

2D Simulation of H-Darrieus type vertical axis wind turbine using transition turbulence model in OpenFOAM

K S S Satya Sasank

Under the guidance of
Mr. Pankaj Negi,
Assistant professor

Dept of Mechanical Engineering, Graphic Era Hill University.

Abstract

Turbulence modeling is used to predict the airflow behavior past objects. Based on the type of flow conditions different turbulence models are available. Transition turbulence model is one such models which can be used for both laminar and turbulent flow conditions. This can be helpful in a situation where the airflow changes from laminar to turbulent and back continuously, like the airflow past wind turbines. A study on this is already performed using ANSYS in [1], a similar environment is setup and a 2D model of flow around vertical axis H Darrieus type wind turbine is developed and analysed with the help of opensource tools.

1 Introduction

Transition turbulence model helps us predict the change from laminar to turbulent flows in fluids and their respective effects on the overall solution. As it is known, CFD resolves the fluid dynamic equations and it is certainly more realistic than the 1D models but there are many other problematic issues like stall and turbulence modeling, unsteady rotational effects and long computation time. In this study, a 2D CFD model of a H Darrieus wind turbine with 3 blades and NACA 0015 symmetrical airfoil is generated using FreeCAD 0.19 which is an open source CAD tool and analysed using the KomegaSSTLM which is also known as the GammaReTheta model, a transition turbulence model that comes with the OpenFOAM.v.2112 .

The GammaReTheta model is based of the K omega SST model which is mainly used for external simulations. The K Omega model solves 2 equations for K and Omega. This model is modified to solve for 2 more equations for Gamma and Re-theta. These 2 terms control the switching of the model from turbulent to laminar. Gamma also known as turbulence intermittency determines the percentage of time the flow is turbulent (E.g., Gamma = 0 = fully laminar; 1 = fully turbulent). Re-theta is the Reynolds number where transition occurs.

2 Problem Statement

To simulate and understand the airflow over a H-Darrieus type vertical axis wind turbine blades in a 2D setup as shown in the figure using the transition turbulence model using OpenFOAM.

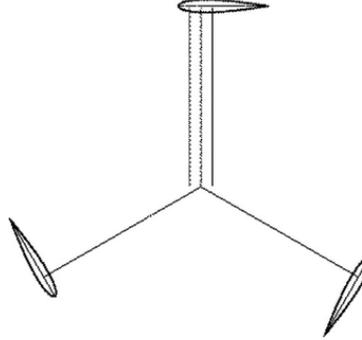


Figure 1: Top view of the vertical axis wind turbine

The chord length is taken as 0.4m and the radius of the blades is taken as 1.25m. The inlet speed being 6m/s from the left side of the image and the wind turbine is rotating with a speed of 15 rad/s in the direction opposite to the wind.

3 Governing Equations and Models

The following are the governing equations involved in the GammaRetheta turbulence model.

$$\frac{\partial(\rho\gamma)}{\partial t} + \nabla(\rho U\gamma) = \nabla\left(\left(\mu + \frac{\mu_t}{\sigma_\gamma}\right)\nabla\gamma\right) + P_\gamma - D_\gamma \quad (1)$$

$$\frac{\partial(\rho\overline{Re_{\theta,t}})}{\partial t} + \nabla(\rho U\overline{Re_{\theta,t}}) = \nabla\left(\left(\mu + \frac{\mu_t}{\sigma_{\theta,t}}\right)\nabla\overline{Re_{\theta,t}}\right) + P_{\theta,t} \quad (2)$$

The above equations solved based on the inputs given and the turbulence model switches between laminar and turbulent accordingly.

4 Simulation Procedure

4.1 Geometry and Mesh

First the 2D CAD model of the blades needs to be generated. This is done with the help of FreeCAD which is an opensource CAD software. For this project the FreeCAD v 0.19 has been used. The part design utility of this tool will be used. The blade profile is extracted from [Airfoiltools](#) which is an online data bank of airfoils. Here the coordinates for the airfoil shape can be found in a dat file. The FreeCAD's Airfoilimport&Scale is a macros for airfoils which takes these coordinates and generates the airfoil shape and with inbuilt options like Pad and array a 2D blade profile is

generated as shown. The dimension in the 3rd axis wont be an important factor since the analysis is conducted in 2D. Then the stl file of this model is exported using the export tool.

