"FLUID STRUCTURE INTERACTION OF WIND TURBINE BLADE USING COMPUTATIONAL FLUID DYANAMICS"

WAGH SUNIL RAJARAM[1], S.U.PATEL[2], D M.PATIL[3]

[1]*Student, Department of Mechanical Engineering, P.S.G.V.P. Mandal'sD.N.Patel College Of Engineering Shahada, North Maharashtra University, Jalgaon, Maharashtra, India*

[2]*Assistant Professor, Department of Mechanical Engineering, P.S.G.V.P. Mandal's D.N.Patel College Of Engineering Shahada, North Maharashtra University, Jalgaon, Maharashtra, India*

[3]*Assistant Professor, Department of Mechanical Engineering, P.S.G.V.P. Mandal'sD.N.Patel College Of Engineering Shahada, North Maharashtra University, Jalgaon, Maharashtra,*

ABSTRACT

In today's rapid increase in energy demands and decaying fossil fuels availability, it is very important to investigate the different alternatives, renewable sources for energy generation. Wind energy is the biggest available source of energy which can easily be harnessed using the mechanical turbines; by converting wind energy into mechanical energy. A wind turbine is a rotary device that extracts kinetic energy from the wind. It has rotor blades mounted on the shaft which is connected to the generator through transmission devices. The blade profiles play an important role which decides the performance of the wind turbine.

In this work, two blade profiles are simulated using CFD modeling and results are validated for coefficient of drag and lift. Further, the two profiles are compared for the flow separation point for different angle of attack. Optimization is performed between the two geometries for two different Re numbers using k-epsilon model for minimum drag. Coefficient of performance and critical angle of attack are also discussed for the two blade profiles.

The pressure coming over the blade is used as input forces for structural analysis for calculation of displacement and structural parameters a like vomises stress and mode shapes. The study uses different shapes of blade to characterize the performance of these blades in wind turbine applications.

KEYWORDS: Wind Energy, NREL Airfoil ,CFD simulation, fluid structure interaction(FSI)

1.INTRODUCTION

1.1. BACKGROUND

Wind energy is an abundant resource in comparison with other renewable resources. Moreover, unlike the solar energy, the utilization could not be affected by the climate and weather. Wind turbine was invented by engineers in order to extract energy from the wind. Because the energy in the wind is converted to electric energy, the machine is also called wind generator. Figure 1 shows the growth rate of wind generator capacities, which has increased significantly in the last ten years. The total installed capacity of wind power generators was 159,213 MW at the end of 2009 (World Wind Energy Report 2009).

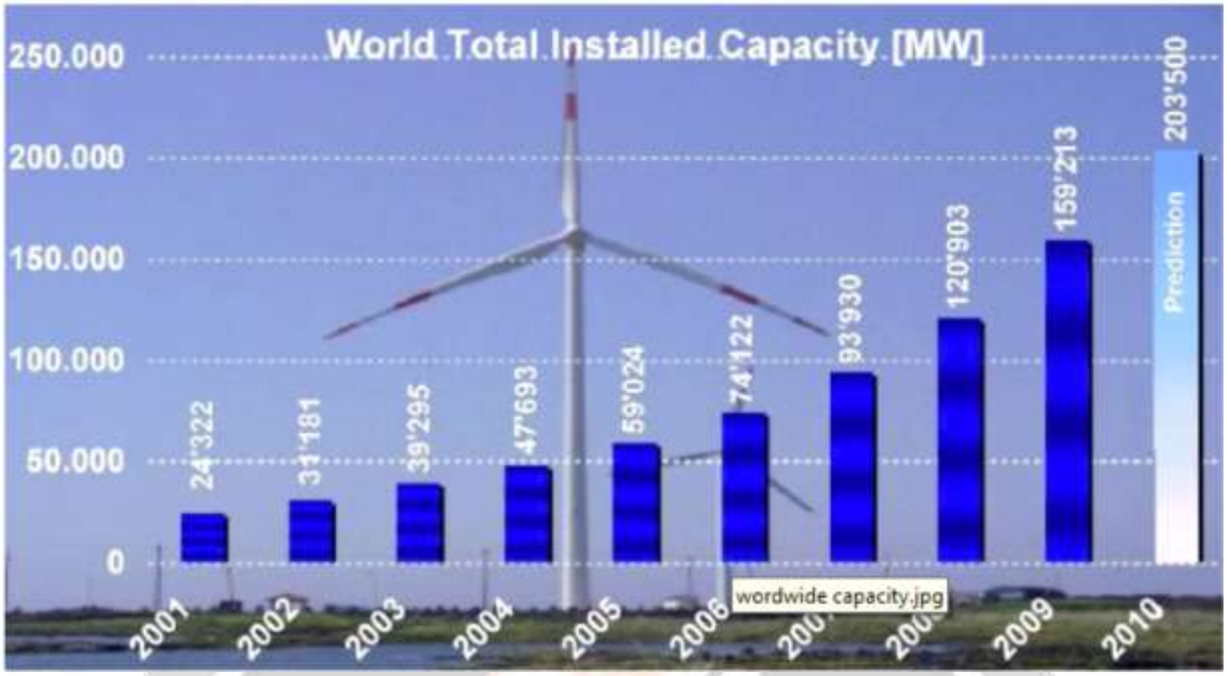


Figure.1. World total installed capacity (World Wind Energy Report 2009[1].

A wind turbine consists of several main parts, i.e. the rotor, generator, driven chain, control system and so on. The rotor is driven by the wind and rotates at predefined speed in terms of the wind speed, so that the generator can produce electric energy output under the regulation of the control system. In order to extract the maximum kinetic energy from wind, researchers put much efforts on the design of effective blade geometry. In the early stage, the airfoils of helicopters were used for wind turbine blade design, but now, many specialized airfoils have been invented and used for wind turbine blade design. Moreover, a rotor blade may have different aerofoils in different sections in order to improve the efficiency, so the modern blades are more complicated and efficient comparing to early wind turbine blades. In the early stage, the research on wind turbine blade design was limited on theoretical study, field testing and wind tunnel testing which need a lot of efforts and resources. Due to the development of computer aided design codes, they provide another way to design and analyse the wind turbine blades. Aerodynamic performance of wind turbine blades can be analysed using computational fluid dynamics (CFD), which is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems of fluid flows. Meanwhile, finite element method (FEM) can be used for the blade structure analysis. Comparing to traditional theoretical and experimental methods, numerical method saves money and time for the performance analysis and optimal design of wind turbine blades [1].

1.2.Types of Wind Turbines

There are mainly two types of wind turbine: horizontal axis and vertical axis. The horizontal axis wind turbine (HAWT) and the vertical axis wind turbine (VAWT) are classified or differentiated by the axis of rotation the rotor shafts.[2]

Horizontal Axis Wind Turbines - Horizontal axis wind turbines, also known as HAWT type turbines have a horizontal rotor shaft and an electrical generator which is both located at the top of a tower.[2]

Vertical Axis Wind Turbines - abbreviated as VAWTs, are designed with a vertical rotor shaft, generator and gearbox which are placed at the bottom of the turbine, and a uniquely shaped rotor blade that is designed to harvest the power of the wind no matter which direction is it blowing.

