Airflow Simulation over Ahmed Body

Viraj Belekar, Project Research Assistant, OpenFOAM team, FOSSEE Project, IIT Bombay

***Abstract*—This report aims to describe the calculation of turbulent flow field around a simple car-like geometry using the software Gmsh and OpenFOAM. It also aims to compare the aerodynamic efficiency of Ahmed Body with ruled frontal surfaces to that with plane frontal surfaces.**

***Keywords*—Ahmed Body, OpenFOAM, LaTeX, simpleFoam.**

I. Introduction

We have used Ahmed Body for this simulation. The Ahmed Body was first described by S. R. Ahmed in 1984 [1]. It represents a simplified, ground vehicle geometry of a bluff body type. The Ahmed Body allows for accurate flow simulation through its simple enough shape. It also retains some important practical features relevant to automobile bodies.

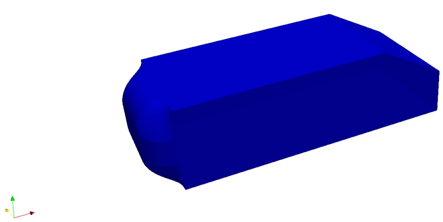

Figure 1: Ahmed Body

A. Geometry

The geometry of the Ahmed Body was created using the software Gmsh(v-2.12.0).The total length of the geometry is 1.044 m from front to end. Its height is 0.288 m and width is 0.389 m. The angle of the rare slanting surface is 30 degrees. The body is placed in a domain which is 5L-2L-2L(length-width-height) where L is the length of the Ahmed Body(1.044m).

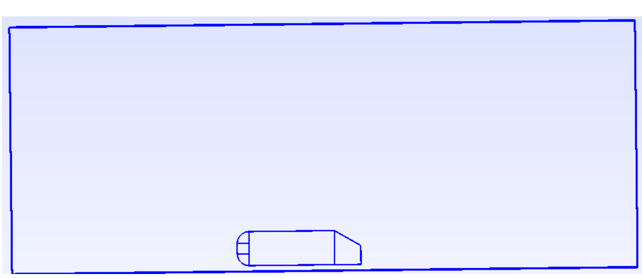

Figure 2: Ahmed Body with its computational domain

B. Meshing

The meshing for this simulation was done using the software Gmsh(v-2.12.0). 1D mesh, 2D mesh and 3D mesh over the Ahmed Body were created using Gmsh. The 3D mesh was later optimized using Netgen.

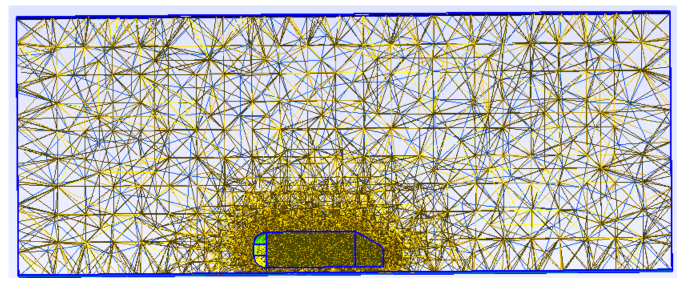

Figure 3: Cut-section of 3D Mesh

II. Analysis

The CFD analysis of the airflow over the Ahmed Body was done using the software OpenFOAM (v-3.0.1).

A. Boundary Conditions

Air enters the computational domain at a freestream velocity $u_\infty = 40m/s$ normal to the inlet surface. The reference pressure was taken to be 1 atm. The pressure at outlet was kept fixed and equal to the atmospheric pressure. The road or the bottom surface of the domain was defined as wall and the front, top and back surfaces of the domain were modeled as slip.

B. Turbulence Model

The Reynold's Number was calculated using the freestream velocity and the length of the body. It came out to be $2.8 * 10^6$. The kOmegaSST turbulence model of OpenFOAM is used for this simulation. This model is a combination of $k-\omega$ and $k-\epsilon$ models. The initial values of k and Omega were calculated to be 6.00 and 2.346 respectively. Following are the formulae used in the calculations:

Reynolds Number,

$$Re = \frac{UL}{\nu} \tag{1}$$

where,
U - Maximum velocity of the object relative to the fluid,
L - Characteristic linear dimension,
ν - Kinematic viscosity