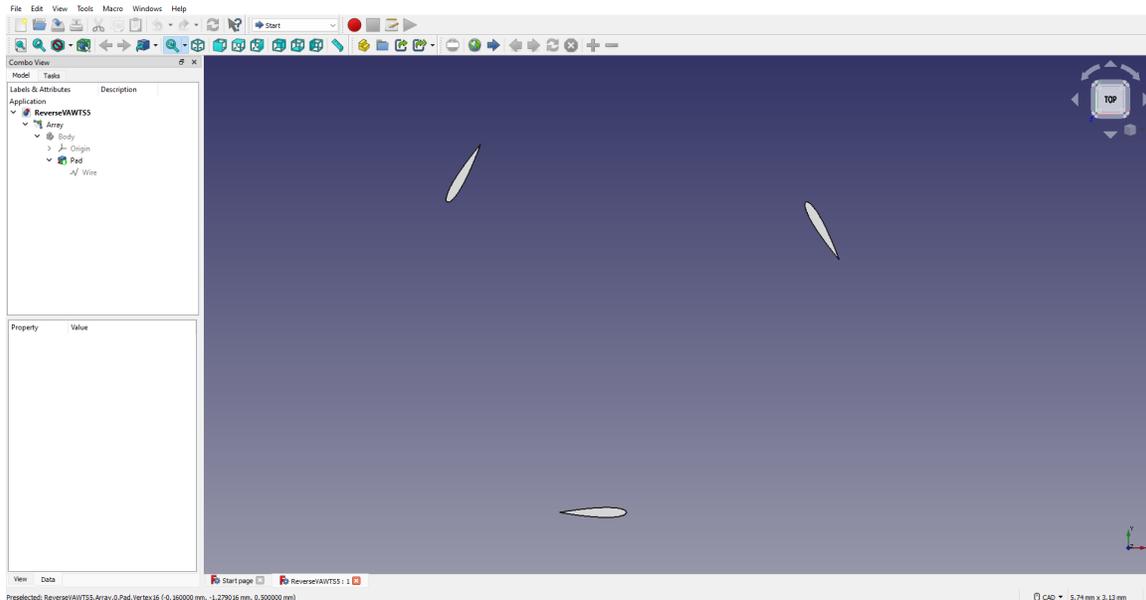


Figure 2: 3D model of the blades

This stl is file is then imported and meshed using the snappyhexmesh utility of openfoam. For this case to get the dynamic mesh we use the overset mesh feature. The overset mesh places one mesh on top of another. This facilitates relative motion between the meshes without having the problem of mesh deformation. The information will be communicated between both the meshes. This meshing requires creation of seperate boundaries of overset type. Also the solvers used for this type of meshing will be slightly different from the normal solvers used. The case directory is created from the airfoil setup in the \$FOAM_TUTORIALS /incompressible /over-SimpleFoam /aerofoil . It is seen that in the case directory there are 4 folders aerofoil _overset, aerofoil _snappyhexmesh, background _overset, background _snappyhexmesh. This is because first the aerofoil will be created on a plane. This will be then made to merge on top of a larger plane which will be acting as a wind tunnel. This facilitates the mesh when the 3 blades are to be rotated. To get the mesh, In the case directory/airfoil _snappyhexmesh/constant/triSurface there already exists a .obj file. The generated stl will be placed here and the name of the file will be changed in casedirectory/airfoil _snappyhexmesh/system/snappyhexmeshdict . Additional features such as the refinement layers are added to get the target mesh size. Also the size of the wind tunnel can be altered in the blockmeshdict of the same folder and the blockmeshdict of the casedirectory/background _snappyhexmesh/system/.

Additional features such as refinement layers are used to obtain a good first layer height. It is mandatory for the KomegaSSTLM model to have a decent first layer height. since it works better at a yplus value near to 1. The correlation between the yplus and first layer height is available online @ sites like [YplusCalculator](#). Upon giving the case parameters the first layer height is obtained as 0.00004m. Obtaining this first cell height will not be easy. After several iterations and playing with several options a decent mesh is obtained will satisfies all the mesh criteria. The first cell height

in the range of $8e-5m$ is obtained with a cell count of 12,64,220. The $yplus$ obtained at the blade varied from 0.7 to 4. The mesh generated is as shown.