The first is the Darrieus wind turbine, which is designed to look like a modified egg beater. These turbines have very good efficiency, but poor reliability due to the massive amount of torque which they exert on the frame. Furthermore, they also require a small generator to get them started.[2]

Figure.2. Wind Turbines, Some types[2].

1.3. **Airfoil**

For the efficient energy extraction, blades of modern wind turbine are made with airfoil sections. Major features of such an airfoil are shown in Fig1.4The airfoils used for the earlier day's wind turbines were the aviation air foils under the NACA (National Advisory Committee for Aeronautics) series. NACA specifies the features of the airfoil by numbers. For example, in a four digit specification, the first number denotes the maximum camber of the airfoil at the chord line (in per cent of chord), the second number gives the location of the point of maximum camber from the leading edge[3].

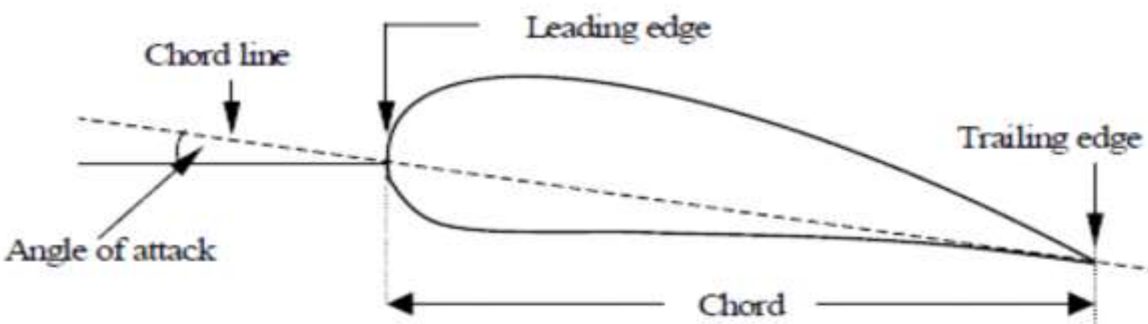


Figure.3. Important parameters of an airfoil[3].

(in tenth of the chord) and the third and fourth numbers indicate the maximum thickness (in per cent of the chord). Thus a NACA 2415 air foil have maximum camber of 2 per cent, located at 0.4 times the chord length from the leading edge and the maximum thickness is 15 per cent of the chord. The five digit specification of airfoil is similar, but information on the lift characteristics are also included. In a five-numbered NACA airfoil, the design lift coefficient in tenth is given by 1.5 times of the first digit. One-half of the distance from the leading edge to the location of maximum camber (in per cent of the chord) is represented by the second and third digits. The fourth and fifth digits are the thickness of the airfoil (in per cent of the chord). Thus a NACA 23012 series airfoil will have a design lift coefficient of 0.3.

The maximum camber of the airfoil is 0.15 times the chord and its thickness is 12 per cent. Blades with higher digit NACA numbers are also available. As far as the requirements of a wind turbine are concerned, NACA specified airfoils are poor in their stall characteristics. Moreover they are insensitive to wide variations in Reynolds number under which a wind machine is expected to work. Adding to these, they have low structural efficiency at the root region. These limitations called for the development of air foils tailored for wind turbines. Several research organizations like the Delft University of Technology[3]

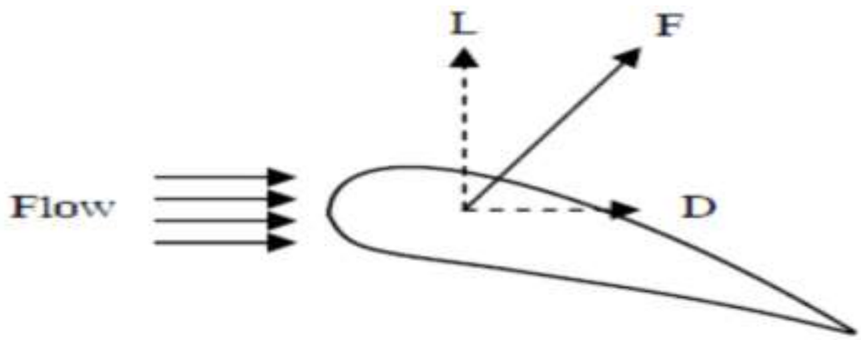


Figure. 4. Airfoil lift and drag[3].

2 .COMPUTATIONAL FLUID DYNAMICS (CFD)

There are many commercial CFD soft ware's used in engineering, such as PHOENICS (it is the first commercial CFD software), STAR-CD, ANSYS FLUENT/CFX and so on. All CFD soft wares have three main structures which are Pre-Processer, Solver and Post-Processor [1].

2.1.**The principle theories relevant to CFD modeling**

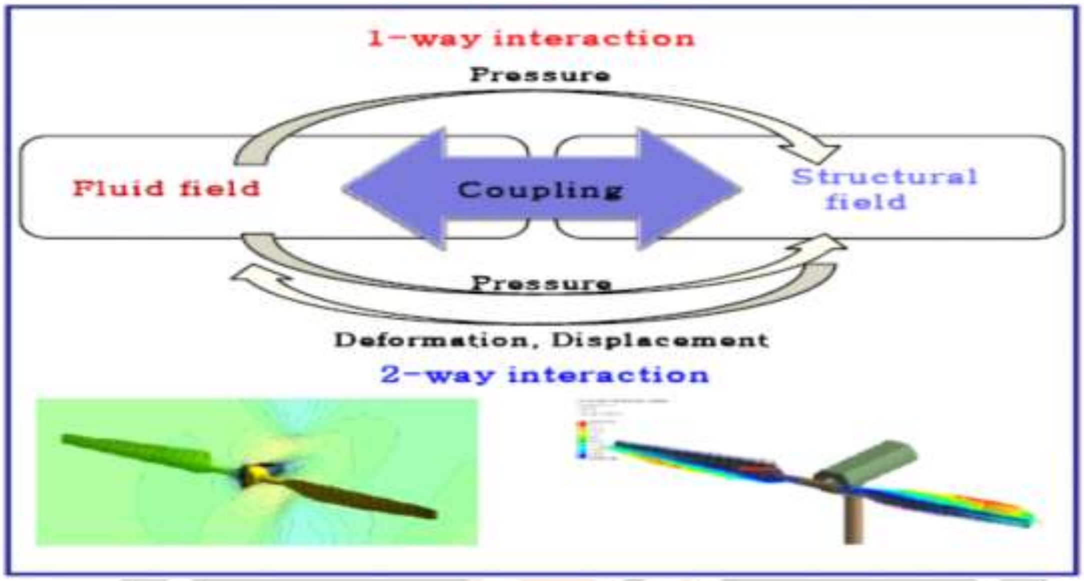
No matter what kind of CFD software is, the main processes of simulation are the same. Setting up governing equations is the precondition of CFD modeling; mass, momentum and energy conservation equation are the three basis governing equations. After that, Boundary conditions are decided as different flow conditions and a mesh is created. The purpose of meshing model is discredited equations and boundary conditions into a single grid. A cell is the basic element in structured and unstructured grid. The basic elements of two-dimensional unstructured grid are triangular and quadrilateral cell. Meanwhile, the rectangular cell is commonly used in structured grid. In three-dimensional simulation, tetrahedral and pentrahedral cells are commonly used unstructured grid and hexahedra cell is used in structured grids. The mesh quality is a prerequisite for obtaining the reasonably physical solutions and it is a function of the skill of the simulation engineer. The more nodes resident in the mesh, the greater the computational time to solve the aerodynamic problem concerned, therefore creating an efficient mesh is indispensable. Three numerical methods can be used to discretise equations which are Finite Different Method (FDM), Finite Element Method (FEM) and Finite Volume Method (FVM). FVM is widely used in CFD software such as Fluent, CFX, PHOENICS and STAR-CD, to name just a few. Compared with FDM, the advantages of the FVM and FEM are that they are easily formulated to allow for unstructured meshes and have a great flexibility so that can apply to a variety of geometries [1].

