

## CFD Study of NACA 0018 for Diffuser Design of Tidal Current Turbines

Nasir Mehmood, Zhang Liang and Jawad Khan

Deepwater Engineering Research Center, Harbin Engineering University, Harbin, 150001, China

**Abstract:** This study is focused on diffuser augmented tidal current turbines that capture the kinetic energy in a tidal stream. The energy that can be extracted from tides is proportional to the cube of the current velocity. The role of the diffuser in diffuser augmented tidal turbines is to help accelerate the incoming current velocity. Consequently, the efficiency of the turbine can be significantly increased by using a diffuser. The research community is investing considerable time and financial resources in this growing domain. The diffuser augmented tidal turbines research data is rather scarce due to their emerging nature, large and costly research and development setup, startup cost and proprietary issues. The purpose of this study is to investigate the effect of length and angle on NACA 0018 airfoil for diffuser design. CFD simulation is carried out to investigate velocity and mass flow rate at the throat. The drag force due to diffuser installation is also calculated. Velocity inside the diffuser increases with diffuser length and angle of attack. Velocity increases up to stall angle and then drops due to flow separation. The drag force is also dominant compared to lift coefficient near stall angle region.

**Keywords:** CFD simulation of diffuser, Diffuser Augmented Tidal Turbine (DATT), ducted turbine, numerical simulation of diffuser, shrouded turbine, tidal current turbine, tidal energy

### INTRODUCTION

Oceans, covering more than 70% of earth surface, are a great resource of untapped and unexplored energy (Mehmood *et al.*, 2012). Tidal energy is one of the oldest forms of energy used by mankind. Tidal energy can be harnessed by two approaches. The conventional method to utilize tides is by capturing potential energy, i.e., by tidal barrage and tidal fence. The modern trend is to capture their kinetic energy by using underwater devices similar to wind turbines. The building of dam or barrage across a tidal bay is not only pricey but also harms the environment. The change in natural flow from building a dam has a fatalistic impact on aquatic and shoreline ecosystem. Tidal energy is not only renewable; unlike fossil fuels, it has no emissions (Mehmood *et al.*, 2012). Fossil fuels are responsible for global warming and acidic rains. The tidal energy will also decrease the need for nuclear power, thus, eradicating the radiation risks.

The horizontal axis tidal turbine is a popular tidal current technology (Mehmood *et al.*, 2011). The efficiency of a horizontal axis tidal turbine depends on marine current velocity and water depth. The ideal marine current velocity for a horizontal axis tidal turbine is around 2 m/s (Mehmood *et al.*, 2012).

However, the average available marine current velocity around the globe is around 1 m/s. In order to harness tidal energy at such low velocity, a much bigger turbine system is required. However, bigger turbine system creates issues such as water depth limitation, huge support structure and increased drag of system. Thus, increase in incoming current velocity is severely needed. Diffuser design for wind turbines is getting very popular and lot of research in under way around the globe (Gilbert *et al.*, 1978; Igra, 1981; Gilbert *et al.*, 1983).

Researchers are investing considerable time and effort in this domain. NACA airfoils have a huge potential as these airfoils can be used to design a diffuser. There is no study available in literature, up until now, which concentrate upon the effect of length and angle of attack at the same time. This study is very important as it gives the insight behavior of the fluid flow and also discussed the potential for NACA 0018 for diffuser design. This study is focused on diffuser design for tidal current turbines. The study explores various diffuser shapes to enhance the performance of the diffuser. The purpose of this study is to present the numerical simulation of NACA 0018 airfoil diffuser for tidal current turbine. CFD simulation of velocity profile at the throat is presented in detail.

