International Journal of Latest Engineering Research and Applications (IJLERA) ISSN: 2455-7137

Volume – 01, Issue – 06, September – 2016, PP – 01-06

Computation and Testing the Potential Flow Surrounded Cylinderical by Using OpenFOAM

Hoai Nguyen[1], Le Chau Thanh Nguyen[2] , Minh Man Pham[3]
[1,2,3]*Department of Mechanical Engineering, College of Technology, University of Danang, Viet Nam*

Abstract: In fluid dynamics, the study of the potential flow plays an important role for aerofoils, water waves, electroosmotic flow, and groundwater flow. There are many closed-source software packages used to model as well as simulate the fluid behavior, and OpenFOAM is one of the most powerful tools which many researchers prefer. In this paper, OpenFOAM is therefore carried out to model potential flow surrounded cylinderical with the purpose of understanding the dynamics structural of the potential flow and their interactions. Also, at-two-dimensional, time accurate and finite-volume-based methodology is conducted to perform the computations. This solution indicates a net zero drag on the body of circular cylinder, pressure to achieve the maximum value on the surface of the cylinder, and it achieve the minimum value on the sides of the cylinder. Conversely, the velocity to achieve the maximum value on the sides of the cylinder, and it achieve the minimum value on the surface of the cylinder. The physical mechanisms of these phenomena are analyzed, is an excellent vehicle for the study of concepts that will be encountered numerous times in mathematical physics, such as vector fields, coordinate transformations, and most important, the physical interpretation of mathematical results.

Keywords: A Computational Fluid Dynamic, Finite Volume Method, Incompressible Flow, Numerical Simulation, OpenFOAM,Potential Flow

I. INTRODUCTION

1. OpenFOAM

Computational fluid dynamics (CFD) software has been developed to become an essential instrument for aerodynamics. This software concentrates on high usage of time as well as energy and specially, the high cost assocated with operating the commercial CFD based programs. This causes a problem that only big companies can afford to utilize these commercial CFD programs. If we want to spread or create new concepts from the available commercial codes, it will entail a higher cost in response. Therefore, it has led the researchers to advance their own codes or to use the existing open source codes [3]. As an open source code, OpenFOAM is used in this project, and the purpose here is to compare the outcomes of the results in OpenFOAM in contrast to the well-known certified.

OpenFOAM is a free, open source CFD software package, licensed and distributed by the OpenFOAM Foundation [6] and developed by OpenCFD Ltd [7]. This software package is utilized broadly in academic and industry to solve the issues ranging from complex fluid flows involving chemical reactions, turbulence, and heat transfer, to solid dynamics and electromagnetics [1-4]. Detailed documentation supporting solvers made using the set of libraries is provided by OpenFOAM [6]. Furthermore, OpenFOAM allows users to easily intervene in the program with many options for the parameters with Linux operating system [5] and programming languages C++ . Besides, it uses the finite volume method and the SIMPLE algorithm as solution procedure.

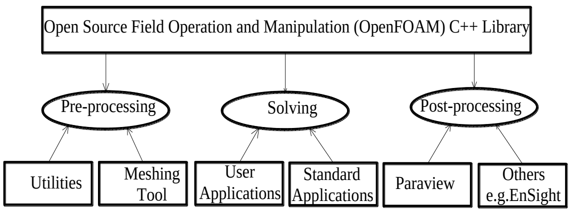

Figure no.1: Overview of OpenFOAM structure [7]

2. Potential Flow Theory

In mathematics, potential flow around a circula cylinder is a classical solution for the flow of an inviscid, incompressible fluid around a cylinder that is transverse to the flow [8].

A cylinder of radius R is placed in two-dimentional,incompressible, invisid flow. The goal is to find the steady velocity vector $\vec{V}$ and pressure p in a plane, subject to the condition that far from the cylinder the velocity vector is:

$$\vec{V} = U\hat{\imath}+0.\hat{\jmath} \quad (1.1)$$

where U is constant, and at the boundary of the cylinder

$$\vec{V}.\hat{n} = 0 \quad (1.2)$$

where $\hat{n}$ is the vector normal to the cylinder surface. The upstream flow is uniform and has no vorticity. The flow is inviscid, incompressible and has constant mass density ρ. The flow therefore remains without vorticity, or is said to be irrotational, with $\nabla\times\vec{V} = 0$ everywhere. Being irrotational there must exist a velocity potential ϕ :

$$\vec{V} = \nabla\phi \quad (1.3)$$

Being incompressible, $\nabla.\vec{V} = 0$, so ϕ must satisfy Laplace's equation:

$$\nabla^2\phi = 0 \quad (1.4)$$

The solution for ϕ is obtained most easily in polar coordinates r or ϕ, related to conventional Cartesian coordinates by x= r cosϕ and y =r sinϕ. In polar coordinates, Laplace's equation is:

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial\phi}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2\phi}{\partial\theta^2} = 0 \quad (1.5)$$

The solution that satisfies the boundary conditions is:

$$\phi(r, \theta) = U(r+\frac{R^2}{r}) \cos\theta \quad (1.6)$$

The velocity components in polar coordinates are obtained from the components of ∇ϕ in polar coordinates:

$$V_r = \frac{\partial\phi}{\partial r} = U\left(1-\frac{R^2}{r^2}\right)\cos\theta \quad (1.7)$$

And

$$V_\theta = \frac{1}{r}\frac{\partial\phi}{\partial\theta} = -U\,(1+\frac{R^2}{r^2})\sin\theta \quad (1.8)$$