3**.Fluid structure interaction (FSI)**

Appropriate moment of inertia is given under a specified wind speed. Wind speed affects the rotational speed and torque of the blades, which can give feedback to flow field simultaneously. The results are stable when the iterations of fluid dynamics and structure dynamics converge; the In order to simulate blade rotational effect, rotational reference frame, sliding mesh and dynamic mesh technologies can be used in ANSYS-Fluent which is one of the most famous CFD commercial software. The rotational reference frame model is used in steady-state solution (FLUENT 6.3 User's Guide). For wind turbine simulation, it requires the rotational speed to remain constant. Sliding mesh model is used in unsteady-state simulation. Normally, it has been adopted to solve periodic problem such as rotation wake and flow separation, but this approach requires large memory and high performance CPU. As the dynamic mesh is based on the moving objects, it can automatically rebuild the mesh in the computational domain, hence can be used to solve the unsteady-state problem and will seize a large number of computational resources. The precondition of solving fluid structure interaction problem is using dynamic mesh. FSI is the interaction of some movable or deformable structures with an internal or surrounding fluid flow**.** It is the cutting edge of wind turbine simulation. Under the interaction of computational fluid dynamics (CFD) and computational structure dynamics (CSD), the rotor rotational speed, pressure distribution and dynamic stress distribution on the blade can be evaluated in a particular wind speed. Normally, ordinary CFD software can simulate wind turbine rotation in a steady condition which means the wind speed is constant and rotational speed needs to be set up manually, this method requires rotational speed to be adjusted with an applicable value, the wind speed and wind turbine rotational speed are both independent so that they could not affect each other. By using fluid structure interaction technology, wind turbine model starts to rotate if an simulated result can be used in an aero elastic analysis[4].

3.1.Concept of fluid structure interaction

The FSI are used in wide range of multi-physical problem in which the fluid effects are dominant but difficult to evaluate for structural part. Especially high curvature free shape of 3Dwind blade is a challenging problem to apply on the FSI simulation, because of its difficulty in evaluating the aerodynamic force of the surface.



Figiure.5. Concept of fluid structure interaction. taken from [4].

The fundamental pre-requirements for FSI are the verification of pressure loading value from CFD and the definition of interface between fluid and structural domain. As shown in Fig. the interface transfers data from fluid to structure or structure to fluid. These interactions are called as1-way. If the interface transfers the data to both sides sequentially, we call it as 2-way. Fluid domain transfers the pressure or temperature to the structure and structure transfers usually the displacement to the fluid. From these interactions, the fluid domain updates the domain mesh from structural deformation. The structural part deforms due to the pressure or temperature load from the fluid domain. Unlike the BEM or VM methods, the structural part can be modeled using FEM procedure. The response of the structure relies on the material and solid modeling quality. The surface of structural part should share identical topology with CFD domain.

The rotating wind blade is vibrating with its frequency to the direction of first Eigen-mode shape which is independent to the wind flow direction but dependent to the local axis. The vibrating frequency of wind blade is dependent on the stiffness and mass of blade. If the tower is considered, the vibrating model of tower will be coupled with blade. Lift causes bending in the flap-wise direction while air flow around the blade cause edge-wise bending. Flap-wise bending involves tension on the pressure (upwind) side and compression on the suction (downwind) side. Edge wise bending involves tension on the leading edge and compression on the trailing edge. Hence, 3D wind blade may experience the unsteady transient of fluctuation and vortex shedding. CFD is one of the options to capture these unsteady effects, even though it has numerical difficulties and limitations. To consider the time dependent unsteady effect of fluctuation and vibration in numerical method, the fluid domain must be changed along with structural vibration intransient analysis. Hence, to simulate these problems, the interface should take into account the interaction of both sides using 2-way FSI. The 1-way FSI is used for steady static problem and 2-way is for unsteady transient problem[4].

4. CFD Simulation of NREL S809 and S813 Airfoil.

4.1. Geometry Modeling

Airfoil NREL S809 And NREL S813 are two wind blade turbine are selected for the optimization. The blade are simulated at different angle of attack at different air velocity