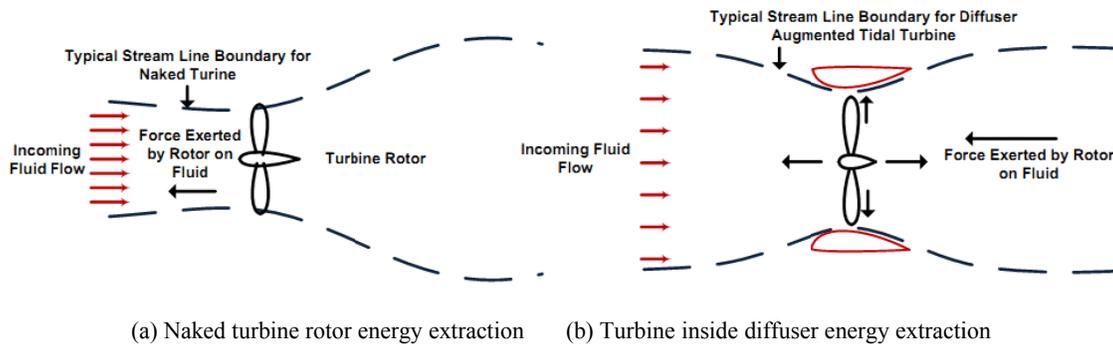


Fig. 1: Tidal energy extraction scheme



Fig. 2: Mesh block cutting scheme

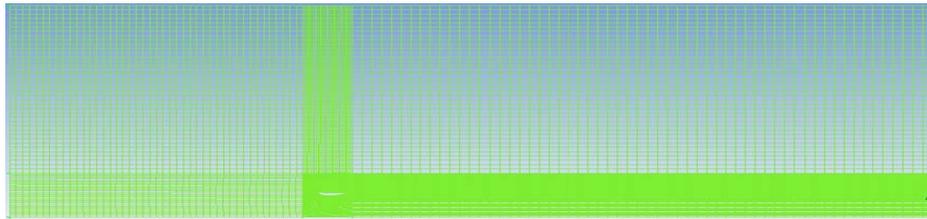


Fig. 3: Mesh generation

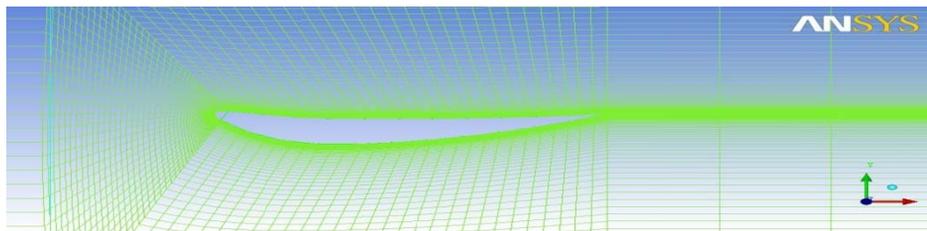


Fig. 4: Close up view of meshed airfoil

The effect of length and angle of rotation on maximum velocity, average velocity, mass flow rate and drag is explored. This research serves as the foundation for using CFD tools for diffuser design for tidal current turbines.

**Diffuser operating principle:** There is very little information available for the use of diffuser around the tidal current turbine. However, diffusers for wind turbines have been investigated in detail.

Naked turbines extract energy from the fluid by reducing fluid velocity while the fluid is passing through the turbine rotor. As a result, the stream lines expand to maintain continuity. Betz's limit dictates the percentage of kinetic energy that can be harnessed from a fluid, which is 59.3% for a single actuator disk. Single actuator disk refers to the cross-sectional area across which the energy is extracted as the flow is passing. In past, two basic principles for increasing the power of turbine were reported:

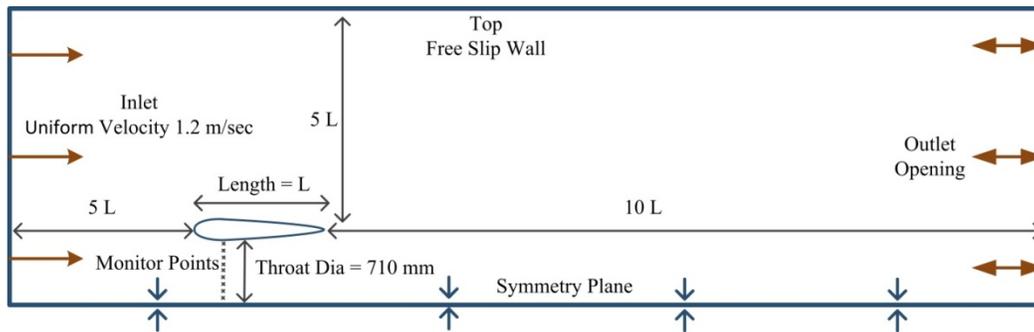


Fig. 5: Schematic view of domain and boundary conditions

- Increasing the mass flow rate.
- Facilitating the turbulent mixing of the wake behind rotor and diffuser.

