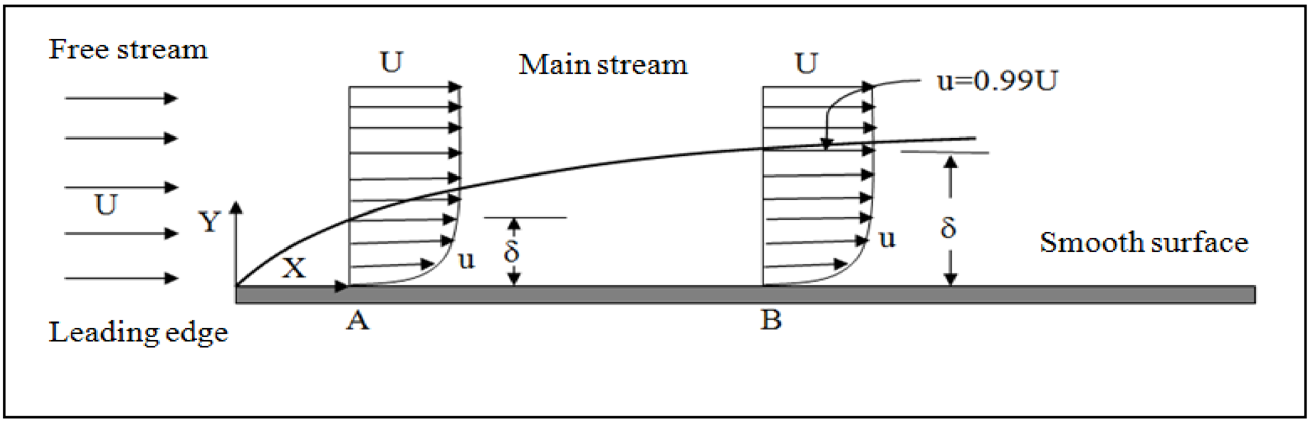

Fig.2. A laminar boundary layer over flat surface.

For practical purpose, the boundary layer region is defined in the following alternative ways:

(a) The Boundary layer thickness (δ):

δ is the distance y where u becomes 99%U, i.e. whene the difference between u and U remains 1%.

(b) Displacement thickness δ*:

The displacement thickness as explained in leterature is the distance y by which the external free stream of the fluid is effectively displaced due to the formation of the boundary layer.

For incompreeesible flow[5]:

$$\delta^* = \int_0^{\delta}\left(1-\frac{u}{U}\right)dy \qquad (3)$$

$\delta^* \cong \delta/3$ for a laminar profile.

$\delta^* \cong \delta/8$ for a turbulent profile.

(c) The Momentum thickness (θ):

The Momentum thickness is the distance y cooresponding to the momentum defect brought about by the actual profile in relation to the uniform profile.

For incompreessible profile, θ given as [5]:

$$\theta = \int_0^{\delta}\frac{u}{U}\left(1-\frac{u}{U}\right)dy \qquad (4)$$

$\theta \cong \delta/8$ for a laminar profile.

$\theta \cong (7/72)\,\delta$ for a turbulent profile.

The differential form of the continuity equation for an incompressible flow in two dimensions given as [6]:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad (5)$$

The x component of momentum for steady and two-dimensional fluid flow is [6]:

$$\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}=-\frac{\partial p}{\partial x}+\mu\frac{\partial^2 u}{\partial x^2}+\mu\frac{\partial^2 u}{\partial y^2} \qquad (6)$$

Similarly, the y component of momentum can be given as [6]:

$$\rho u\frac{\partial v}{\partial x}+\rho v\frac{\partial v}{\partial y}=-\frac{\partial p}{\partial y}+\mu\frac{\partial^2 v}{\partial x^2}+\mu\frac{\partial^2 v}{\partial y^2} \qquad (7)$$

