Turbulence Model Prediction Capability in 2D Airfoil of NREL Wind Turbine Blade at Stall and Post Stall Region

Shrabanti Roy, Ziaul Huque, Kyoungsoo Lee, and Raghava Kommalapati

***Abstract*—Aerodynamic force that generated on 2D section of a blade is important for measuring the blade performance. Therefore in this current work Computational Fluid Dynamics (CFD) analysis was performed on 2D S809 airfoil. S809 airfoil was designed by National Renewable Energy Laboratory (NREL). Experimental analysis of this airfoil was done and available for the validation purpose. Aerodynamic forces like lift and drag coefficients were measured by using CFD in this work. Pressure coefficients around the airfoil were also generated to compare with experimental results. A wide range of angle of attack cases with a fixed Reynolds number of 2×106 were considered which helped to analyze all stall and post stall flow conditions. It is clear that capturing all practical phenomena of 2D airfoil through CFD simulations are difficult. Over predictions of lift-coefficient and under-prediction of drag coefficient from the simulations as compared to experimental data were observed. Five different model equations were used to find the accuracy of various turbulence models in CFD calculation. The main emphasis of the result was on the variation at stall and post stall region. It has found that SST gamma-theta model is more accurate in predicting the effect of flow transition and separation than the other equations used in this work.**

***Index Terms*—Wind energy, wind blade, S809, airfoil, *k-ε*, *k-ω*, SST.**

I. Introduction

Wind has been considered as a source of energy for more than 100 years. Wind turbine is a device that helps to extract the wind energy in an environment friendly way. One of the important components of a wind turbine is the blades. Considerable amount of research has been performed on the performance of the blade. Blade performance highly depends on the sectional aerodynamic force distribution. The clear understanding of a blade section and its effect under various wind speed cases is important in calculating the efficiency. Therefore, this work focuses on aerodynamic characteristics of 2D S809 airfoil as shown in Fig. 1 which is used for National Renewable Energy Laboratory (NREL) Phase VI blade [1], [2]. Lift and drag force coefficients along with pressure distribution were calculated under various wind speed cases. Several research works were conducted on blade airfoil by using CFD [3]-[10]. These works tried to capture the effect of 2D S809 aerofoil by using several CFD codes. The effect of stall and post stall on a 2D airfoil is not clearly described and captured in most of the works. Walter & Stuart [9] in their work used S809 airfoil and performed CFD simulation on it. They varied the angle of attack from zero to 20 degree. The lift and drag coefficients were generated. With the increase of angle of attack, the simulation results of lift coefficient failed to agree well with experimental results. Angles more than 20 degree were not taken into consideration in their work. Guerri, Bouhadef and Harhad [10] also used the same airfoil. They analyzed turbulent flow simulation of the airfoil using CFD code. Their range also varied from 0 and 20 degree angles of attack. Both research used SST k-ε and RNG *k-ω* models for calculation. These model equations are good in predicting the turbulent flow condition but sometimes they over predicts the effect of turbulence under turbulent and transition condition. In the current research, CFD simulations were done on S809 airfoil. A wide range of angles of attack were considered. Several models were implemented in calculating the turbulence and separation/transition effect of S809 airfoil. Reynolds number was taken as 2×10^6 which corresponds to wind velocity of 27.4 m/s. The simulation was performed with Ansys CFX solver. Lift and drag coefficients along with Cp distribution were generated. The results of five different models are compared with NREL S809 airfoil experimental result to see the performance and the accuracy of different models. The main concentrations were in stall, separation, and post stall regions. The results helped to predict the best model for CFD simulation. The separation and transitional effect of 2D airfoil were also provided. Various models were also considered to look at the accurate prediction of the flow behavior. The results of different models were compared with NREL experimental results to find the accuracy of the CFD simulation.