The net effect or sole purpose is to increase the mass flow rate by creating a suction effect ahead of the diffuser inlet. This suction effect is created by creating a pressure drop at the outlet of diffuser. According to momentum theory, when a naked turbine is operating at maximum Betz limit, the fluid flow is decelerated to 2/3 of the free fluid stream velocity as it reaches the turbine rotor plane. Due to decrease in velocity of fluid, there is an increase in pressure just in front of the turbine rotor, causing the mass flow to push sideways around the rotor of turbine as shown in Fig. 1(a). The influence of continuous and incompressible flow is that the turbine rotor exploits the kinetic energy contained in a free fluid stream from by an effective surface which is 2/3 of the swept area of the turbine rotor. In order to maximize the effective surface area for capturing the kinetic energy from the fluid stream, a mechanism is required to exert a perpendicular force on the incoming fluid. This can be achieved by placing a diffuser around the turbine rotor. According to Newton's third law, it can be demonstrated that the incoming fluid stream will exert a counter acting force to achieve force equilibrium. This counteracting force can only be achieved if mass flow rate is increased. Thus increase in mass flow rate is achieved by using a diffuser. This outward force will cause the area ahead of the turbine rotor to widen as shown in Fig. 1(b).

**Diffuser modeling and boundary conditions:** NACA 0018 airfoil is used to create the 2D models of the diffuser. Modeling is carried out using MATLAB. The diffuser is explored at fourteen different lengths, i.e., 400, 450, 500, 550, 600, 650, 700, 710, 750, 800, 850, 900, 950 and 1000 mm, respectively. The diffuser is also explored at seventeen different angles of attack, i.e.,

0 to 16°. The diameter of the throat is fixed at 710 mm. For in-depth analysis, 10 monitoring points are created at the throat of the diffuser. For the dimensionless analysis coefficient of velocity and coefficient of mass flow rate has been introduced. Coefficient of velocity can be express as:

$$C_v = \frac{\text{Average velocity at throat}}{\text{Inlet velocity}} = \frac{V_{Avg}}{V_{Inlet}} \quad (1)$$

Coefficient of mass flow rate can be express as:

$$C_q = \frac{\text{Mass flow rate at throat}}{\text{Mass flow rate at inlet}} = \frac{Q_t}{Q_i} \quad (2)$$

$$C_q = \frac{Q_t}{Q_i} = \frac{V_{Avg} \text{ Area of throat}}{V_{inlet} \text{ Area of inlet}} = \frac{V_{avg} \pi r_t^2}{V_{inlet} \pi r_i^2} \quad (3)$$

Ansys ICEM is used to create the structured mesh. ANSYS scripting language is used to automate the mesh. Data points generated from MATLAB are imported into the ICEM as "Formatted Data Points". The domain inlet is taken 5 times the length of the diffuser. Outlet of the domain is taken as 10 times the length of the diffuser. Top is taken 5 times the length of the diffuser. Diffuser is meshed along symmetry due to the symmetrical shape and to reduce the mesh size and computational time. The block cutting scheme is shown in Fig. 2. Diffuser 2D mesh is shown in Fig. 3 and 4. Schematic view of the domain is shown in Fig. 5. The y+ value is less than 10 for mesh spacing. K-omega SST turbulence model is used due to its good performance for predicting boundary-layer separation with adverse pressure gradient. The simulation has been carried out using commercially available software Ansys CFX. The simulation has already been validated (Mehmood *et al.*, 2012).

## RESULTS AND DISCUSSION

In Fig. 6, maximum velocity at throat is plotted against angle of attack from zero degree minimum to 16