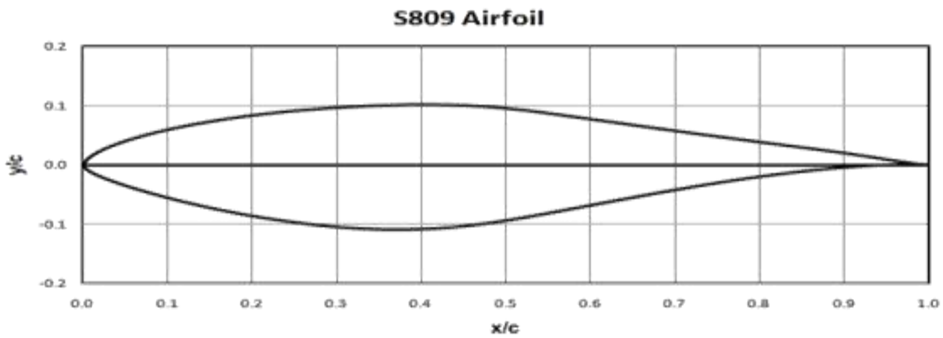


Figure.6. Airfoil NREL S809

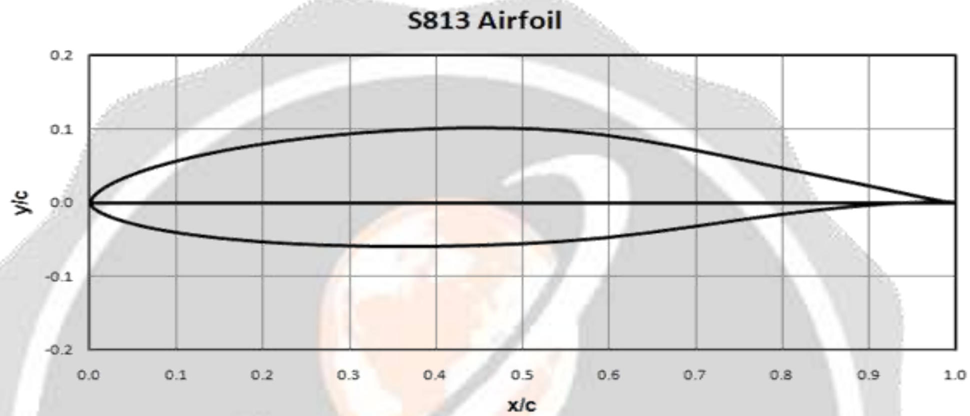


Figure.7. Airfoil NREL S813

Figure shows Two-dimensional modeling of airfoil NREL S809 which is used for the CFD analysis. GAMBIT is the pre-processor of ANSYS-Fluent Airfoils is modeled in GAMBIT with existing airfoil coordinates**.**

4.2.Mesh Generation

Meshing is a critical operation in CFD. In this process, the geometry is discretized into large numbers of small Element and nodes. The arrangement of nodes and element in space in an appropriate manner is called mesh. The CFD analysis accuracy and duration depends on the mesh size and orientations. With the increase in mesh size (increasing no. of element), the CFD analysis speed decrease but the accuracy increase.

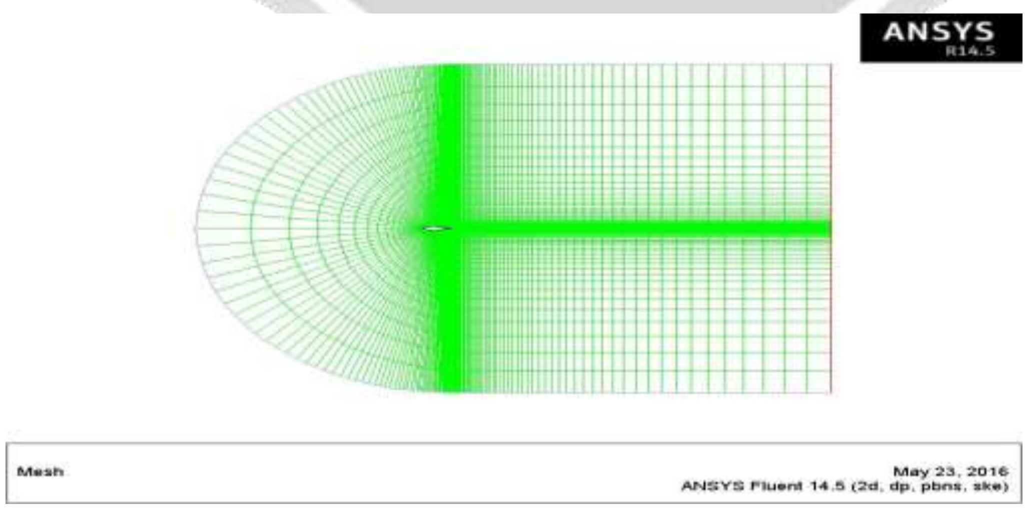


Figure.7. Complete Mesh

A large number of grids around the airfoil surface are used to capture the pressure gradient accurately at the boundary layer

4.3 .Boundary condition:

Velocity inlet far-field and pressure outlet boundary condition was used in both meshing methods as the computational domain is large enough. Aerofoil is treated as stationary wall condition with no slip shear condition. For different velocity and attack angle results, horizontal and vertical component was used in solution for stationary wall condition with no slip shear condition.

Airfoil	NREL S809 ,NERL S813
Fluid Material	Air
Temperature	288 K
Kinematic Viscosity	1.7894e-05 Kg/m s
Angle of attack	0, 5, 10, 15, 20, 25 degree
Air Velocity	15m/s 30m/s
Density	1.225kg/m3
CFD algorithm	SIMPLE(default option)
Turbulent model	Viscous- Realizable k-e
Interpolating scheme	Gradient (least squares cell based) Pressure (Standard) Momentum (Second Order Upwind) Turbulent kinetic energy (First Order Upwind) Turbulent dissipation rate(First Order Upwind)
Boundary condition	Velocity inlet far field, pressure outlet Stationary wall with no slip shear condition

Table.1. Computational condition

5.Results and Discussion

Geometric model NREL S809 and NREL S813 has different pressure distribution, velocity distribution and path line flow results contours at different 15m/s velocities and at 10 degree angle of attack which are given below For inlet velocity **15m/s and 30m/s (Low and high velocity zones)** Angle of Attack: **0, 5, 10, 15, 20, 25 degree (selection of angle of attack based on the permissible limit for the wing turbine blade)** respectively

