

Airflow Simulation over Convex Surface

keshav sharma, Project Research Assistant, OpenFOAM Team, FOSSEE Project, IIT Bombay

November 2, 2017

Abstract

This Report aim to describe the turbulent flow field through a Nozzle over an simple Convex Surface geometry by using the Gmsh and OPEN-Foam Softwares. It also demonstrate the Coanda Effect by comparing the flow field over Convex Surface to the flow field over Blunt Surface Geometry. The Coanda effect is the phenomena in which a jet flow attaches itself to a nearby surface and remains attached even when the surface curves away from the initial jet direction. In free surroundings, a jet of fluid entrains and mixes with its surroundings as it flows away from a Nozzle.

keywords - Coanda effect, Openfoam, Gmsh, SimpleFoam

1. Introduction

OpenFOAM (Open Source Field Operation and Manipulation) it's an Open Source Software project claim to be one of the best CFD tools currently available, the constant development and its highly technical structure, the fine implementation of common solvers and the possibility to edit and create equations and mathematical cases make OpenFOAM useful tool on researching. OpenFOAM is a C++ toolbox for the development of customized numerical solvers, and pre-/post-processing utilities for the solution of continuum mechanics problems, including computational fluid dynamics (CFD). The code is released as free and open source software under the GNU General Public License.

The external flow CFD analysis are commonly used to study the behavior of objects submerged in continuous non static fluids, it aims to determine how efficiently a body can move throw the medium and how they affect each other in the process. Here the fluid it's no confined between wall type conditions such as pipes or confined flow cases, the fluid its free to move around the object and interact only with its external "wet" layer, this means that there is the flow involving the body shape what must be analyzed not the core. Evaluating the pressure distribution on the structure generated by the resistances of the fluid the engineers can optimize the design to decrease the amount of energy needed to move the body or the disturb impact on the fluid patch that leaves the pass of the structure.

The Coandă effect is the tendency of a fluid jet to stay attached to a convex surface. As described by the eponymous Henri Coandă in different patents: "the tendency of a jet of fluid emerging from an orifice to follow an adjacent flat or

curved surface and to entrain fluid from the surroundings so that a region of lower pressure develops." The pressure effect, which is usually not indicated, is fundamental for the comprehension of the Coandă effect.

The principle was named after Romanian aerodynamics pioneer Henri Coandă, who was the first to recognize the practical application of the phenomenon in aircraft development.

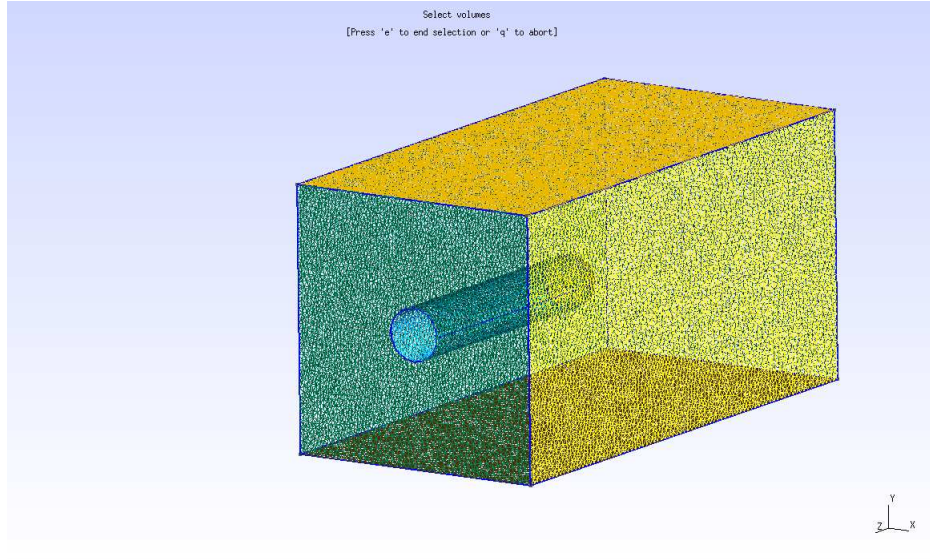
Many devices use Coanda effect. One notable application of the Coanda effect is the NOTAR™ helicopter. NOTAR (No Tail Rotor) is the name of a helicopter system which replaces the use of a tail rotor. The system uses a fan inside the tailboom to build a high volume of low-pressure air, which exits through two slots and creates a boundary layer flow of air along the tailboom utilizing the Coanda effect.

A. Geometry

The geometry of the curved Body was created using the software FreeCAD(v-0.16). The total length of the geometry is 0.5 m from front to end. Its height is 0.1 m and width is 0.1 m. The body is placed in a domain which is 2L-L-L (length-width-height) where L is the length of the curved Body(0.5m).