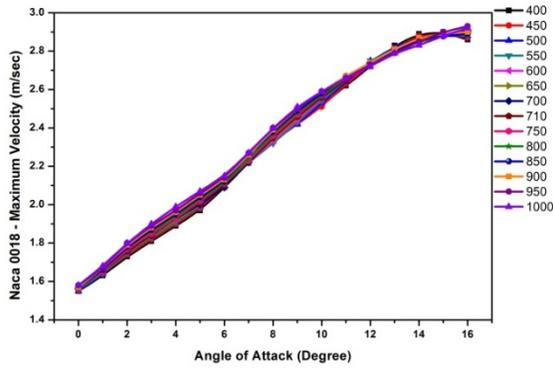


Fig. 6: Maximum velocity Vs angle of attack

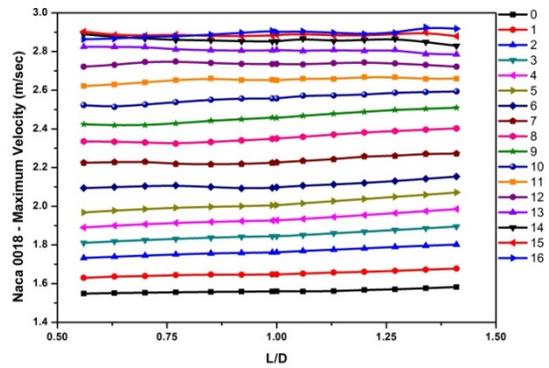


Fig. 7: Maximum velocity Vs L/D ratio

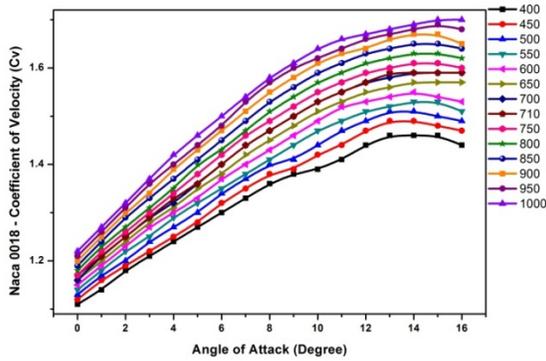


Fig. 8: Coefficient of velocity Vs angle of attack

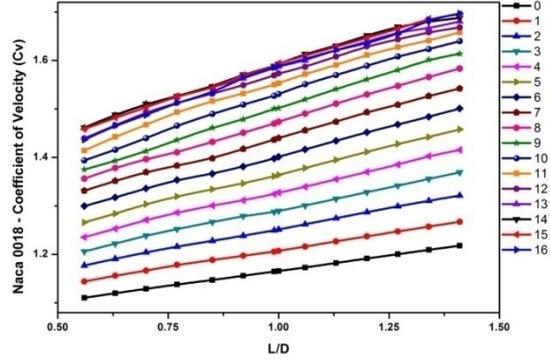


Fig. 9: Coefficient of velocity Vs L/D ratio

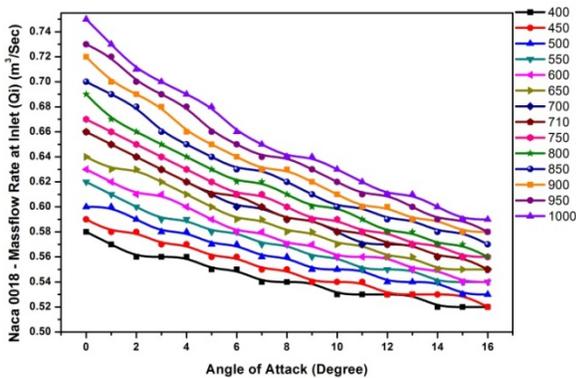


Fig. 10: Mass flow rate at inlet Vs angle of attack

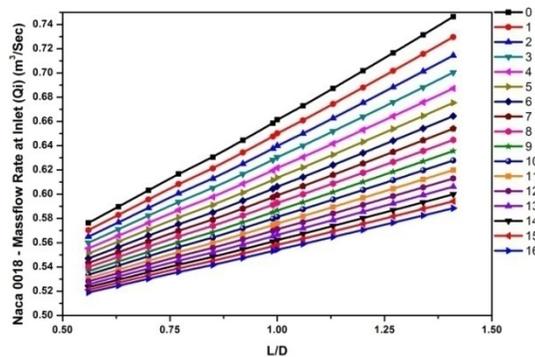


Fig. 11: Mass flow rate at inlet Vs L/D ratio

degree maximum. Figure 7 shows the maximum velocity at throat line against Length to Diameter ratio (L/D). There is a gradual increase in maximum velocity from 1.5 to 2.9 m/sec. It can be seen that the curve of maximum velocity is almost linear. Maximum velocity increases with the increase in length and angle of attack. In Fig. 8, Coefficient of Velocity (CV) against angle of attack is shown. Figure 9 shows the coefficient

of velocity against L/D ratio. Coefficient of velocity increase till 14° and after that drops a little. Overall the coefficient of velocity increases with the increase in length and angle of attack. Mass flow rate at inlet (Qi) against angle of attack is shown in Fig. 10. Similarly mass flow rate at inlet (Qi) against L/D ratio is shown in Fig. 11. Mass flow rate at inlet decrease with the increase in angle of attach as shown in Fig. 10.