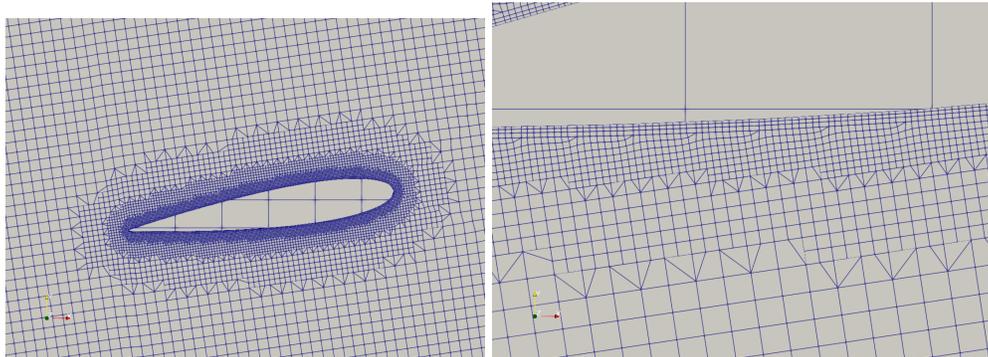


Figure 3: meshing of the blades

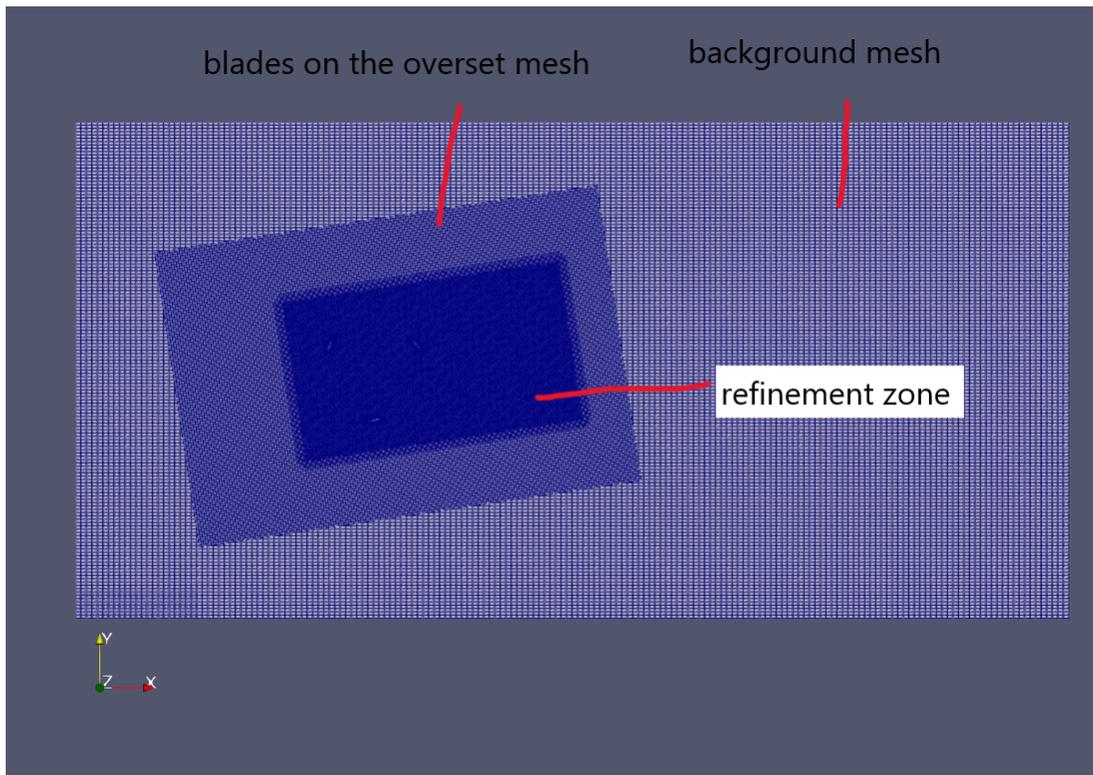


Figure 4: overall mesh

The wind tunnel shown above has a dimension of 5m in x-direction and 3m in the y-direction. However, running test cases on this mesh will demand a lot of computational power, time and storage. Hence a single airfoil is meshed with the same mesh settings. This mesh is used to run test cases on boundary conditions, dynamic mesh settings, etc.

4.2 Initial and Boundary Conditions

All the initial and boundary conditions are to be set up in the single folder which is the 0.orig folder present in background_overset in the case directory. The Boundaries assigned while meshing ultimately boil down to inlet, Outlet, walls. Walls are the wings and wall BC is applied here. Some of the other patches are front and back which will be empty and the top and bottom. The inlet BC will be a fixed value and the outlet BC is given as zero-gradient for all the parameters and wall functions have been applied for the wings. The inlet velocity is given as 6m/s and the inlet pressure is given as zero-gradient and a uniform pressure is to be maintained at the outlet.