As a literature background, the skin friction serves as important quantity in a variety of flow-control applications. Therefore the possibility of changing this parameter even by a few percent is desirable. More recently, P. Niegodajew, et al. studied skin friction of turbulent boundary layer flow under the adverse pressure gradient conditions [7]. On the other hand, Y. Li et al. studied the laminar boundary layer separation over a fluttering panel induced by an oblique shock wave. The effectiveness of a surface velocity feedback strategy and the effect of the boundary layer thickness are investigated. The investigation has been performed in terms of the panel vibration behavior, surface pressure distribution, and laminar boundary layer separation [8]. H. Medina et al. studied a novel laminar kinetic energy model for the prediction of pre transitional velocity fluctuations and boundary layer transition. A new model of laminar kinetic energy has been proposed and has been validated [9]. Also numerical study of a flat-plate boundary layer has great attention in literature [10].

2. Computational Details

Smooth and flat surface of length 1 m has been used as a base for boundary layer fluid flow investigation. The altitude of the computational domain which include the boundary layer and the free stream of fluid flow was 0.1 m as illustrated in figure 3. The fluid materials were air and water which have been investigated over the flat surface and compared to each other. The fluid velocity at the leading edge of the flat surface was 10 m/s for both air and water. Viscous fluid flow has been considered and hence no-slip condition has been adopted for both air and water. The Reynolds number (Re) was based upon the distance *x* from the leading edge as a characteristic length. Computational fluid dynamic has been used and finite volume method has been performed as a numerical technique to solve the continuity equation for an incompressible fluid flow in two dimensions and momentum steady two-dimensional equation [11]. The study has been performed under room temperature for both fluids and the flat surface. Hence no heat transfer has been considered. Also atmospheric pressure has been considered during the calculation of fluid flow over the flat surface. The meshing size of the computational domain was fine near the wall surface to provide high resolution. In the boundary conditions, the fluid which inters the computational domain has been set as velocity-inlet whereas that leave the computational domain has been set as outflow. The monitor convergence has been set to $1e^{-6}$. The boundary layer thickness (δ) and wall shear stress (τ) have been investigated for both air and water over the flat surface. Also some other parameters such as displacement thickness (δ^*), momentum thickness (θ) and the coefficient of drag (C_D) have been calculated for both air and water and have been compared to each other. In order to obtain high resolution for the boundary layer, fine meshing has been used for the computational domain. The length of the flat surface was long enough to investigate Reynolds number (Re) in both laminar and turbulent boundary layer region. Also the length of the flat surface was including the transition region. In fact the value of the Reynolds number (Re) at the transition location is complex function of various parameters involved such as roughness of the surface and surface curvature. To avoid the roughness effect on the boundary layer velocity near the surface, the flat surface has been chosen to be smooth.