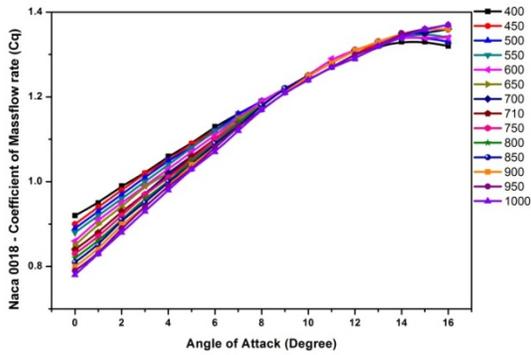


Fig. 12: Coefficient of mass flow rate Vs angle of attack

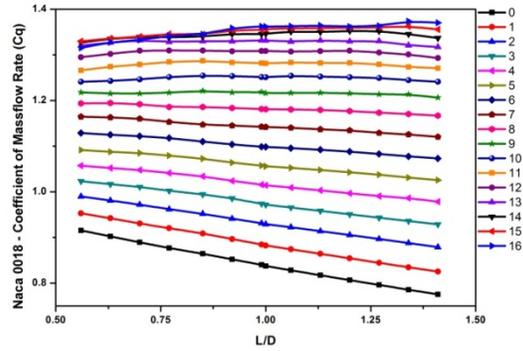


Fig. 13: Coefficient of mass flow rate Vs L/D ratio

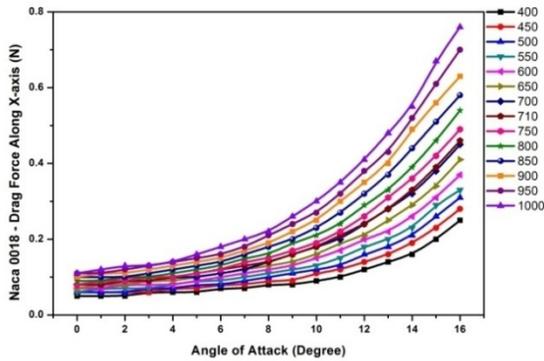


Fig. 14: Drag force along X-axis (FX) Vs angle of attack

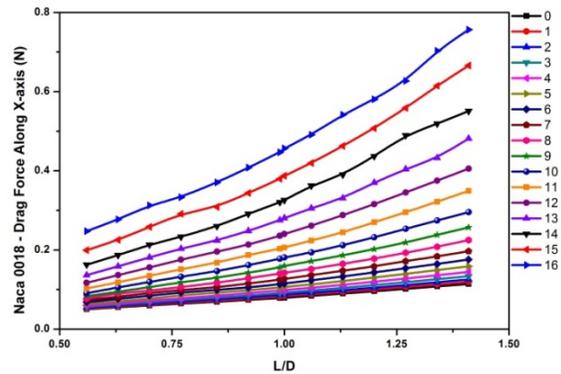


Fig. 15: Drag force along X-axis (FX) Vs L/D ratio

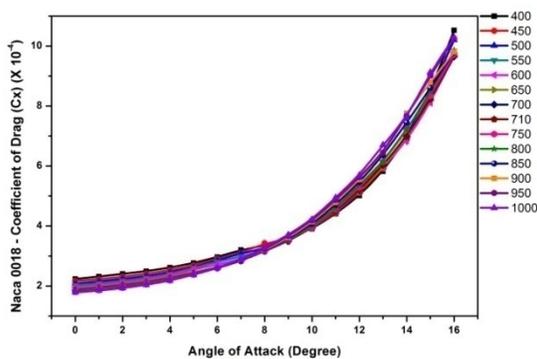


Fig. 16: Coefficient of drag Vs angle of attack

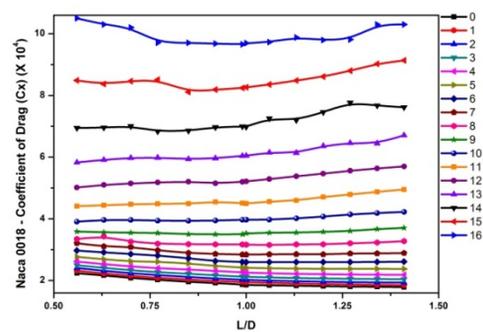


Fig. 17: Coefficient of drag Vs L/D ratio

Mass flow rate decrease as the inlet diameter of the diffuser is reduced due to the change in angle of attack from the leading edge. Figure 11 shows the mass flow rate at the throat which increases with the increase in L/D ratio. Coefficient of mass flow rate (CQ) against angle of attack and against L/D ratio is shown in Fig. 12 and 13, respectively.

Drag force (Fx) along the axis of fluid flow (i.e., X-Axis) against angle of attack and against L/D ratio is shown in Fig. 14 and 15, respectively. Drag force increase with the increase in angle of attack and L/D ratio. Similarly, Coefficient of drag (Cx) against angle of attack and L/D ratio is shown in Fig. 16 and 17, respectively. Velocity stream lines for NACA 0018 at