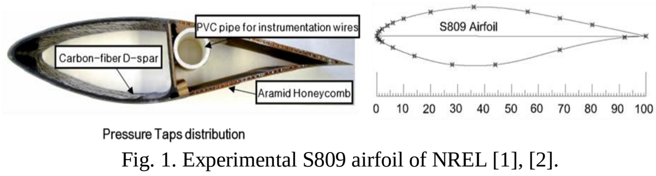

Fig. 1. Experimental S809 airfoil of NREL [1], [2].

II. Methodology

The coordinates of S809 airfoil were collected from NREL website and imported in the design modeler of Ansys to draw the shape of the airfoil. The computational fluid domain is 3m x 4m with an additional 2m radius semicircular section at the inlet as shown in Fig. 2. Fine unstructured grids were generated, keeping the minimum value of the mesh as 0.003m. Around 0.2 million nodes were generated. An

Manusript received October 6, 2016; revised February 16, 2017. This research was supported by the National Science Foundation (NSF) through the Center for Energy and Environmental Sustainability (CEES), a CREST Center (Award No. 1036593).

Shrabanti Roy, Ziaul Huque, Kyoungsoo Lee, and Raghava Kommalapati are with the Prairie View A & M University, Prairie View, TX 77446 USA (e-mail: shroy@pvamu.edu; zihuque@pvamu.edu; lks1838@naver.com; rrkommalapati@pvamu.edu).

doi: 10.18178/jocet.2017.5.6.423

inflation tool was used to satisfy the near wall Y+ value of less than one. Fig. 3 shows the near wall grids around the airfoil.

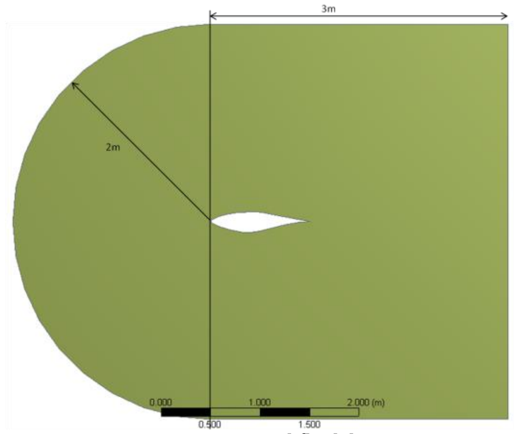

Fig. 2. Computational fluid domain.

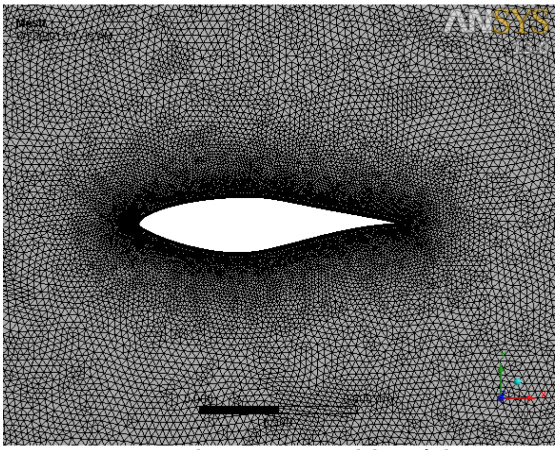

Fig. 3. Grid generation around the airfoil.

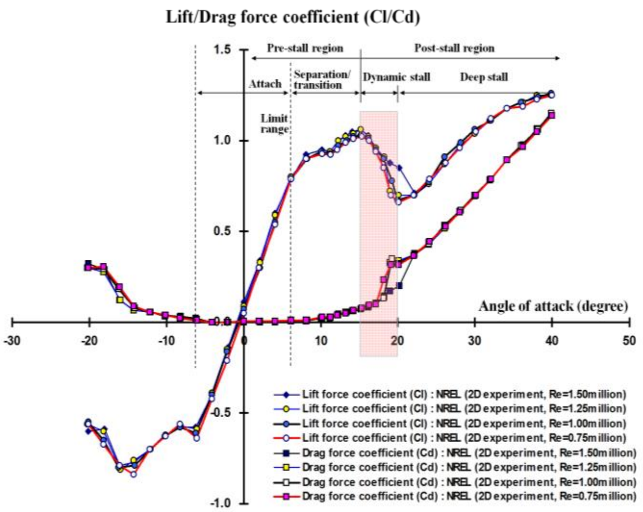

Fig. 4. Experimental lift and drag co-efficient result of S809 airfoil and the definition of stall at different angle of attack [1], [2].