4.3 Solver

The system folder consists of controldict where the type of solver and end time are to be mentioned. The type of solver used here is overSimpleFoam, overPimpleDyMFoam which are the overset variants of simplefoam and pimplefoam. These solvers track the information between the two meshes while accounting for the motion between the meshes. To keep the courant number to a minimum a low time-step difference is implemented with adjustable time-step to keep the solution from not blowing up. The final inputs to be given are the type of scheme to be used and the solvers residual control for each parameters. These options are set in fvSchemes and fvSolutions. Linearupwind schemes are used for pressure, velocity, gammaInt and Rethetat. The solver settings are as follows

Pressure:

```
solver          GAMG;
smoother        GaussSeidel;
tolerance       1e-6;
relTol          0;
```

For all other parameters

```
solver          smoothSolver;
smoother        GaussSeidel;
tolerance       1e-6;
relTol          0.01;
```

A relaxation factor of 0.3 is given for all the parameters in the study. Running the test cases with a setup like this will require a lot of computation power for the test cases computer with a 6core processor with a base speed of 3.59GHz is used. The case needs to run in parallel hence the decomposePardict is given the entry of 6 subdomains in a scotch method. The case setup is now complete and the simulation can be initiated in the casedirectory using the ./Allrun command in the terminal. This command executes the following commands in each folder.

```
cd aerofoil_snappyhexmesh
blockMesh
snappyHexMesh
cd ..
cd aerofoil_overset
extrudeMesh
createPatch -overwrite
```

```
cd ..
cd aerofoil_snappyhexmesh
blockMesh
snappyHexMesh
cd ..
cd aerofoil_snappyhexmesh
extrudeMesh
createPatch -overwrite
mergeMeshes ../aerofoil_overset .
topoSet
cp -r 0.orig 0
setFields
decomposePar
$SOLVER
```

The simulation takes around 5-7 mins for a single blade setup and takes several hours for a 3 blade setup and can go even further based on the end time. The results need to be reassembled using the reconstructPar command once the domain is reassembled the results are then viewed and post processed using paraView.

5 Results and Discussions

The simulation is first run on a steadystate to check for convergence for a 5 degree angle of attack.