B. Meshing

The meshing for this simulation was done using the software Gmsh(v-2.15.0). 1D mesh, 2D mesh and 3D mesh over the Curved Body were created using Gmsh. The 3D mesh was later optimized using Netgen.



2. Analysis

The CFD analysis of the airflow over the Ahmed Body was done using the software OpenFOAM (v-5.0).

A. Boundary Conditions

Air enters the computational domain at a freestream velocity $u_\infty = 40\text{m/s}$ 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 ,top and side surfaces of the domain was defined as wall.

B. Turbulence Model

The Reynold's Number was calculated using the freestream velocity and the length of the body. The kOmegaSST turbulence model of OpenFOAM is used for this simulation. This model is a combination of k -w and k - 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,

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

where,

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

- Turbulent Energy,

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

where,

- U - Mean Flow Velocity
- I - Turbulent intensity

- Specific Turbulent Dissipation Rate

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

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).

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

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

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.

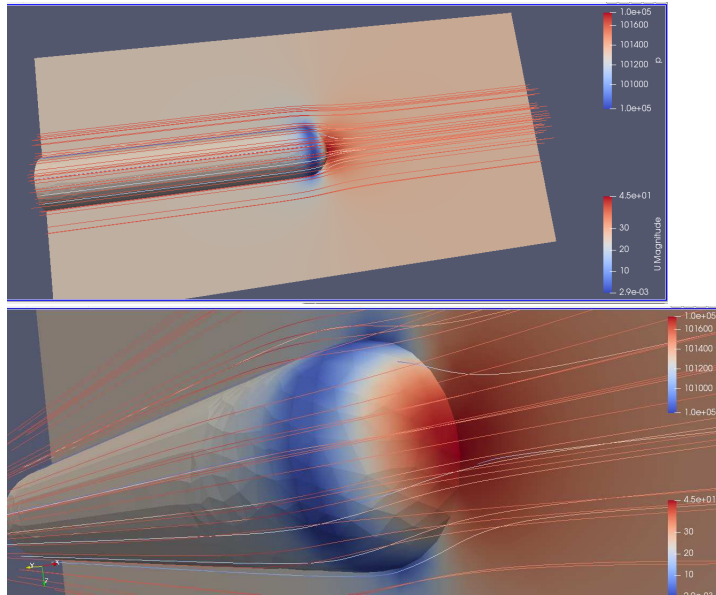
D. Force coefficients

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:

```
{ type forceCoeffs;
functionObjectLibs ( "libforces.so" );
outputControl timeStep;
timeInterval 1;
log yes;
patches ( body );
rhoName rhoInf; // Indicates incompressible
rhoInf 1.225; // Redundant for incompressible
liftDir (0 0 1);
dragDir (1 0 0);
CofR (0 0 0); // Axle midpoint on ground
pitchAxis (0 1 0);
magUInf 40;
lRef 1; // Wheelbase length
Aref 10; // Estimated
}
```

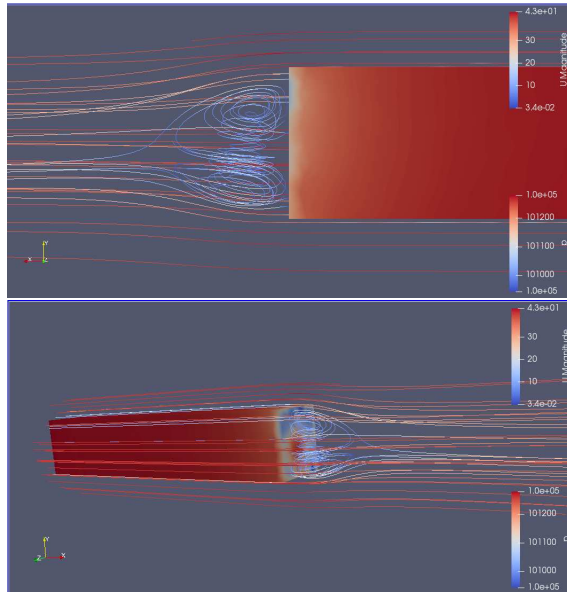
III. Result and Discussion

We can visualize the pressure variation over the Curved Body from the pressure contour(Figure 4). As can be seen, the pressure around the curved surface while the fluid is flowing around is creating a lower pressure region.



IV. AirFlow over edgy surface

If we replace the curved surface with plane surface the fluid flow around the edges is directly flow in the fluid flow direction .it does not attached itself to the surface as comparsion to curved surface.



REFERENCES

- Caroline Lubert, On some recent application of coanda effect, International journal of Accoustic and vibration, Vol 16, No.3, 2011

- https://wikipedia.org/wiki/Coanda_effect
- <https://openfoamwiki.net>