Inlet velocities are defined by the velocity components to create the effect of varying AoA. Flow simulations were performed for AoA varying from -2.1° to 34°. Reynolds number of 2×10^6 was used and was defined as the inlet wind speed for all the cases. The semicircular boundary was defined as the inlet and its opposite side as the outlet. The various turbulence models used are *k-ε*, *k-ω*, SST, SST turbulence and SST gamma theta for the simulations.

III. Turbulence Models

A. k-ε Model

The standard *k-ε* model is a semi-empirical model based on model transport equations for the turbulent kinetic energy, *k*, and its dissipation rate, *ε*. ε-equation is only solved in the outer part of the boundary layer, whereas the inner portion of the logarithmic layer and the viscous sub layers are treated by a mixing length formulation [11].

B. k-ω Models

An alternative to ε equation is the ω equation in the form developed by Wlcox (1993). Instead of the equation for the turbulent dissipation rate, *ε*, an equation for the turbulent frequency, *ω*, of the large scales is used. The ω-equation has significant advantages near the surface and accurately predicts the turbulent length scale in adverse pressure gradient flows, leading to improved wall shear stress and heat transfer predictions. One of the main advantages of the *k-ω* model is its robustness even for complex applications, and the reduced resolution demands for integration to the wall. It was shown by Menter [11] that the main deficiency of the standard *k-ω* model is the strong sensitivity of the solution to free stream values for ω outside the boundary layer.

C. SST, SST Turbulence and SST Gamma Theta Models

In order to overcome the problem related to k-ε and k-ω models, a combination of the effects of near wall and away from the wall has been proposed which is named as shear stress transport (SST) model. The SST also has the capabilities of solving the near wall separation effect. Although this model predicts both near-wall and larger-scale boundary effects, it is inaccurate for the viscous-sub-layer formulation and transitional flow and it sometimes also bypasses the transitional effect. This transitional prediction has two modelling concepts. The first is the use of low-Reynolds number turbulence models, where the wall damping functions of the underlying turbulence model trigger the transition onset. This concept is attractive, as it is based on transport equations and can therefore be implemented without much effort. However this concept fails to predict various flow effects like transition, flow stream turbulence and separation.

In order to correct the problem another approach is developed which correlates the turbulence intensity, τ, in the free-stream to the momentum-thickness Reynolds number, Re, at transition onset. The full model is based on two transport equations, one for the intermittency and one for the transition onset criteria in terms of momentum thickness Reynolds number. It is called the 'Gamma Theta Model' and is the recommended transition model for general-purpose applications. It uses a new empirical correlation [11]-[13] that has been developed to cover standard bypass transition as well as flows in low free-stream turbulence environments.

IV. Results

Experimental result of lift and drag coefficients of 2D NREL S809 airfoil is shown in Fig. 4 [1], [2]. Results of four different Reynolds numbers are reported. The figure also shows 2D stall definition of the airfoil at various AOA. It is found that up to 9 degree AoA, lift coefficient (C_L) increases linearly. This is the attached flow region. After that it starts to deviate. The highest value of C_L is at 15 degree AoA which is considered as stall angle. From 15 degree to higher angles of attack is considered as post stall region. Flow gets completely separated when AoA becomes 20 degree which is considered as the onset of separation. Since the flow gets completely separated at 20 degree AoA and higher it is called the deep stall region. Region between 15 to 20 degree, where the flow starts to separate but not completely separated is the dynamic stall region. This stall definition is helpful for further explanation of results.