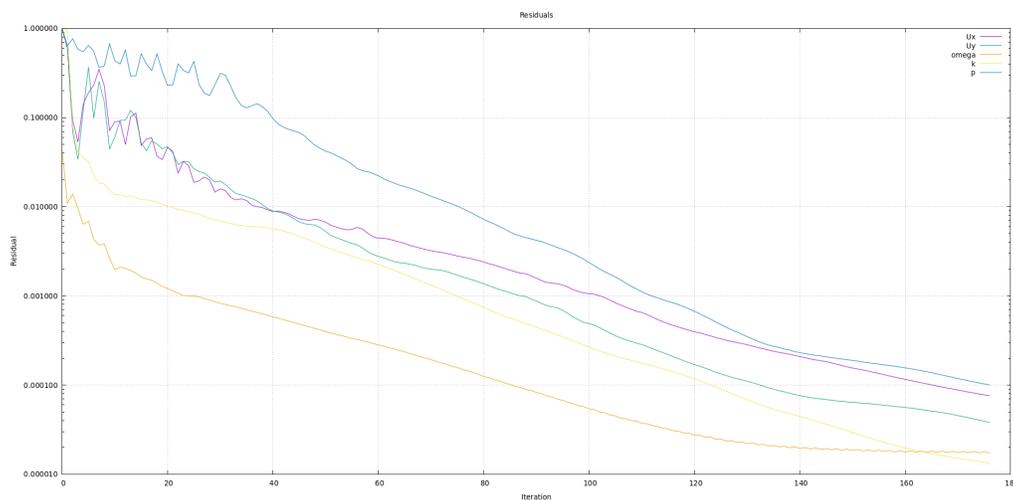


Figure 5: residuals

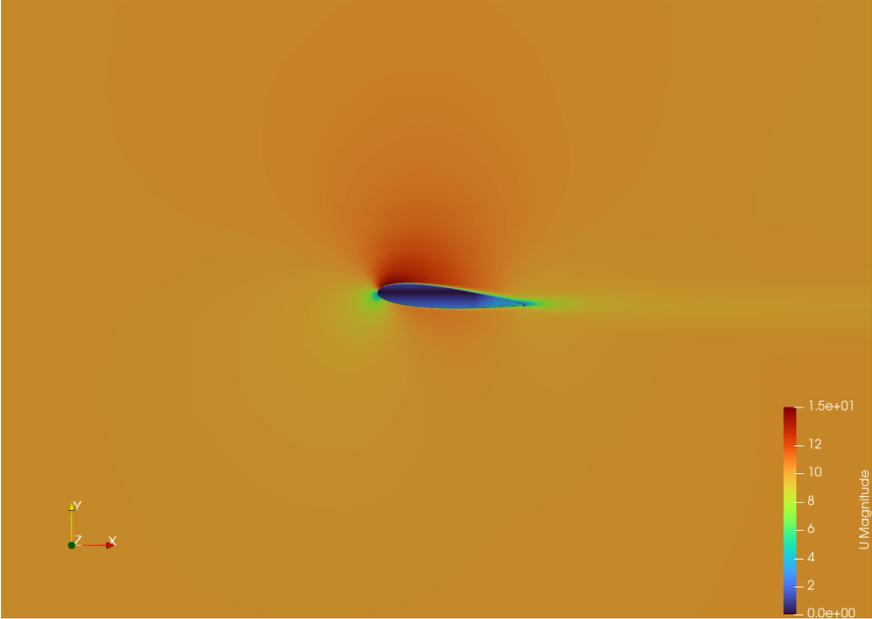


Figure 6: Velocity profile 5 deg AOA

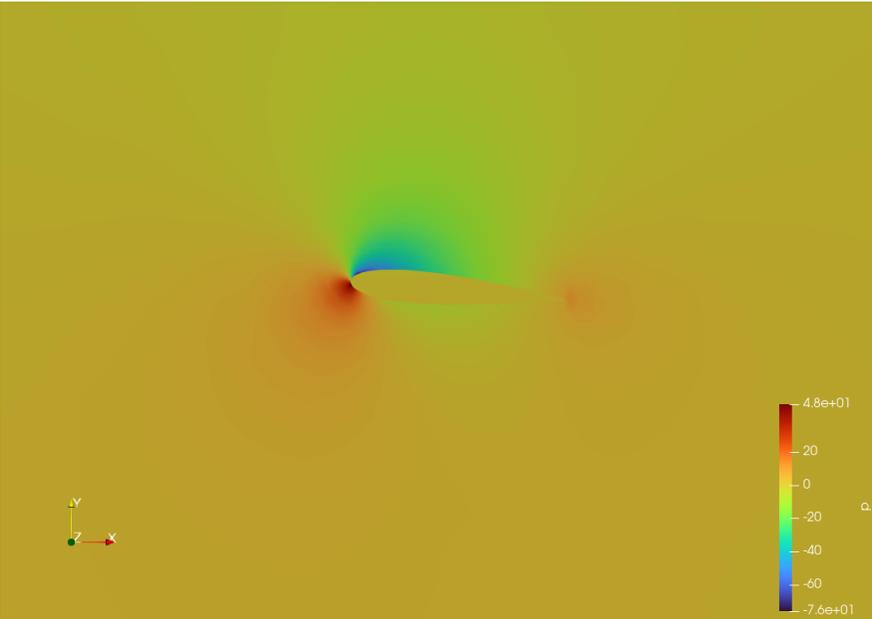


Figure 7: Pressure profile 5 deg AOA

It can be seen that the solution has converged and the contours display the physics accurately. The simulation is now run on different AOA independently and the results are compared with the xfoils results and the plots are as shown. The Plus indicate the percentage error of 5% in each axis.