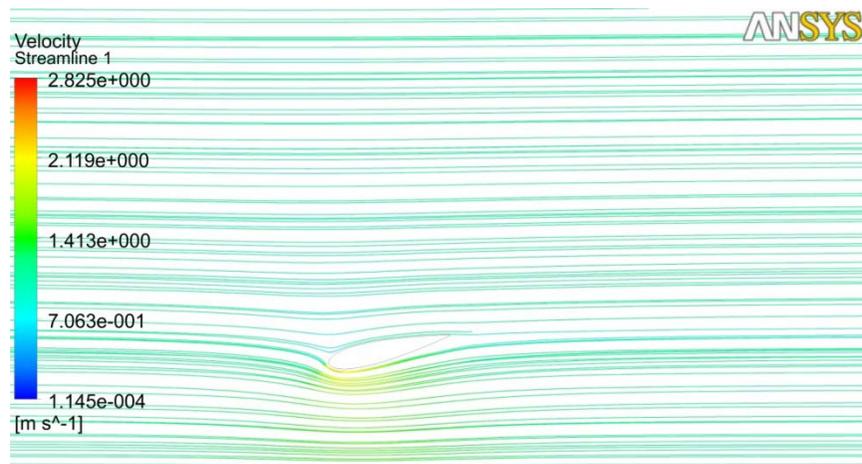


Fig. 18: Velocity streamline for NACA 0018 (Length = 400 mm, Angle of attack = 13°)

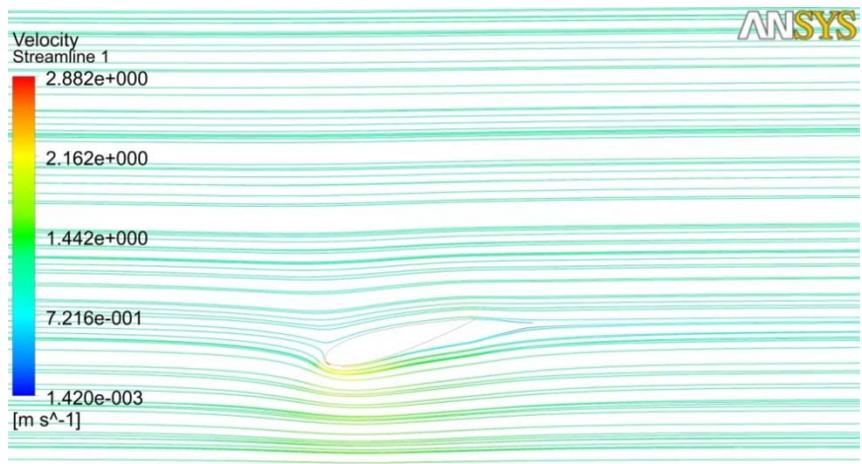


Fig. 19: Velocity streamline for NACA 0018 (Length = 500 mm, Angle of attack = 15°)

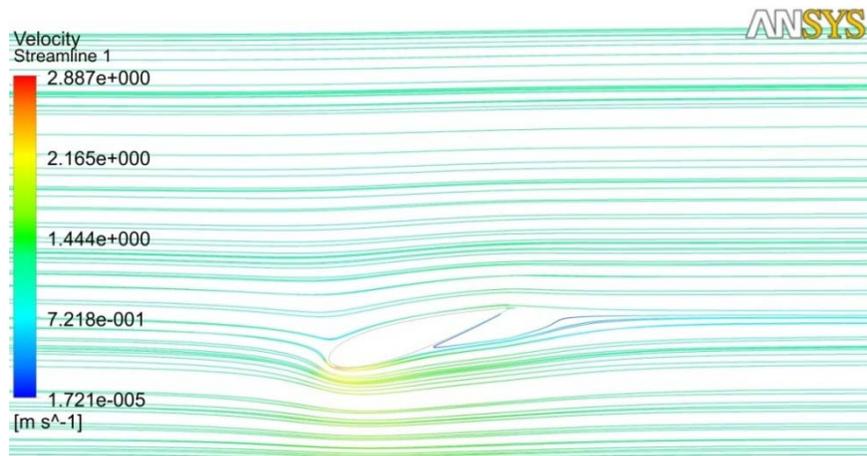


Fig. 20: Velocity streamline for NACA 0018 (Length = 600 mm, Angle of attack = 16°)