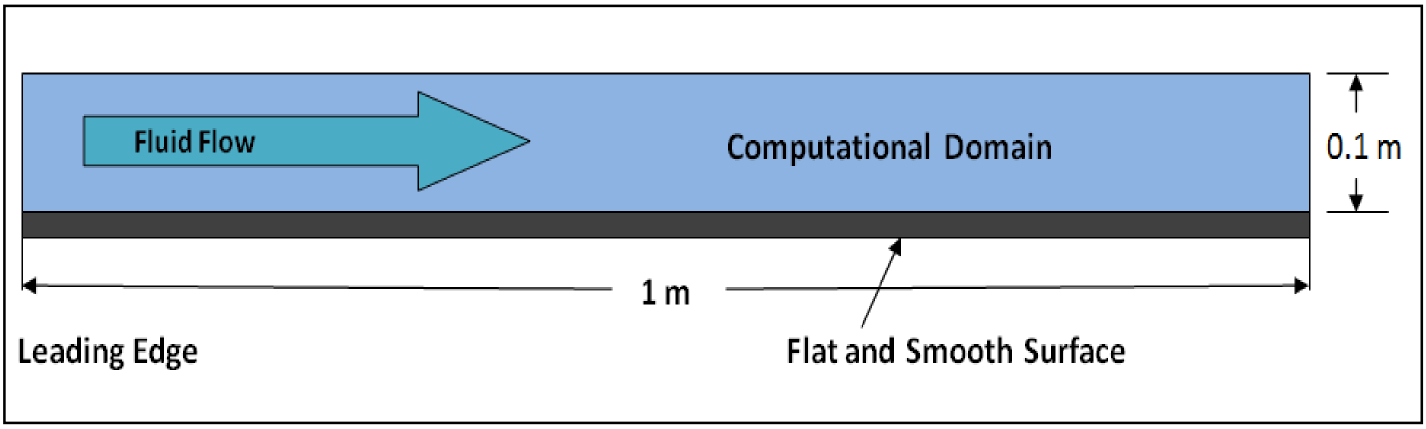

Fig.3. Computational sketch of fluid flow over flat surface.

3. Results and discussion

Computational investigation of air flow over flat surface showed the air streamlines and clear boundary layer constructed over the smooth surface as illustrated in figure 4. The boundary layer thickness increased from leading edge toward the end of the solid surface. The boundary layer is laminar from the leading edge until 0.8 m. Beyond 0.8 m up to the surface end, the boundary layer in transition region. These results are consistent with the Reynolds number (Re) measurements of air and water flow in this study as shown in figure 5. The Reynolds number of the air is less than the critical value of which transition takes place from laminar to turbulent flow. The Reynolds number Re at the transition location is complex function of various parameters such as the roughness of the surface [12].

Also the curvature of the solid surface and some measure of the disturbances outside the boundary layer in free stream region can affect the value of the Reynolds number. Over a flat surface of a sharp leading edge in a typical air stream, the transition take place at a distance *x* from the leading edge given by Re=2×10^5 to 3×10^6 [13]. Hence the air boundary layer did not reach the turbulent region because of small Reynolds number for air in these conditions of flow. On the other hand the computational investigation of water flow over the flat surface showed that the transition from

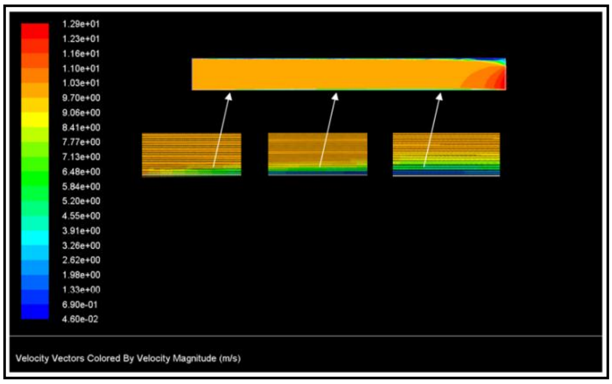

Fig.4. Simulation of air flow over flat surface.

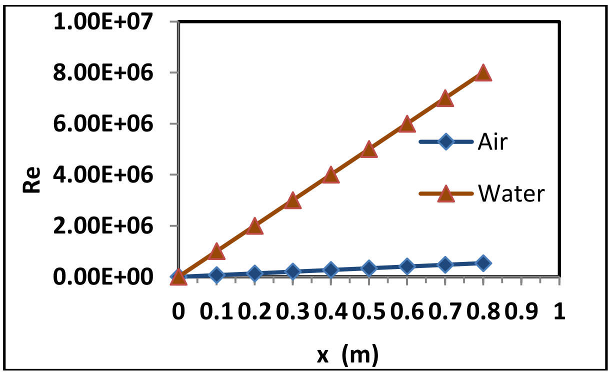

Fig.5. Reynolds numbers for air and water along the flat surface.

laminar to turbulent commenced in position near the leading edge of the flat surface at approximately 0.2 m as illustrated in figure 6. The actual transition of fluid from laminar flow to turbulent flow boundary layer may occur over a region of the plate, not at a specific single position. This early transition is due to the large Reynolds number of water as illustrated in figure 5. The boundary layer thickness increased from leading edge toward the end of the surface with transition from laminar boundary layer to turbulent boundary layer. Also the top of the free stream of water flow included vortexes until the end of the surface. This is because of high density and high viscosity of water in addition to high Reynolds number compared to air.