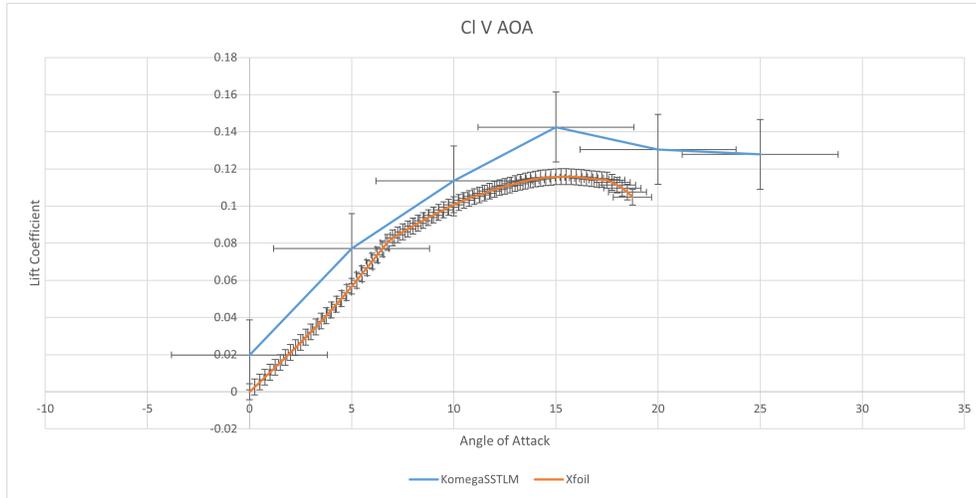


Figure 8: Lift coefficient

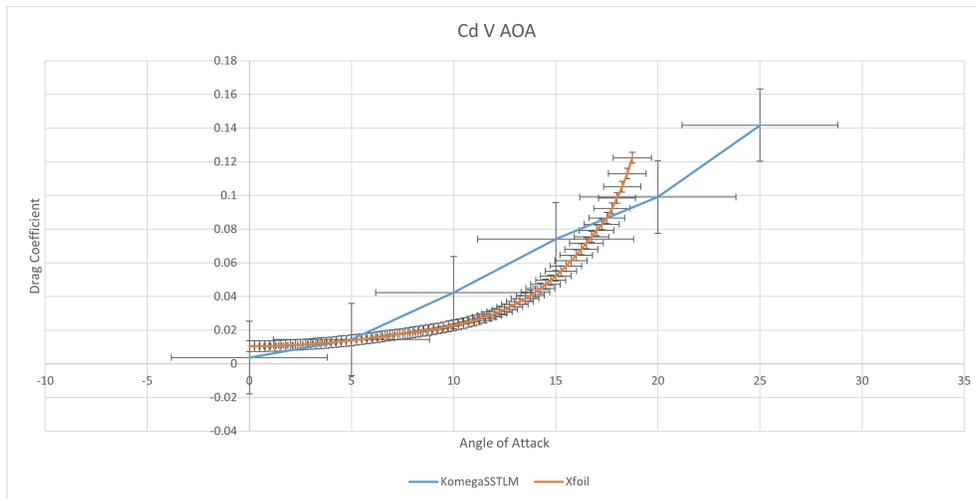


Figure 9: Drag coefficient

The 3 foil configuration is run on a more powerful machine and the contours for velocity and pressure are as shown.

