Flow simulation over 2D airfoil using OpenFOAM

Background
With the growing size and cost of modern wind turbines it is important that the aerodynamic tools are improved in accuracy and are capable of predicting well the time dependent loads in yaw, wind shear and dynamic inflow. In the IEA-MEXNext project, a large validation study was carried out where different codes of increasing complexity were compared with an extensive of experimental wind tunnel results from the MEXICO (Model Experiments in Controlled environment) model rotor.

Project Objectives
The project aim is to develop a fully three dimensional CFD code based on OpenFOAM to predict the unsteady aerodynamic loads on wind turbine rotors, in conditions of axial flow, yawed flow and dynamic inflow. Meanwhile, the code will be developed such that CFD/FSI (Fluid Structure Interaction) will be possible in a later stage.

After some verification and validation exercise, the code will be used in MEXNext II project, where present MEXICO experiment data are used for further validation.

Research Methodology:
An open source computational software package, OpenFOAM is used for the flow simulation over wind turbine. Many solvers and models are compiled in it and high-level syntax written by C++ language, which means the code can easily be modified further.

The momentum equation:

\[
\frac{\partial \rho U_i}{\partial t} + \nabla \cdot (\rho U_i U_j) = - \nabla p + \mu \nabla^2 U_i
\]

is represented by the code:

```c
solve
{
    fvms::ddt(rho, U);
    fvms::div((p/\mu) U);
    fvms::laplacian(mu, U);
    fvms::grad(p);
}
```

2D airfoil flow simulation:
The flow over NACA 63618 airfoil which is used in wind turbine blade is investigated using OpenFOAM, the steady incompressible solver simpleFoam and k-Omega SST turbulence model from Menter are chosen for this simulation. Pressure and velocity coupling for Navier-Stokes equation is solved by SIMPLE algorithm. The convective term is discrete with upwind scheme. Central scheme is used to discrete diffusion term and gradient term with second order.

Two types of grids are generated regarding of two cases simulation: low Reynolds number and high Reynolds number simulation. The previous case set y+ less than 1 in order to simulate sub layer flow in boundary layer. The later one used wall function for reducing computation time.

Mesh independency study
The result focus on at different angle of attack, the lift coefficient and drag coefficient compared with experimental data, which are defined as:

\[
C_L = \frac{L}{\frac{1}{2} \rho U^2 A} \quad C_D = \frac{D}{\frac{1}{2} \rho U^2 A}
\]

Lift coefficient comparison
Drag coefficient comparison

The lift coefficient agrees very well with experimental results before a.o.a (angle of attack) 10 degree, but deviation become larger with a.o.a increases. The flow becomes separate and stall in that region.

The drag coefficient obtained in OpenFOAM is quite higher than the experiment data, even at 0 a.o.a. That is because for this kind of airfoil, the flow is mostly dominated by laminar flow, but the k-omega SST turbulence model is assumed fully turbulent flow, which results in high drag.

Further Work:
A. Do simulation with transitional model, and compared with 'local flow' pressure distribution with proper airfoil.
B. Try different turbulence model and do 3D simulation.

Reference