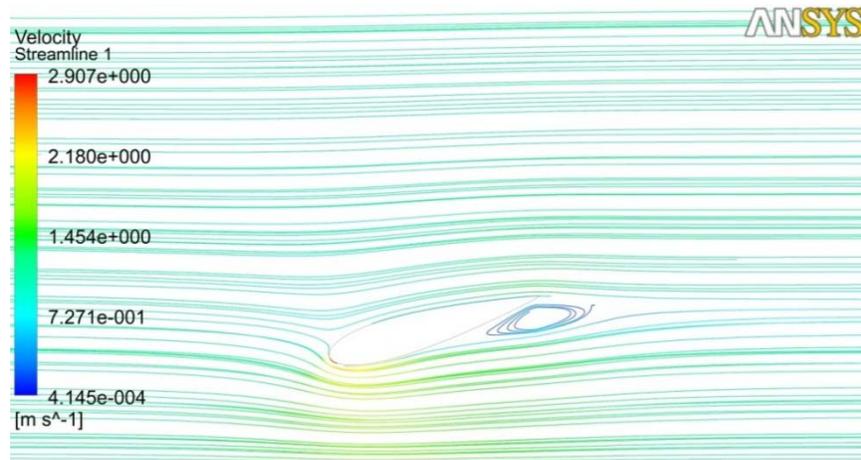


Fig. 21: Velocity streamline for NACA 0018 (Length = 700 mm, Angle of attack = 16°)

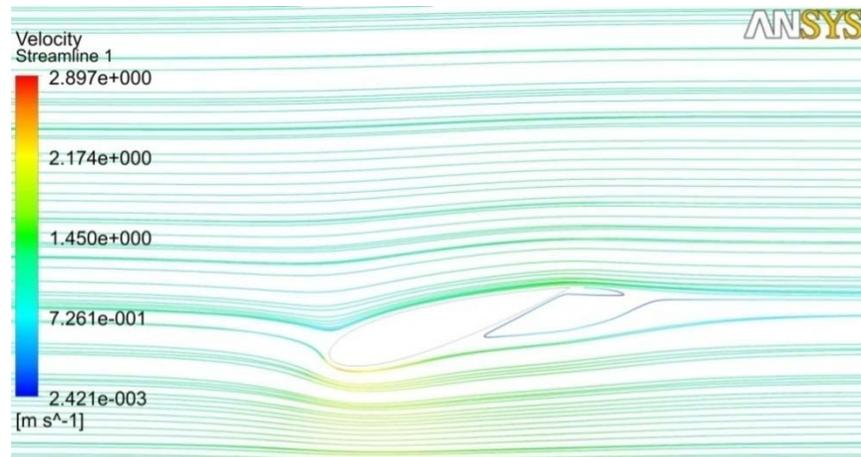


Fig. 22: Velocity streamline for NACA 0018 (Length = 800 mm, Angle of attack = 16°)

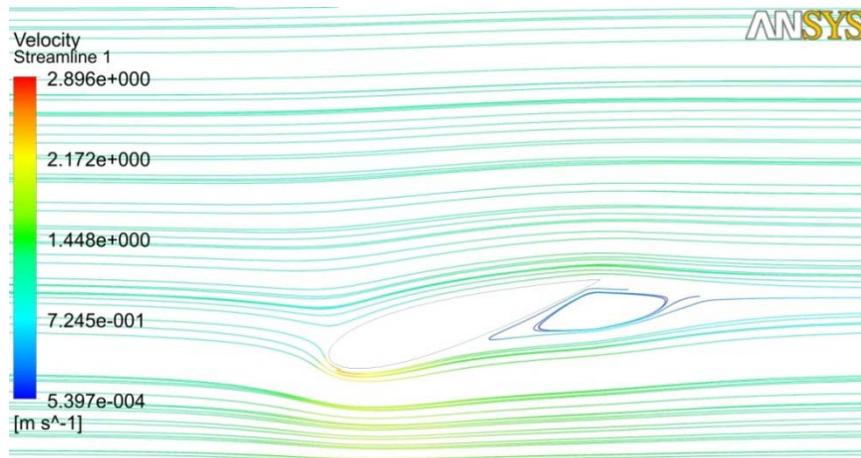


Fig. 23: Velocity streamline for NACA 0018 (Length = 900 mm, Angle of attack = 16°)

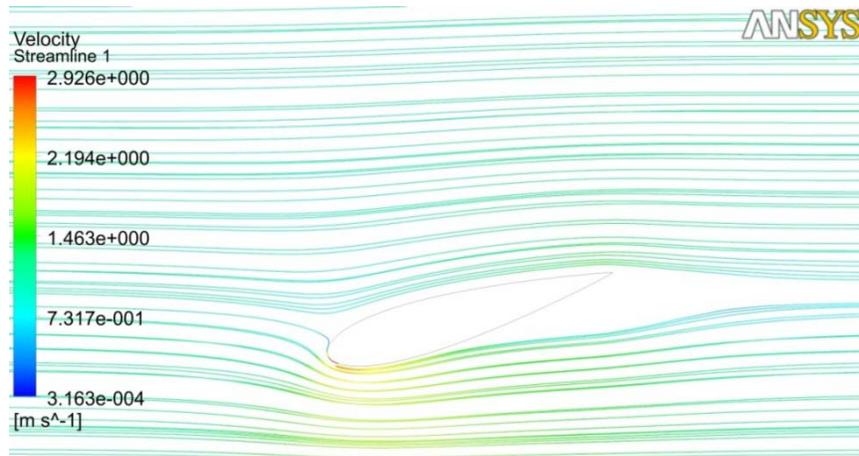


Fig. 24: Velocity streamline for NACA 0018 (Length = 950 mm, Angle of attack = 16°)