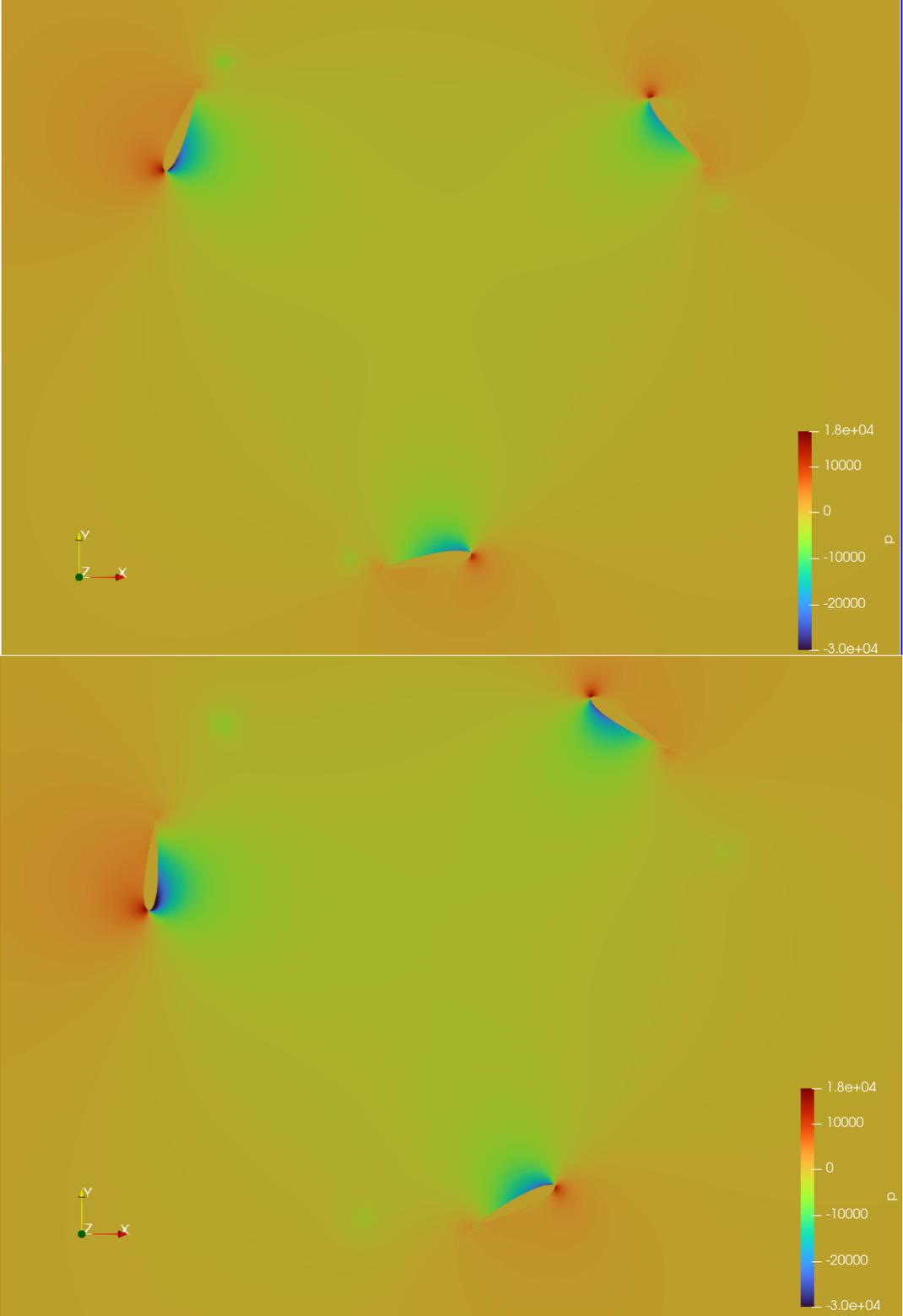


Figure 10: Pressure contours at initial and 3 sec later

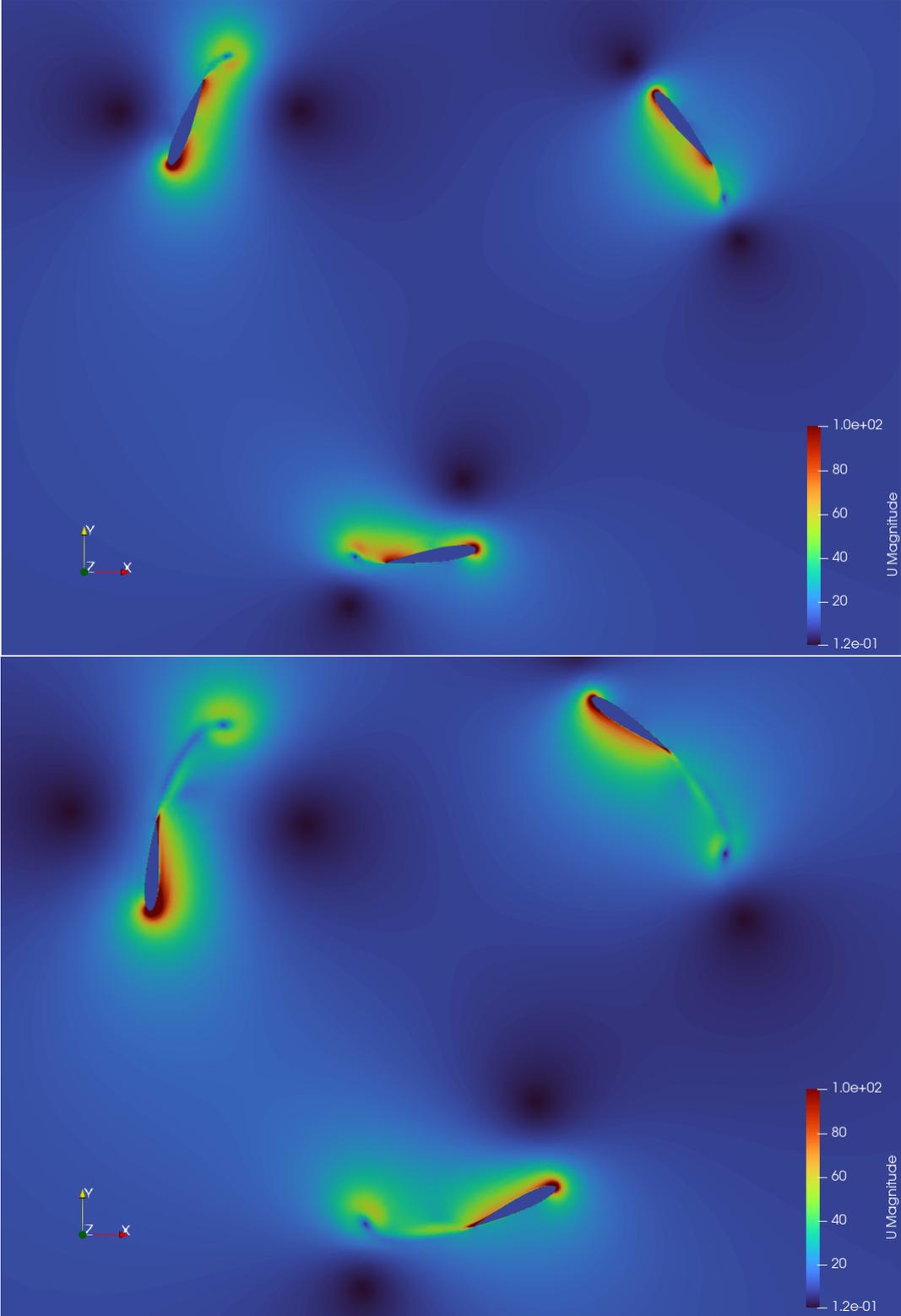


Figure 11: Velocity contours at initial and 3 sec later

It can be seen that the model is able to capture the physics involved.

The usage of transition turbulence models for an engineering application has been explored in this study. Different meshing techniques have been implemented to get a mesh suitable to the problem statement. In this aspect the commercial solvers such as ANSYS have an upper-hand as they have better control over the geometry and the mesh. The OpenFOAM's capability to solve a complex problem such as the current problem under study has been experimented. The complete solver's capabilities are yet uncovered and cannot be commented based on a simple experiment. However, the performance of the blade cannot be evaluated in this current paper since it involves more complex physics in 3D simulation and is scope of work for the future.

References

1. 2D CFD Modeling of H-Darrieus Wind Turbines using a Transition Turbulence Model Rosario Lanzafame, Stefano Mauro*, Michele Messina Department of Industrial Engineering, University of Catania, Viale A. Doria 6, 95125, Catania, Italy
2. CFD simulation of a vertical axis wind turbine operating at a moderate tip speed ratio: Guidelines for minimum domain size and azimuthal increment Abdolrahim Rezaeiha a, *, Ivo Kalkman a, Bert Blocken a, b a Building Physics and Services, Department of the Built Environment, Eindhoven University of Technology, the Netherlands b Building Physics Section, Department of Civil Engineering, KU Leuven, Kasteelpark Arenberg , Belgium
3. CFD ANALYSIS OF A THREE BLADE VERTICAL AXIS WIND TURBINE KALAKANDA ALFRED SUNNY*, PRADEEP KUMAR*, PRISCILLA. S* and NALLAPANENI MANOJ KUMAR** *Department of Aerospace Engineering, Karunya University, Coimbatore-641114, Tamil Nadu, India