Different turbulence equations have been used to calculate the lift and drag coefficients (C_L &C_D) at different AoA of airfoil. In Fig. 5, C_L of the different models are shown and compared with NREL S809 airfoil experimental result. C_L of all models follows almost the similar trend as the experimental data but is over predicted. However the deviation of result is high after 7 degree AoA. At 15 degree AoA all models show the highest value of C_L because of the stall effect. After 15 degree AoA C_L starts to drop. The variation of C_L is found to be relatively higher for k-ω and k-ε models. Results from SST model also deviates from experimental result but the deviation is lower than k-ε and k-ω models. There is fluctuation effect in SST, SST turbulence and SST gamma theta equation from 15 degree to 30 degree AoA. Compared to the three different SST based models, SST gamma theta model agrees better with NREL experimental results in dynamic stall region. In dynamic stall region the effect is almost similar with each other for all other models which deviate highly from experimental results. The predicted C_L values become closer to experimental value after 30 degree angle of attack.

Except k-ε, all other model equations show fluctuation after certain AoA of dynamic stall region. The value of C_L in those cases considered as the average of fluctuating values.

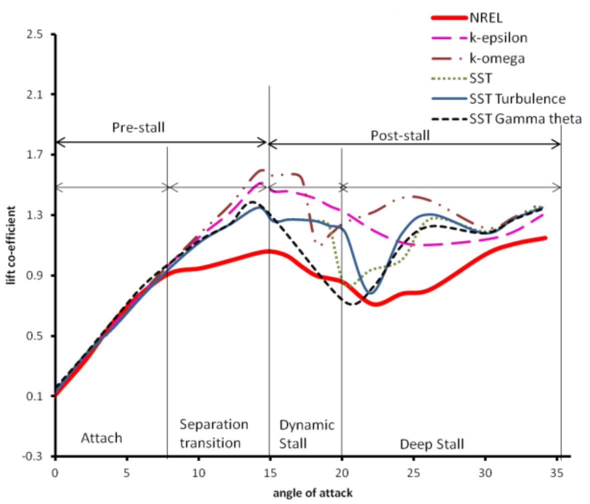

Fig. 5. Lift co-efficient result of current simulation by using different model equations and its comparison with NREL results.

Comparison of C_D of different model equation at different AOA is shown in Fig. 6. It shows compared to C_L of different model equations C_D has less variation of result from experimental values. The deviation starts mainly after around 17 degree AoA when high fluctuation takes place. C_D from SST gamma theta model shows the best comparison compared with experimental data than other models. *k-ε* shows the highest deviation of C_D with other results. *k-ω* under predicts the experimental values. This high deviation is because the two models cannot capture effect of separation accurately.

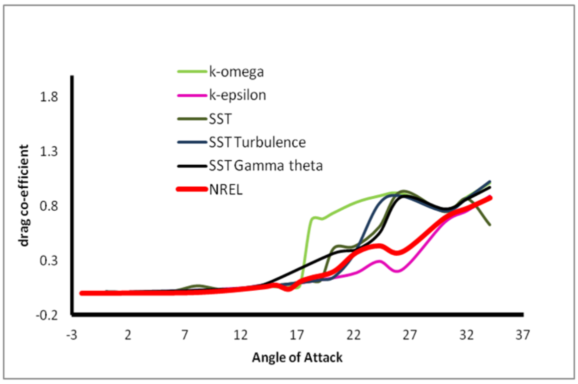

Fig. 6. Drag co-efficient result of current simulation by using different model equations and its comparison with NREL results.