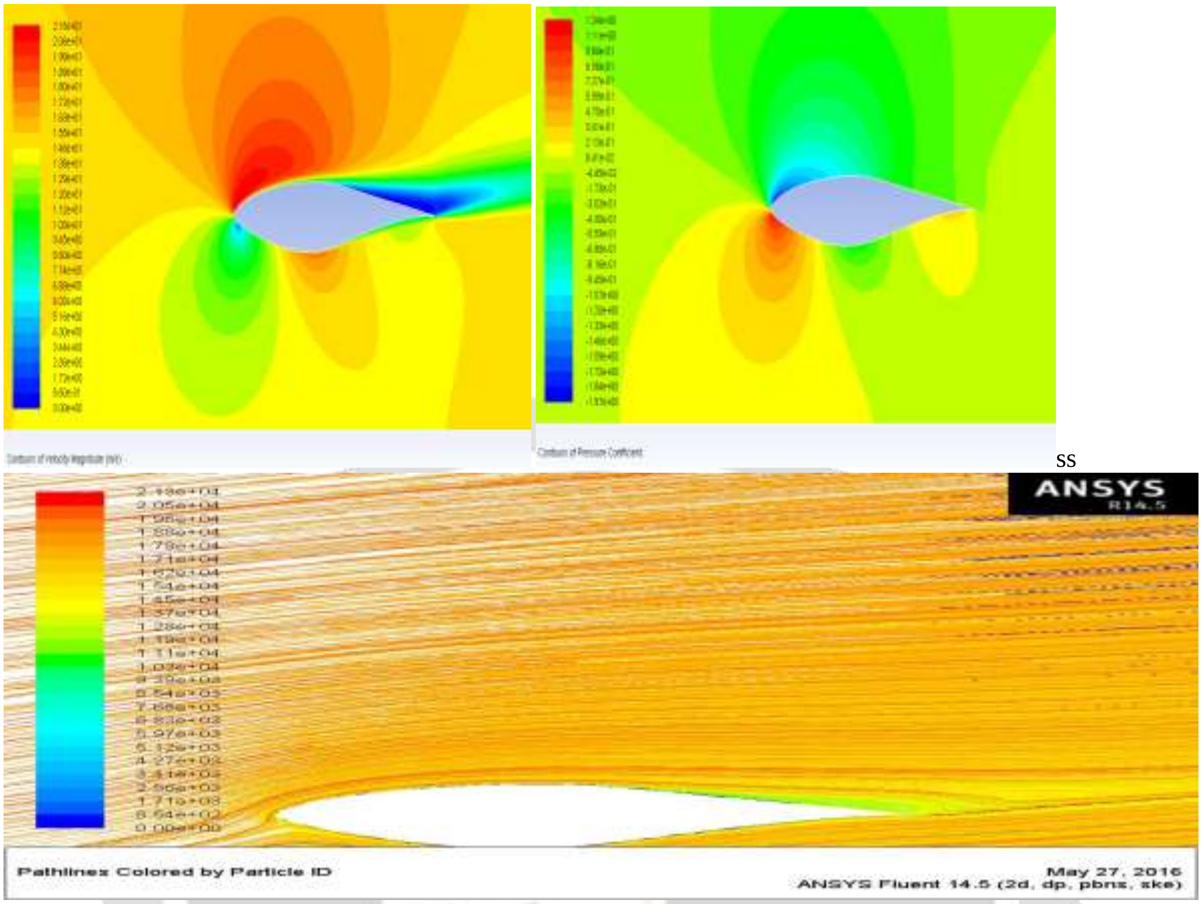


Figure.8.velocity Static pressure contours and path line flow for the optimized profile for S809

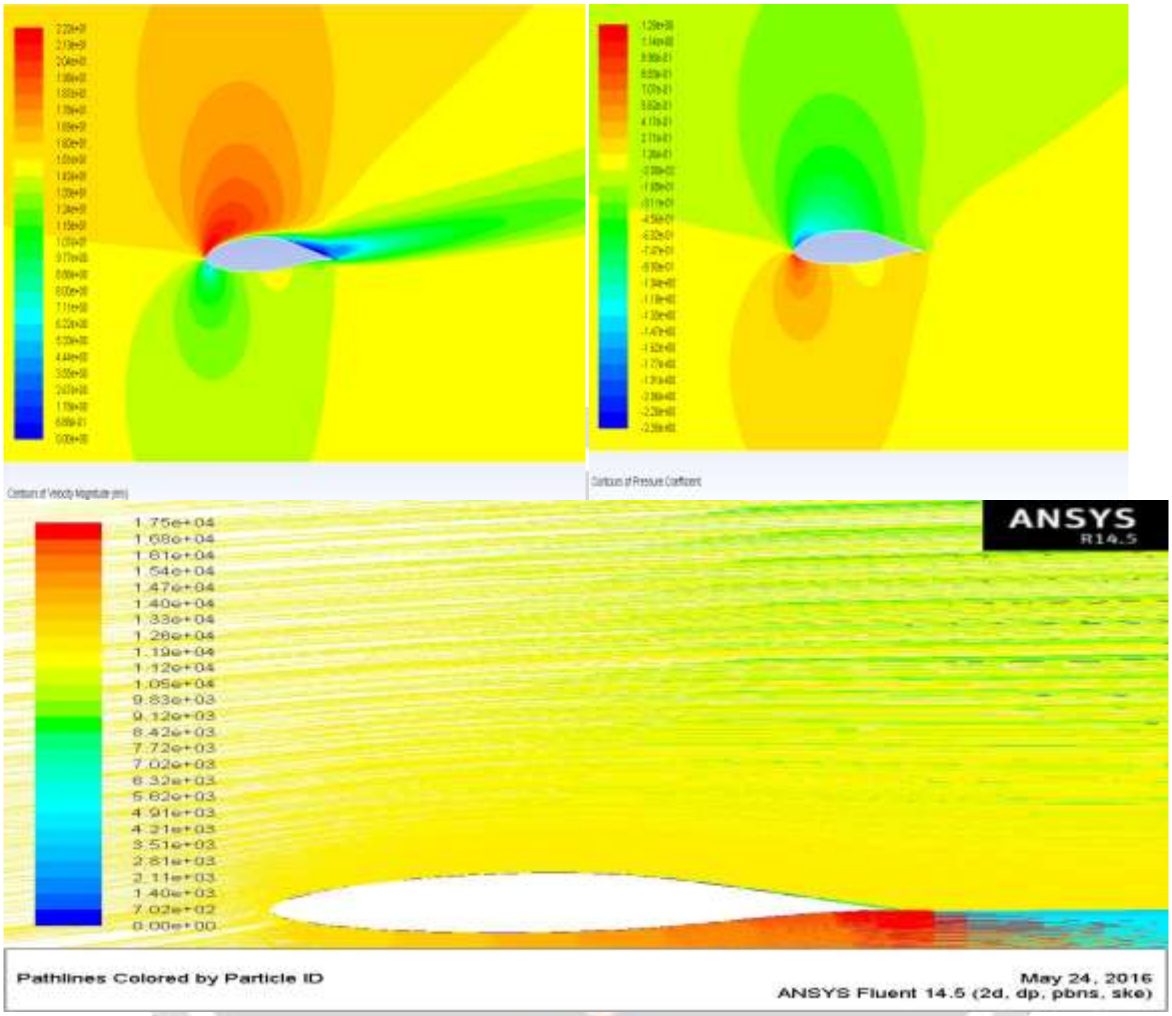


Figure.9.velocity Static pressure contours and path line flow for the optimized profile for S813

Above figures 8 andf 9 show at ten degree of angle of attack the pressure variation in air foil by different colours. Red colour show high pressure and blue coloure show low pressure thus at the upper porsion of air foil low and compare to high pressure at lower surface and at the leading edge high presure because it is stagnation point in flow field that show by red colour. And same for velocity that velocity variation in air foil by different coloure. Red colour show high velocity and blue colour show low velocity thus at the upper porsion of air foil velocity is high and lower portion compare to velocity low and at the leading edge of airfoil aproximate zero velocity because it is stagnation point in flow field that show by blue coloure also the direction of velocity vector and path line flow over the airfoil.