4. 3D CFD Analysis of a Vertical Axis Wind Turbine Andrea Alaimo *, Antonio Esposito, Antonio Messineo, Calogero Orlando and Davide Tumino Faculty of Engineering and Architecture, Kore University of Enna, Cittadella Universitaria, 94100 Enna, Italy
5. Development and Validation of CFD 2D Models for the Simulation of Micro H-Darrieus Turbines Subjected to High Boundary Layer Instabilities y Rosario Lanzafame 1, Stefano Mauro 1,* , Michele Messina 1 and Sebastian Brusca 2 1 Department of Civil Engineering and Architecture (DICAR), University of Catania, Via Santa Sofia 64, 95125 Catania, Italy 2 Department of Engineering, University of Messina, Contrada Di Dio, 98166 Messina, Italy
6. Implementation and application of the actuator line model by OpenFOAM for a vertical axis wind turbine L Riva1*, K-E Giljarhus2, B Hjertager2 and S M Kalvig2 1Department of Engineering Sciences, University of Agder, Norway 2Department of Mechanical and Structural Engineering and Material Science, University of Stavanger, Norway
7. Validation of the Gamma -Re-Theta Transition Model for Airfoils Operating in the Very Low Reynolds Number Regime Manuel Carreño Ruiz1 · Domenic D'Ambrosio1
8. Comparison of Aerodynamics Characteristics of NACA 0015 NACA 4415 Robiul I. Rubel , Md. K. Uddin , Md. Z. Islam and Md. Rokunuzzaman Department of Mechanical Engineering, Bangladesh Army University of Science Technology

9. Performance of Overset Mesh in Modeling the Wake of Sharp-Edge Bodies Suyash Verma and Arman Hemmati
10. COMPARISON OF OVERSET MESH WITH MORPHING MESH: FLOW OVER A FORCED OSCILLATING AND FREELY OSCILLATING 2D CYLINDER MICHAEL ALLETTO
11. <https://www.mdpi.com/1996-1073/13/21/5564>
12. <https://ui.adsabs.harvard.edu/abs/2017MS>