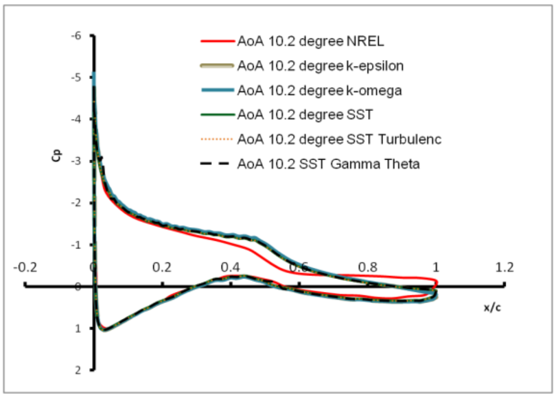

Fig. 7. Comparison of Cp distribution of different model equations at 10 degree angle of attack.

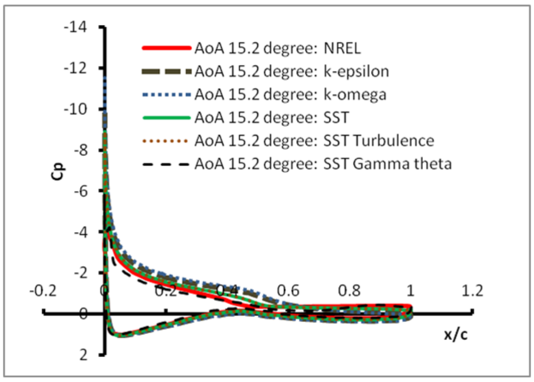

Fig. 8. Comparison of Cp distribution of different model equations at 15.2 degree angle of attack.

C_L is almost similar for all model equations up to 7 degree AoA. This is the attached flow region. C_L over predicts the NREL results for larger than 7 degree AoA. This deviation is high for *k-ε* and *k-ω* models. All the SST based models predicts almost similar C_L from 9 to 15 degree AoA.

C_L & C_D of a 2D aerofoil is the result of pressure co-efficient (*Cp*) distribution around it. *Cp* distributions of a 2D airfoil are shown in Fig. 7 to Fig. 10. It is found that at 10 degree AoA, *Cp* of all models have almost similar value as shown in Fig. 7. However all of them predict higher value of *Cp* than experimental values. This is the reason why C_L is also high for all models. In Fig. 8, Cp distribution at stall

AoA of 15 degree is shown. At this AoA pressure co-efficient distribution around the airfoil of all model equations differs from experimental value. *k*-ε and *k*-ω predicts higher value than any other cases at the leading edge. Compared to that, SST and SST turbulence models predict better *Cp* values. C_L of the *k*-ε and *k*-ω is also higher due to the same reason.

With the increase of AoA flow separation increases in dynamic stall region. *Cp* distribution of 20 degree AoA is shown in Fig. 9 which is under dynamic stall region. All the models show high value of *Cp* at the leading edge except SST and SST gamma theta model. SST gamma theta shows lower prediction at this point. There is a high fluctuation with every model. SST, SST turbulence and SST Gamma theta models are capable in predicting the transition and turbulent effect accurately. That is why it shows the fluctuation. *k*-ε fails to predict near wall effects and cannot capture flow turbulence.

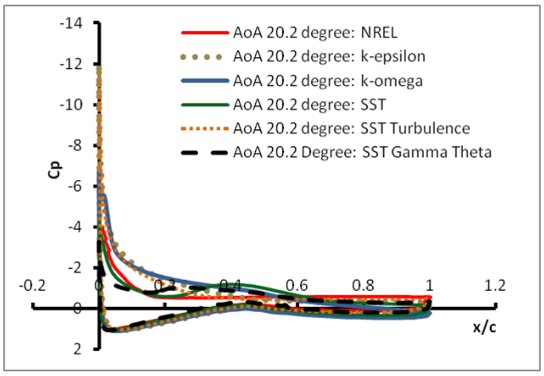

Fig. 9. Comparison of Cp distribution of different model equations at 20.2 degree angle of attack.