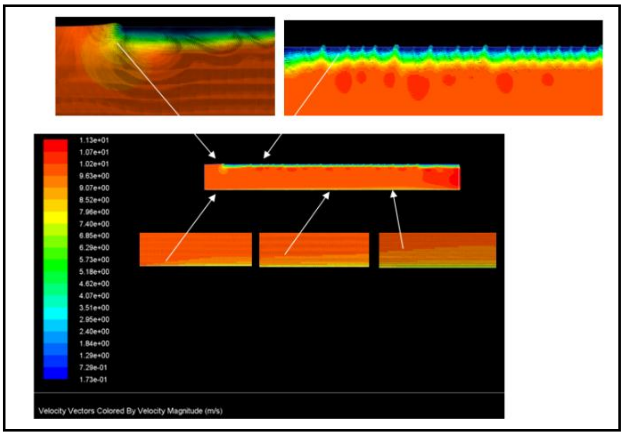

Fig.6. Simulation of water flow over flat surface.

The boundary layer thickness (δ) for air and water have been plotted and compared to each other as depicted in figure 7. The result showed clear difference between the two cases after 0.2 m. This is because of big differences of Reynolds number between air and water. Also the boundary layer of water transited to turbulent region at 0.2 m while the air boundary layer is still laminar. This is consistent with the theoretical background. As expected, the momentum thickness (θ) showed similar behavior as boundary layer thickness (δ) as shown in figure 8.

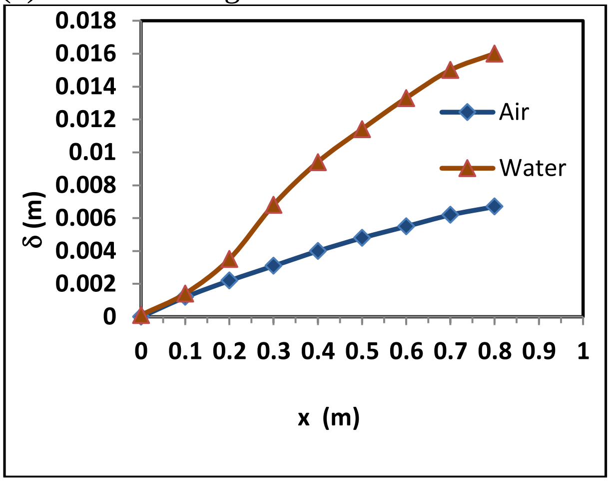

Fig.7. The boundary layer thickness for air and water along the flat surface.

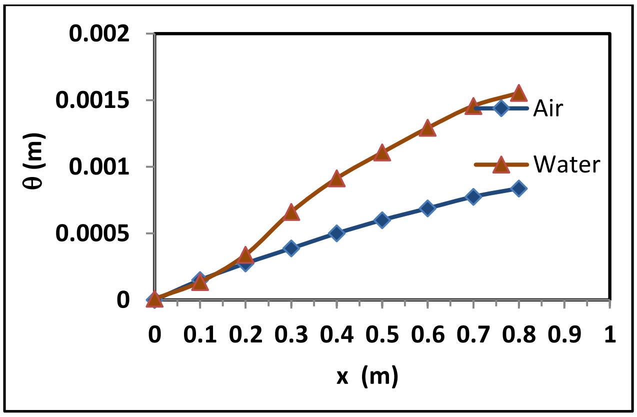

Fig.8. The momentum thickness for air and water along the flat surface.