Turbulent Energy,

$$k = \frac{3}{2}(UI)^2 \tag{2}$$

where,
U - Mean Flow Velocity
I - Turbulent intensity

Specific Turbulent Dissipation Rate,

$$\omega = \frac{\sqrt{k}}{l} \tag{3}$$

where,
k - Turbulent Energy
l - Turbulent length Scale

C. simpleFoam solver

Since we want to analyze steady-state turbulent flow for an incompressible fluid, we have used the simpleFoam solver. We do not need to solve the energy equation due to the incompressibility. The SIMPLE(Semi-Implicit Method for Pressure-Linked Equations) algorithm[2], which the simpleFoam solver is based upon, is solving the momentum equation (Equation 1) and the Poisson pressure equation (Equation 2).

$$(\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_j u_i}{\partial x_j}) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\mu \frac{\partial u_i}{\partial x_j}) + \rho f_i \tag{4}$$

$$\frac{\partial}{\partial x_i}(\frac{\partial p}{\partial x_i}) = -\frac{\partial}{\partial x_i}[\rho u_i u_j] \tag{5}$$

As OpenFOAM utilizes a collocated grid, Rhie-Chow interpolation is used for the pressure-velocity coupling. Following is the SIMPLE algorithm which is the basis of simpleFoam solver:

1) Set the boundary conditions.
2) Solve the discretized momentum equation to compute the intermediate velocity field.
3) Compute the mass fluxes at the cells faces.
4) Solve the pressure equation and apply under-relaxation.
5) Correct the mass fluxes at the cell faces.
6) Correct the velocities on the basis of the new pressure field.
7) Update the boundary conditions.
8) Repeat till convergence.

D. Force coefficients

For studying the airflow over an Ahmed Body, we have to calculate the force coefficients or the aerodynamic coefficients, viz. Co-efficient of Lift(C_L), Co-efficient of Drag(C_D) and Co-efficient of Moment(C_M). A force coefficient function was called in the controlDict file. It was defined as follows:

```
forceCoeffs1
{
type                     forceCoeffs;
functionObjectLibs ( "libforces.so" );
outputControl            timeStep;
timeInterval             1;
log                      yes;
patches                  (ahmed);
rhoName                  rhoInf;
rhoInf                   1.225;
liftDir                  (0 0 1);
dragDir                  (1 0 0);
CofR                     (0 0 0);
pitchAxis                (0 1 0);
magUInf                  40;
lRef                     1044;
Aref                     112032;
}
```

NOTE: Although the actual length of the Ahmed Body is 1.044 m, its length according to the scale used in Gmsh is 1044 units. Hence the value of lRef is taken to be 1044 and not 1.044. Similarly, the value of ARef is taken to be 112032 rather than 0.112032.

III. Results and Discussion

The C_L, C_D and C_M values are key figures in aerodynamics studies. We successfully obtained their values as C_L = 0.42, C_D = 0.32, C_M = -0.29. The plot of the force co-efficients against the simulation time is as follows:

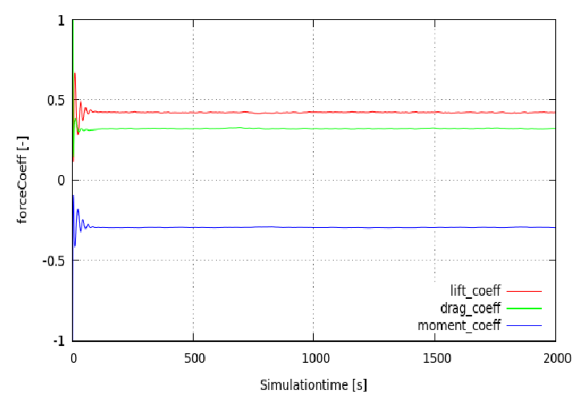

Figure 4: Force coefficients versus Simulation Time

We can visualize the pressure variation over the Ahmed Body from the pressure contour(Figure 4). As can be seen, the pressure is maximum on the front face and the sharp edges of the body.

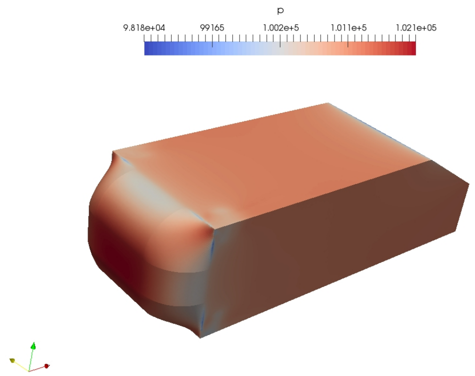

Figure 5: Variation of pressure over the Ahmed Body

We can also visualize the streamlines due to the flow using the Stream Tracer option in OpenFOAM(Figure 5). Vortices from the trailing edges of the body merge into two counter rotating vortices(one vortex on each side of the symmetry plane).