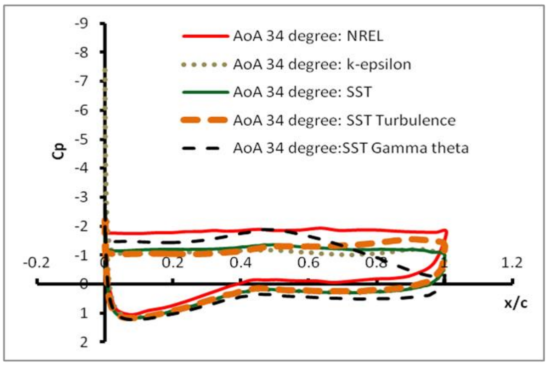

Fig. 10. Comparison of Cp distribution of different model equations at 34 degree angle of attack.

Deep stall region starts after 20 degree AoA. Every model predicts higher Cp at the leading edge as described in Fig. 10 which shows Cp at 34 degree AOA. That is why C_L are found to be higher than experimental value. There is high fluctuating effect still in SST gamma theta model equation. This is because SST gamma theta predicts the effect of transition and turbulence more accurately. It can also predict the effect near airfoil surface when flow gets separated.

V. Conclusion

This work presents CFD calculation of S809 airfoil under various angle of attack conditions. The main purpose is to see the effect of stall and post stall of 2D airfoil on aerodynamic forces like lift and drag co-efficient. It compares the result of five different turbulence models in predicting the transition and separation flow condition. The results were validated by comparing with NREL experimental results.

High deviation of lift and drag coefficient results were observed in stall and post stall region. The deviation starts mainly after 9 degree AoA when flow transition starts and it become higher at stall angle of 15 degree. It has been found that predicting the effect of high AoA of a 2D airfoil by using CFD is a challenging area.

While comparing different turbulence models, the highest deviation is found at *k*-ε and *k*-ω model. Though SST and SST turbulence models are better compared to previous two, but they have a tendency to over predict the turbulence effect since they neglect the effect of transition.

Comparing all the turbulence models the prediction capability of SST Gamma theta equation at higher AoA is better because of its capability in computing the effect of onset of stall and flow transition. Therefore, this SST Gamma theta model is highly recommended in calculating the airfoil aerodynamics for future works.

References

[1] D. M. Sommers, "Design and experimental results for the S809 Airfoil," Tech. rep., *National Renewable Energy Laboratory*, 1997.

[2] R. R. Ramsay, J. M. Janiszewska, and G. M. Gregorek, "Wind tunnel testing of three S809 aileron configurations for use on horizontal axis wind turbines," Tech. rep, *National Renewable Energy Laboratory (NREL)*, 1996.

[3] W. A. Timmer, "Aerodynamic characteristics of wind turbine blade airfoils at high angles of attack," Delf University of Technology, TORQUE 2010: The Science of Making Torque from Wind, June 28-30, Crete, Greece.

[4] G. Kobra and A. J. David, "Numerical modeling of an S809 airfoil under dynamic stall, erosion and high reduced frequencies," *Applied Energy Journal*, vol. 93, pp. 45-52, 2011.

[5] H. Yilei and A. Ramesh, "Shape Optimization of NREL S809 Airfoil for Wind Turbine Blade Using Multiobjective Genetic Algorithm," *International Journal of Aerospace Engineering*, vol. 2014, pp. 13.

[6] G. Sandeep and L. Gordo, "Dynamic Stall Modeling Of The S809 Aerofoil And Comparison With Experiment," *Wind Energy Journal*, vol. 9, pp. 521-547, 2006.

[7] A. O. Gomesa, R. F. Britob, H. M. P. Rosaa, J. C. C. Camposa, A. M. B. Tibiriçaa, and P. C., "Treto, Experimental Analysis Of An S809 Airfoil," *Thermal Engineering*, vol. 13, no. 2, pp. 28-32, December 2014,.

[8] D. Eleni and M. Dionissios, "Aerodynamic Characteristics Of S809 Vs. Naca 0012 Airfoil For Wind Turbine Applications," 5 th International Conference from Scientific Computing to Computational Engineering, 5 th IC-SCCEAthens, 4-7 July, 2012.