The displacement thickness which is the distance y by which the free stream of fluid flow is effectively displaced because of the formation of the boundary layer have been calculated from the simulation results for both air and water and illustrated in figure 9. The displacement thickness of air is higher than the displacement thickness of water. This is beacause of the turbulent boundary layer thickness of water is larger than the laminar boundary layer thickness of air. Also this result is consistant with the theortical background.

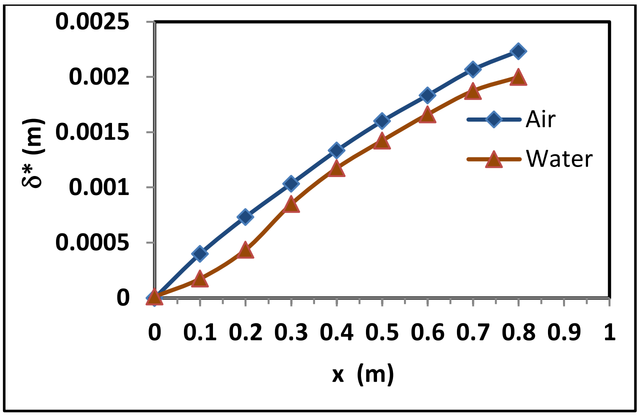

Fig.9. The displacement thickness of air and water along the flat surface.

In fluid dynamics, the drag coefficient C_D is used to measure the resistance of an object in a fluid environment. The lower drag coefficient indicates that the object will have less aerodynamic or hydrodynamic drag. However the drag coefficient is associated with a particular surface area. In this study the drag coefficients for air and water have been calculated and illustrated in figure 10. The results showed that the air drag coefficient has less value than water drag coefficient from approximately 0.2 m up to the downstream. This is because water flow transitioned from laminar to turbulent flow at 0.2 m. Furthermore water has bigger density and viscosity compared to air. It could be noted that this result is quite consistent with the available information in literature [14].

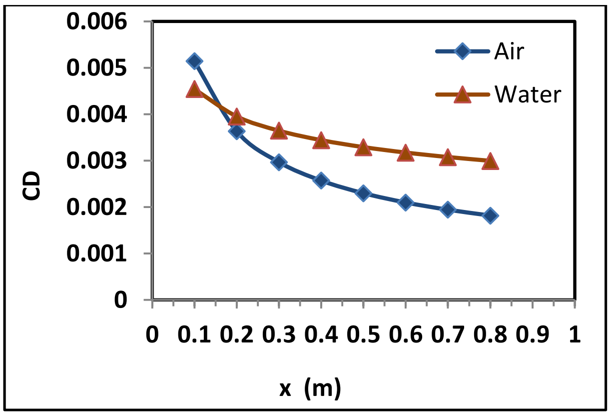

Fig.10. The drag coefficients for air and water along the flat surface.

It is well known that a fluid at rest cannot resist shearing forces. Therefore, under the action of such forces it deforms continuously. Also the resistance to the shearing forces in a fluid appears only when the fluid is in movement action. For solids the resistance to a shear deformation depends on the deformation itself. Hence the shear stress is a function of the shear strain. However for fluids the shear stress is a function of the rate of the strain. In current study and for more information and more details the wall shear stress of air and water has been plotted in figures 11 and 12 respectively. The results showed that water flow affected the flat surface by bigger wall shear stress than air. This result is reasonably accepted because of bigger density and viscosity of water compared to air density and viscosity.

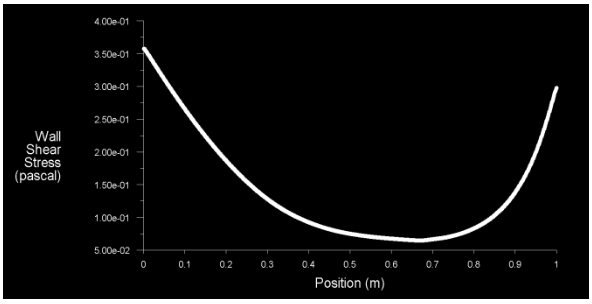

Fig.11. The wall shear stress of air.