4.4. Lift and Drag Coefficient

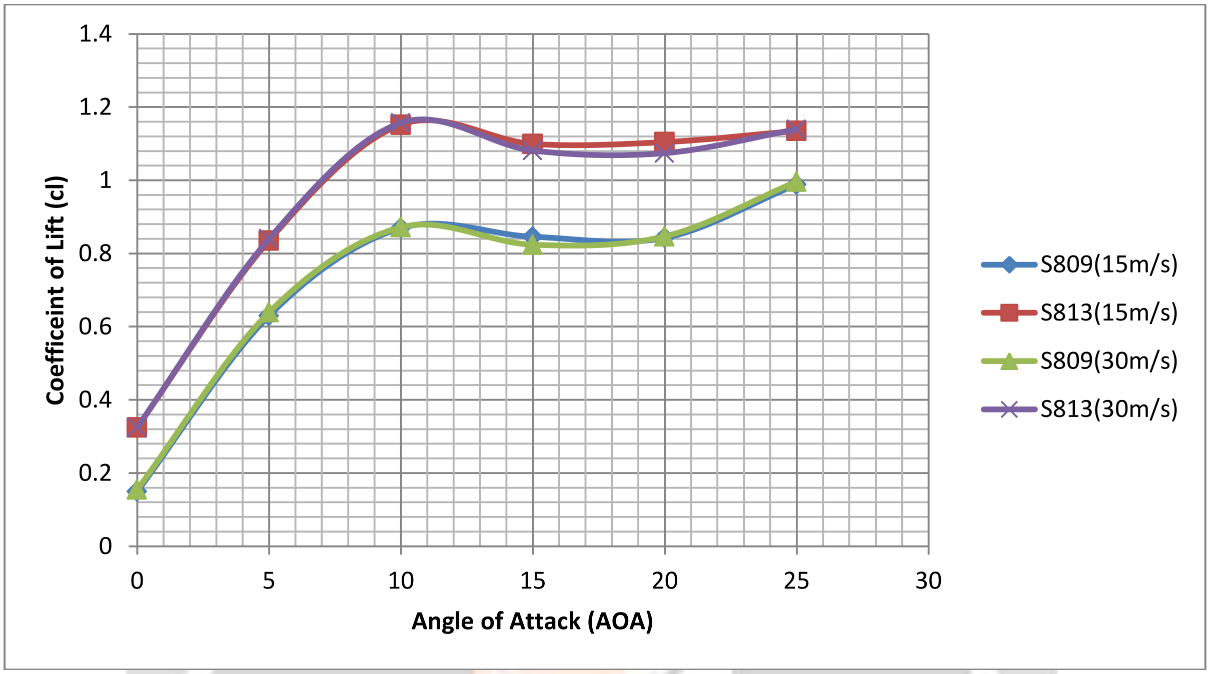


Figure.10. Coefficient of lift At different angle of attack(AOA

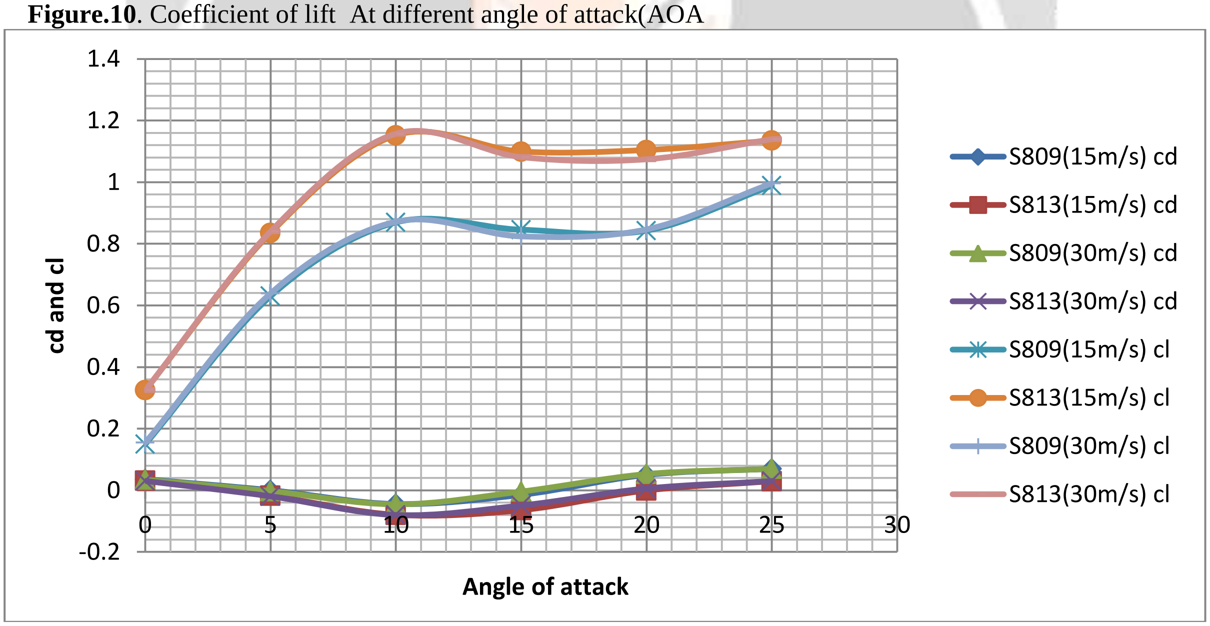


Figure.11. Coefficient of lift At different angle of attack(AOA)

The coefficient of Drag and Lift For airfoil NREL S809 and NREL S813 for the angle of attack from 0 to 25 degree and 15m/s,30m/s velocity. NRELS809 and NREL S813 is showing different Coefficient of Drag and Lift at low velocity 15m/s and high velocity 30m/s airfoil NREL S813 has maximum lift and minimum drag compare with the F airfoil NREL S809.Different color of curve shows the maximum and minimum coefficient of drag and lift.

6. Conclusion from CFD Study

In this work for low velocity and high velocity wind turbine blade are investigated at various angle of attack and comparison and examine of the overall aerodynamic performance of airfoil . Airfoil NREL S809 and NREL S813 with various angle of attack, it can see

Critical angle of attack 11 degree

- Optimum cd
  cd= -0.04 for S809 at 15m/s and 30m/s
  = -0.08 for S813 at 15m/s and 30m/s
- Optimum cl
  cl= 0.92 for S809 at 15m/s and 30m/s
  cl= 1.16 for S813 at 15m/s and 30m/s

- As angle of attack increase separation point shift from trailing edge to leading edge

7. FSI Simulation for airfoil S809 and S813

The verified CFD results are used for wind blade pressure load and transferred to the structural surface load. Only the interaction of wind blade surface considered to verify the wind blade aerodynamic load Pressure coming on the blades from CFD Study is used as input Boundary condition in Structure Analysis to calculate the stresses and defection due to wind pressure coming on the blades. The Analysis is performed at low to high wind zones to check the structure integrity of the blades