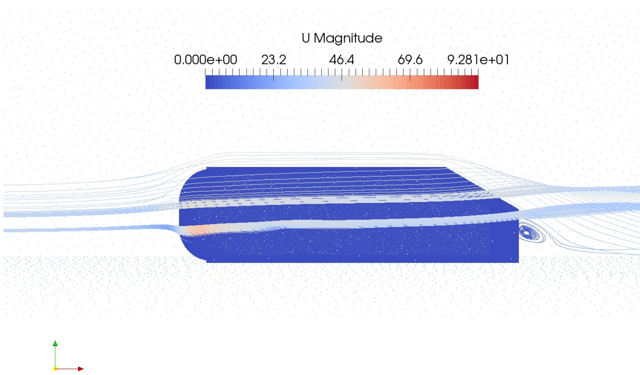

Figure 6a: Streamlines with first vortex

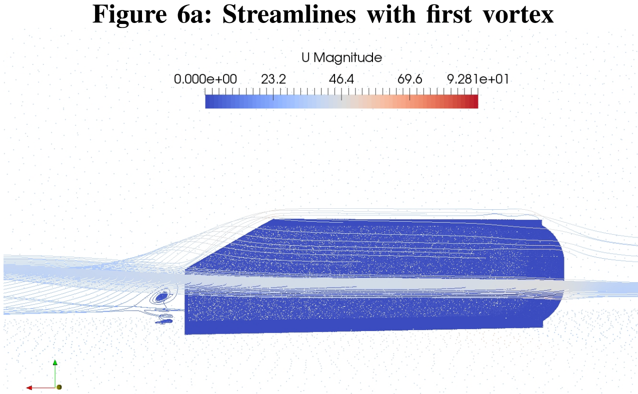

Figure 6b: Streamlines with second vortex

Figure 6 represents a plot of initial residuals against the timesteps. As we can see, the residuals are decreasing with the increase in timesteps on an average.

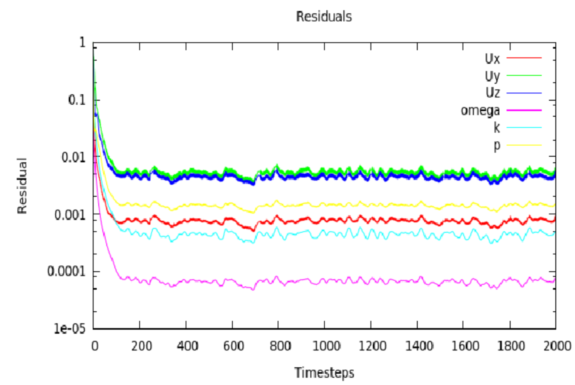

Figure 7: Residuals vs Timesteps

To conclude, the major features of the flow are captured very well by the kOmegaSST turbulence model.

IV. Airflow over Ahmed Body with plane surfaces

If we replace the ruled frontal surfaces with plane surfaces, there is a significant change in the values of aerodynamic coefficients.

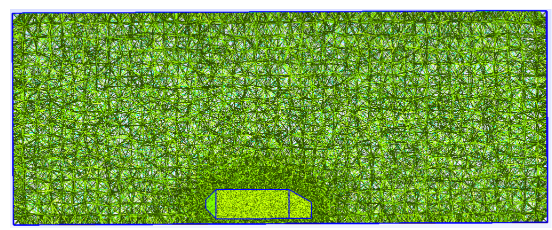

Figure 8: Cut section of 3D Mesh

The aerodynamic or force coefficients come out to be: C_L = 0.12, C_D = 0.44, C_M = -0.13.

Notice that the coefficient of drag increases by 37.5 % and the coefficient of lift decreases by 71.43 % . This is a significant change and it may considerably increase the fuel consumption of the vehicle. Hence, we can conclude that the design of Ahmed body with ruled surfaces is more aerodynamically efficient than that with plane surfaces.

References

[1] S.R. Ahmed, G. Ramm, Some Salient Features of the Time-Averaged Ground Vehicle Wake, SAE-Paper 840300, 1984

[2] The SIMPLE Algorithm `https://openfoamwiki.net/index.php/OpenFOAM_guide/The_ SIMPLE_ algorithm_ in_ OpenFOAM`