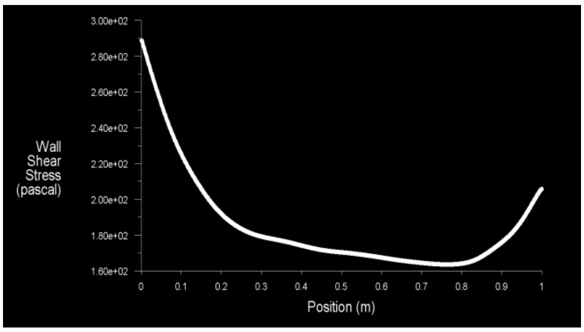

Fig.12. The wall shear stress of water.

4. Conclusion

Boundary layers of air and water flow have been investigated over flat surface. Numerical investigation by using computational fluid dynamics has been used as a numerical technique to solve the continuity equation and momentum equation. A 1 m length of smooth and flat surface has been used as a base for boundary layer fluid flow. Also the altitude of the computational domain which includes the free stream fluid flow was 0.1 m. Furthermore, the fluid velocity at the leading edge of the surface was 10 m/s for both air and water. Viscous fluid flow was used and hence no-slip condition has been adopted for both air and water. The Reynolds number (Re) was based upon the distance *x* from the leading edge as a characteristic length. Different parameters have been calculated and studied such as boundary layer thickness (δ), wall shear stress (τ), displacement thickness (δ^*), momentum thickness (θ) and the coefficient of drag (C_D). The results showed clear difference between air and water for all the parameters which have been calculated and studied in this study. This is mainly due to the Difference between the Reynolds number of water and air where the results showed that the Reynolds number of water is bigger than the Reynolds number of air based on the conditions of the current study such as the values of viscosity and density of the fluid flow. The boundary layer of air is laminar from leading edge until 0.8 m. Beyond 0.8 m up to the surface end, the boundary layer in transition region. Also the boundary layer of water transited to turbulent region at 0.2 m while the air boundary layer is still laminar. The displacement thickness of air is higher than the displacement thickness of water. The results showed that water flow affected the flat surface by bigger wall shear stress than air. Also the top of the free stream of water flow included vortexes until the end of the surface. The results showed that the air drag coefficient has less value than water drag coefficient from approximately 0.2 m up to the downstream. As expected, the momentum thickness (θ) showed similar behavior as boundary layer thickness (δ).

Acknowledgements

The author gratefully acknowledges the Department of Physics, College of Education for Pure Science, University of Basrah.

References

[1] J. Cousteix, J. Mauss, Interactive boundary layers in turbulent flow, C. R. ScienceDirect, Mecanique, 335 (2007), pp. 590–605.

[2]E. Ciappi, F. Magionesi ,Characteristics of the turbulent boundary layer pressure spectra for high-speed vessels, Journal of Fluids and Structures 21 (2005), pp. 321–333.

[3] A. Jocksch, L. Kleiser, Growth of turbulent spots in high-speed boundary layers on a flat plate, International Journal of Heat and Fluid Flow 29 (2008), pp. 1543–1557.

[4] K. Mayes, Boundary-Layer Theory, Springer-Verlag Berlin Heidelberg (2017).

[5] R. K. Rajput, Fluid Mechanics and Hydraulic Machines, S.CHAND & COMPANY LTD, (1998).

[6]J. J. Bertin, R. M. Cummings, P. V. Reddy, AERODYNAMICS FOR ENGINEERS, Pearson Education Limited (2014).

[7] P. Niegodajew, A. Drozdz, W. Elsner, A new approach for estimation of the skin friction in turbulent boundary layer under the adverse pressure gradient conditions, International Journal of Heat and Fluid Flow 79 (2019), 108456.

[8] Y. Li, H. Luo, X. Chen, J. Xu, Laminar boundary layer separation over a fluttering panel induced by an oblique shock wave, Journal of Fluids and Structures 90 (2019), pp. 90–109.