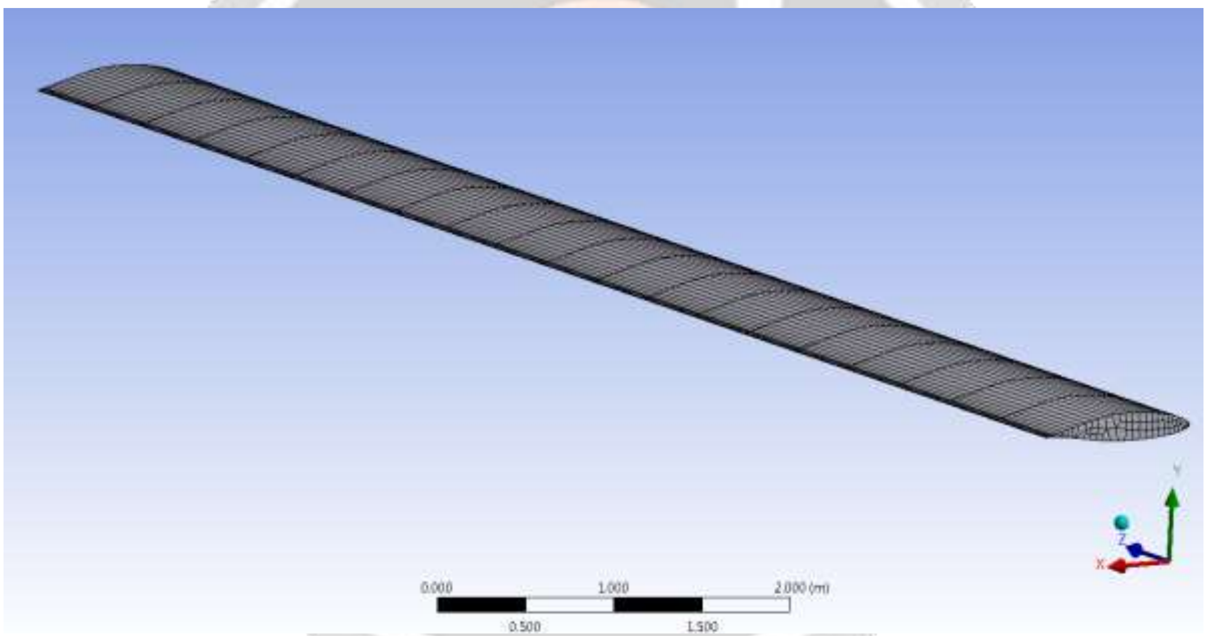


Figure.12 Structural Geometry and Meshing

7.1. Boundary condition

In structural Analysis we have two wind turbine blade Airfoil NREL S809 and NREL for structural investigation considered to verify the wind blade aerodynamic load Pressure coming on the blades from CFD Study is used as input Boundary condition. One end has Fixed support and pressure acting on another end.

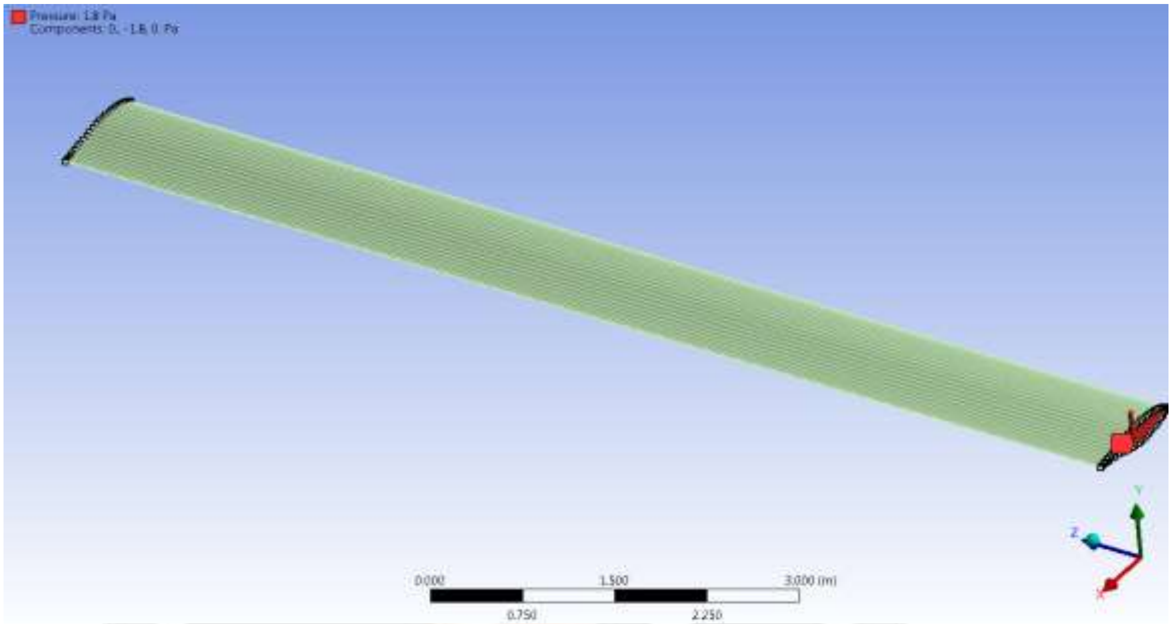


Figure.13.Boundary condition

8.Result and discussion of Fluid Structure Interaction (FSI) for S809 and S813

For the structural analysis the different colour shows the maximum and minimum stresses and deformation at 10 degree of angle of attack at fixed support end and pressure acting end. At fixed support stresses are maximum and at free end deformation is maximum.