[9] P. W. Walter and S. O. Sturart, "CFD Calculation Of S809 Aerodynamic Characteristics," AIAA Meeting Papers, (January 1997).

[10] G. Ouahiba, B. Khadidja, and H. Ameziane, "Turbulent Flow Simulation Of The NREL S809 Airfoil," *Wind Engineering*, vol. 30, no. 4, pp. 287-302, 2006.

[11] F. R. Menter, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *AIAA-Journal*, vol. 32, no. 8, pp. 269-289, 1994.

[12] F. R. Menter, "Review of the Shear-Stress Transport Turbulence Model Experience from an Industrial Perspective," *International Journal of Computational Fluid Dynamics*, vol. 23, no. 4, pp. 305-316, 2009.

[13] F. R. Menter, R. Langtry, and S. Volker, "Transition Modeling for General Purpose CFD Codes," *Flow Turbulence Combust*, vol. 77, pp. 277-303, 2006.

Shrabanti Roy received her B.SS degree in Mechanical engineering from Military Institute of Science & Technology, Bangladesh, 2011. She is currently a Masters student at Prairie View A & M University, Texas and also works under Center for Energy & Environmental Sustainability (CEES). Her major concentration is on Mechanical engineering and interest area is CFD, Thermo fluid science and wind energy.

Ziaul Huque received his BS degree in mechanical engineering from Bangladesh University of Engineering and Technology, Bangladesh, MS in mechanical engineering from Clemson University, USA and Ph.D. degree in mechanical engineering from Oregon State University, USA. He is currently a professor in the department of mechanical engineering and the director of Computational Fluid Dynamics Institute at Prairie View A&M University. Professor Huque published over 50 journal and conference papers. His current research interests are wind turbine noise reduction, fluid-structure interaction, propulsion, inlet-ejector system of rocket based combined cycle engines, clean coal technology, self-propagating high-temperature synthesis. He received several excellence in teaching and service awards from Roy G. Perry College of Engineering, Lockheed-Martin Tactical Aircraft Systems Teaching Excellence Award, Welliver Summer Faculty Fellowship from Boeing in in 2002 and NASA Summer Faculty Fellowship in 2003.

Kyoungsoo Lee received his BS, MS, Ph.D in Depart. of Architectural Engineering from Inha University, Incheon, South Korea. He was working for the CEES, Prairie View A&M University, Prairie View, Texas, USA as a post doc. Researcher. He was a research professor in department of Civil & Environmental Engineering, KAIST in South Korea. His professional areas are the structural engineering and design, CFD, FSI and Impact & Blast simulation. Currently, he is focusing on the developing the sound noise simulation for the wind blade. Dr. Lee is the member of AIK, KSSC in South Korea. Currently he is at Samsung Electronics in Samoo Architect & Engineers.

Raghava Kommalapati received MS and PhD degrees in Civil (Environmental Engineering) from Louisiana State University, Baton rouge, LA, USA in 1994 and 1995 respectively. Prior to that he received his B. Tech in Civil engineering and M. Tech in Engineering Structures from India. His major field of study is environmental engineering with particular focus on energy and environmental sustainability and air quality. He is a Director of Center for Energy and Environmental Sustainability and Professor of Civil Environmental Engineering at Prairie A & M University System. He is editor of 1 book, and published more than 30 peer reviewed journal articles and more than 70 proceedings and presentations at regional, national and international conferences. Dr. Kommalapati's research interest include remediation of contaminated soils, industrial waste separations, air quality atmospheric air-fog interactions and environmental impacts of energy technologies.

Dr. Kommalapati is a licensed Professional Engineer (PE) in the state of Texas and Board Certified Environmental Engineer (BCEE). He is a member of several professional organizations including, AEESP, AAEES, ACS, ASCE, ASEE and Honor societies of Tau Beta Phi, Phi Kappa Phi and Sigma Xi. He is an editorial board member of Journal of Studies in Atmospheric Science (SIAS), Current Advances in Environmental Science.