[9] H. Medina , A. Beechook, H. Fadhila, S. Aleksandrova, S. Benjamin, A novel laminar kinetic energy model for the prediction of pretransitional velocity fluctuations and boundary layer transition, International Journal of Heat and Fluid Flow 69 (2018), pp. 150–163.

[10] J. P. Hickey, S. Blakie, C. Gray, X. Wu, Further studies on the statistics and structures of flat-plate boundary layer with passing wakes, International Journal of Heat and Fluid Flow 31 (2010), pp. 315–326.

[11] J. Tu, G. H. Yeoh, C. Liu, Computational Fluid Dynamics, A Practical Approach, Elsevier Ltd (2018).

[12] M. P. Schultz, K. A. Flack, Turbulent Boundary Layers Over Surfaces Smoothed by Sanding, Journal of Fluids Engineering, by ASME, (2003), pp. 863-870.

[13] B. R. Munson, T. H. Okiishi, W. W. Huebsch, A. P. Rothmayer, Fundamentals of Fluid Mechanics, John Wiley & Sons, Inc, (2013).

[14] E. L. Houghton and P. W. Carpenter, Aerodynamics for Engineering Students, Butterworth-Heineman, (2003).

التحقق العددي لطبقة حدود الموائع على سطح مستو

الخلاصة

في الدراسة الحالية تم إجراء تحقيق عددي لنوعين من الموائع وهما تدفق الهواء والماء على سطح مستو لاستكشاف الطبقة الحدودية. تم استخدام ديناميات الموائع الحسابية كتقنية عددية لحل معادلة الاستمرارية ومعادلة الزخم. تم استخدام الهواء في هذه الدراسة بسبب أهميته الواسعة في تطبيقات الديناميكا الهوائية. بالإضافة إلى ذلك ، تم استخدام الماء للمقارنة. تم استخدام سطح مستو بطول 1 متر كقاعدة لتدفق مائع الطبقة الحدودية. كان ارتفاع المجال الحسابي الذي يتضمن مائع التدفق الحر 0.1 متر. كانت سرعة المائع عند الحافة الأمامية للسطح 10 م / ث لكل من الهواء والماء. وقد تم التدفق على سطح أملس. تم استخدام تدفق الموائع اللزجة وبالتالي تم اعتماد حالة عدم الانزلاق للهواء والماء. اعتمد عدد رينولد (Re) على المسافة x من الحافة الأمامية كطول مميز. تم حساب سمك الطبقة الحدودية (δ) بالإضافة إلى ذلك تمت دراسة إجهاد قص السطح (τ) للهواء والماء. كما تم حساب بعض المعاملات الأخرى مثل سمك الإزاحة (δ^*) ، سمك الزخم (θ) ومعامل السحب (C_D). أوضحت النتائج بأن عدد رينولد للماء أكبر من عدد رينولد للهواء حسب ظروف الدراسة الحالية. تكون الطبقة الحدودية للهواء صفائحية من الحافة الأمامية حتى 0,8 م. أبعد من 0,8 متر حتى نهاية السطح تكون الطبقة الحدودية في المنطقة الانتقالية. في حين تنتقل الطبقة الحدودية من الماء إلى المنطقة المضطربة عند 0,2 م بينما لا تزال الطبقة الحدودية للهواء صفائحية. كما هو متوقع , اظهر سمك الزخم (θ) سلوكا مشابها لسمك الطبقة الحدودية (δ). كما أظهرت الدراسة بأن سمك الإزاحة للهواء أعلى من سمك الإزاحة للماء. أظهرت النتائج أن معامل سحب الهواء أقل قيمة من معامل السحب المائي من حوالي 0,2 متر إلى المصب. أوضحت النتائج أن تدفق المياه أثر على السطح من خلال اجهاد قص أكبر من الهواء.

الكلمات الدليلية: ديناميكية الموائع الحسابية ، طبقة الحدود ، تدفق الموائع.