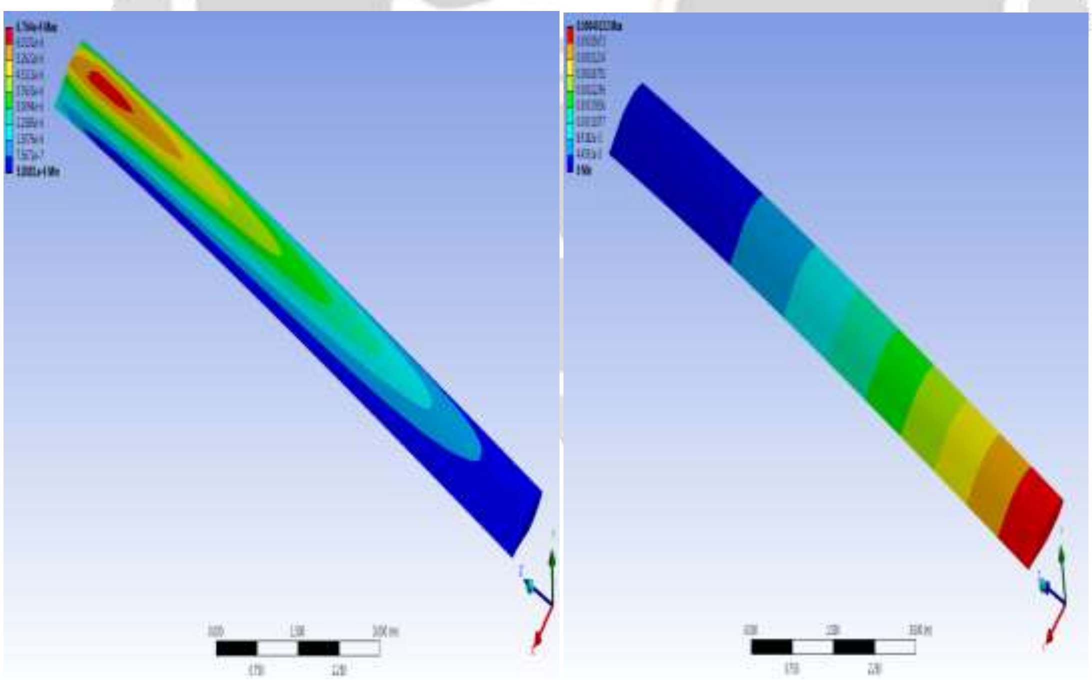


Figure.14. Structural stress and Deformation at 10°AOA of S809

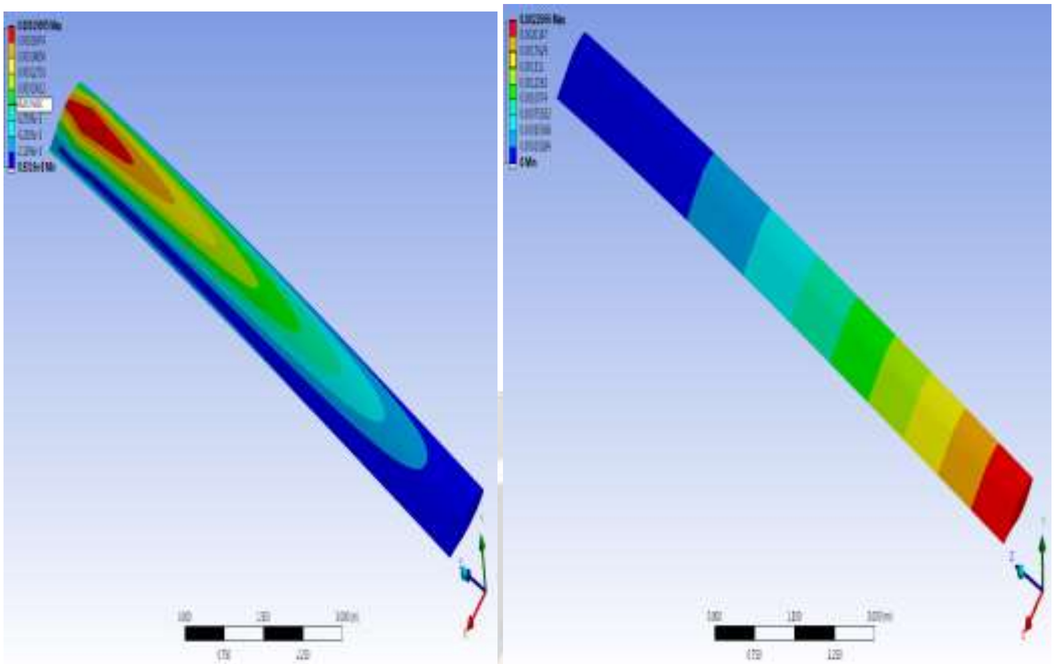


Figure.15. Structural stress and Deformation at 10°AOA of S813

- S809 - S813

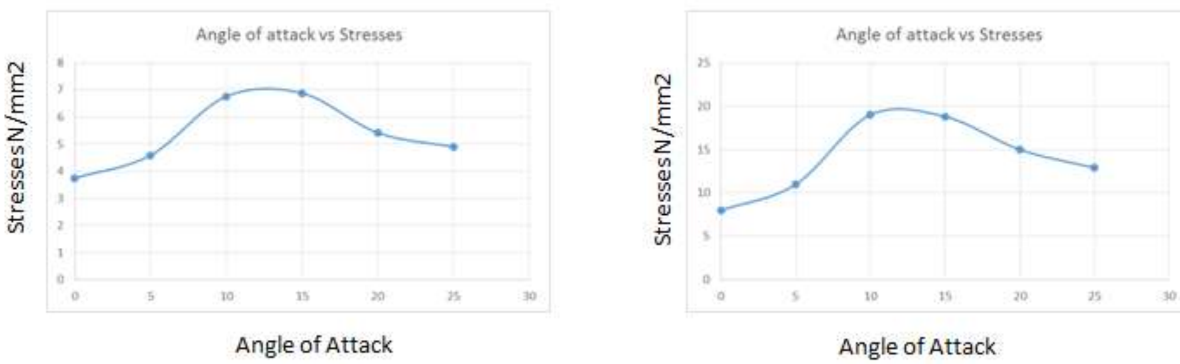


Figure.16.structural analysis for stresses from 0° to 25°AOA

- S809 - S813

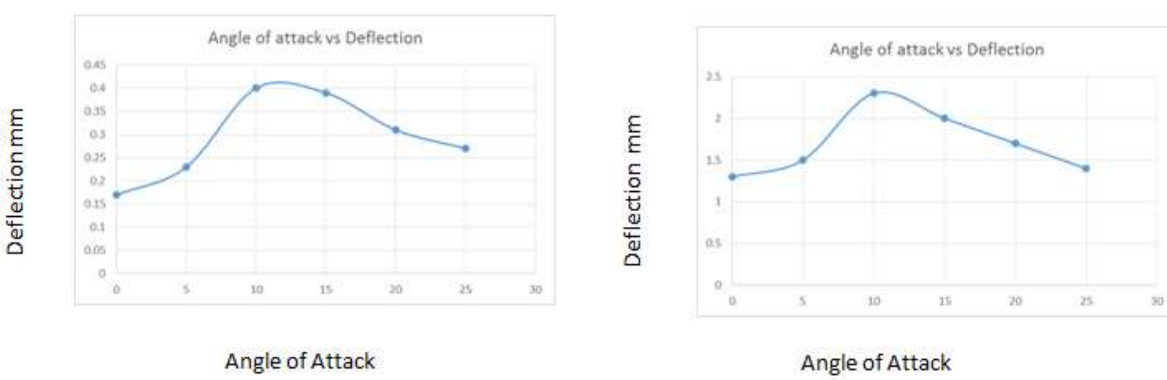


Figure.17. structural analysis for Deflection from 0° to 25°AOA

Conclusion from structural analysis

- Stresses and Deflection is maximum for S 813 blade compared to S 809 blade
- The Stresses produced in both the blades is < permissible young's modulus i.e. Design of Both blades are safe
- In both blades, with increase in angle of attack the stresses and deflection increases at certain point, then decreases due to the wing load variation on the blade
- The maximum stress and deflection for both blades found to be 12-15 degree.

Conclusion from Fluid Structure Interaction

- The optimum angle from CFD Study is found to be 11 degree
- At 11 degree stresses and deflection is maximum
- The design angle for both blades should be between 7 degree to 10 degree for better structural performance.

References

[1] Aerodynamics Analysis of Small Horizontal Axis Wind Turbine Blades by Using 2D and 3D CFD Modelling by Han cao

[2]T. Al-Shemmeri Wind Turbines© 2010 T. Al-Shemmeri & Ventus Publishing ApSISBN 978-87-7681-692-6

[3]Dr. Sathyajith MathewWind Energy Fundamentals, Resource Analysis and Economics

[4]The Evaluation of Aerodynamic Interaction of Wind Blade Using Fluid Structure Issnteraction Method Kyoungsoo Lee, Ziaul Huque, Raghava Kommalapati, and Sang-Eul Han.