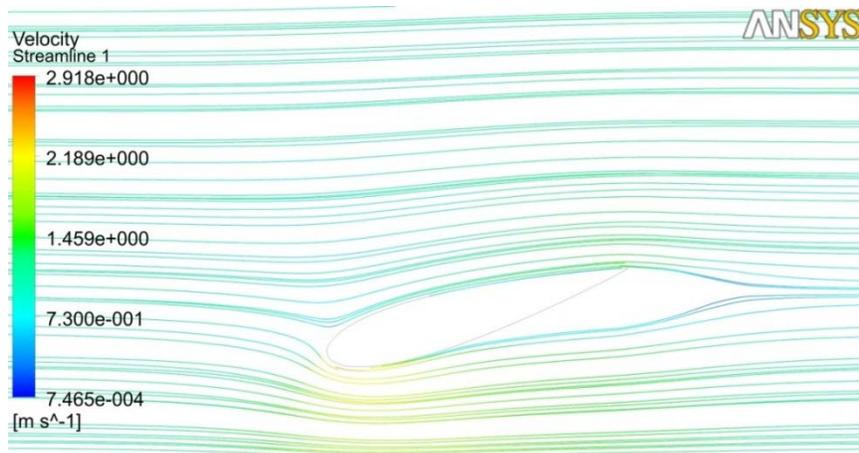


Fig. 25: Velocity streamline for NACA 0018 (Length = 1000 mm, Angle of attack = 16°)

different lengths and angle of attack are shown in Fig. 18 to 25.

### CONCLUSION

NACA 0018 airfoil has been investigated at different lengths and angles of attack to explore its potential for designing a diffuser. The main conclusions are:

- Velocity increases as length and angle of attack are increased.
- It has been observed that inlet velocity has increased from 1.5 to 2.9 m/s which are around 141.6% increase in velocity.
- Average velocity drops after 14° angle of attack.
- The mass flow rate is a function of cross-sectional

area and velocity; it also drops after 14° angle of attack.

- Drag is a very important parameter while considering commercial projects since it governs the required support structure. Drag increases with length and angle. There is always a compromise between maximum velocity and drag.

### ACKNOWLEDGMENT

This research is financially supported by National Special Foundation for Ocean Commonweal (Grant No. 200805040), S & T Program (Grant No. 2008BAA15B06), and Ocean Renewable Energy (Grants No. GHME2010GC02 and GHME2010GC03), '111 Project', Foundation from State Administration of Foreign Experts Affairs in China and Ministry of Education of China (Grant No. B07019).

**REFERENCES**

- Gilbert, B.L., R.A. Oman and K.M. Foreman, 1978. Fluid dynamics of diffuser-augmented wind turbines. *J. Energy.*, 2: 368-374.
- Gilbert, B.L. and K.M. Foreman, 1983. Experiments with a diffuser-augmented model wind turbine. *J. Energy. Res. Tech.*, 105: 46-53.
- Igra, O., 1981. Research and development for shrouded wind turbines. *Energy Conversion Manage.*, 21: 13-48.
- Mehmood, N., S. Qihu, W. Xiaohang and Z. Liang, 2011. Tidal current turbines. *Proc. Int. Conf. Mech. Electr. Technol.* 3: 445-450.
- Mehmood, N., Z. Liang and J. Khan, 2012. Harnessing ocean energy by tidal current technologies. *Res. J. Appl. Sci. Eng. Technol.*, 4(18): 3476-3487.
- Mehmood, N., Z. Liang and J. Khan, 2012. Diffuser augmented horizontal axis tidal current turbines. *Res. J. Appl. Sci. Eng. Technol.*, 4(18): 3522-3532.
- Mehmood, N., Z. Liang and J. Khan, 2012. Numerical simulation of 2D model of diffuser for tidal turbines. *Appl. Mech. Mater.*, 20: 20-20.
- Mehmood, N., Z. Liang and J. Khan, 2012. CFD study of 2D model of diffuser for harnessing tidal energy. *Adv. Mater. Res.*, 482-484: 2270-2274.