Being invisicid and irrotational, Bernoulli' s equation allows the solution for pressure field to be obtained directly from the velocity field:

$$p = \frac{1}{2}\rho(U^2 - V^2) + p_\infty \quad (1.9)$$

where the constants U and p_∞ appear so that $p\rightarrow p_\infty$ far from the cylinder, where V=U. Using

$$V^2 = V_r^2 + V_\theta^2, \quad (1.10)$$

$$P = \frac{1}{2}\rho U^2\left(2\frac{R^2}{r^2}\cos(2\theta) - \frac{R^4}{r^4}\right) + p_\infty \quad (1.11)$$

II. NUMERICAL METHOD

1. Finite Volume Method

The Finite Volume Method (FVM) is one of the most versatile discretization techniques used in CFD. Based on the control volume formulation of analytical fluid dynamics, the first step in the FVM is to divide the domain into a number of control volumes (aka cells, elements) where the variable of interest is located at the centroid of the control volume. The next step is to intergrate the differential form of the governing equations over each control volume. Interpolation profiles are then assumed in order to describe the variation of the concerned variable between cell centroids. The resulting equation is called the discretized or discretization equation. Figure no 2 represents finite volume discretization.

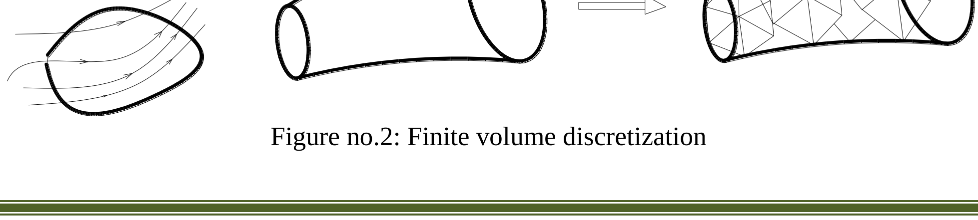

Figure no.2: Finite volume discretization

OpenFOAM is an objected-oriented C++ framework that can be used to build a variety of computational solvers for problems in continuum mechanics with a focus on finite volume discretization. OpenFOAM also includes several ready solvers, utilities, and applications that can be directly used. At the core of these libraries are a set of object classes that allow the programmer ro manilulate meshes, geometries, and discretization techniques at a high level of coding.

Tables 1.1, 1.2, 1.3, 1.4 present a list of the main OpenFOAM classes and their functions

Table 1.1 Numerics and discretizations

Object	Type of data	OpenFOAM Class
Interpolation	Differencing schemes	surfaceInterpolation<template>
Explicit discretization: differential operator	ddt, div, grad, curl	fvc::
Implicit discretization: differential operator	ddt, d2dt2, div, laplacian	fvm::

Table 1.2 Computational domain

Object	Type of data	OpenFOAM Class
Variable	Primitive variables	Scalar, vector, tensor
Mesh components	Point, face, cell	Point, face, cell
Finite volume mesh	Comutational mesh	fvMesh, polyMesh
Time	Time database	Time

Table 1.3 Field operation

Object	Type of data	OpenFOAM Class
Field	List of values	Field< template>
Dimentions	Dimension set up	DimentionSet
Variable field	Field+mesh+boundaries+dimension	GeometricField<template>
Algebra	+, -, pow, =, sin, cos,…	Field operators

Table 1.4 Linear equation systems and linear solvers

Object	Type of data	OpenFOAM Class
Sparse matrix	Matrix coefficients and manipulation	IduMatrix, fvMatrix
Iterative solvers	Iterative matrix solver	IduMatrix::solver
Freconditioner	Matrix precondioner	IduMatrix::preconditioner

$$\underbrace{\frac{\partial}{\partial t}(\phi)} + \underbrace{\nabla.(v\phi)} = \underbrace{\nabla.(D\phi\nabla\phi)} + \underbrace{P\phi - C} \quad (1.12)$$

unsteady team convection term diffusion term source and sink term

is basically written in OpenFOAM as Listing 1.1

```
(
    fvm::ddt(phi)
  + fvm::div(mDot, phi)
  - fvm::laplacian(Dphi, phi)
 ==
    fvm::Sp(P,phi)
   -fvc::(C)
);
```

Listing 1.1 Scipt for solving a simple transport equation using OpenFOAM

2. Typical Potential Flow Around a Cylinder

2.1. Problem

Fluid flow moving with a velocity of 1m /s through a cube domain, with 1 hole 1 meter diameter cylindrical drilled in the center. Based on the given parameters, test simulation on OpenFOAM, the distribution of velocity, pressure, ... the flow around cylindrical hole.

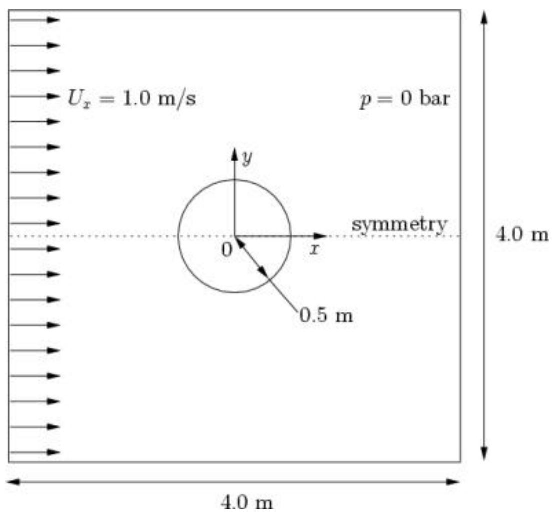

Figure no.3 Flow around a cylinder problem

2.2. Boundary conditions

Inlet:

U_x = 1m/s,

$$\frac{\partial p}{\partial n} = 0$$

Oulet:

p =0Pa

$$\frac{\partial U}{\partial n} = 0$$

2.3. Initial conditions

U=1 m/s, p= 0Pa

III. RESULT AND DISCUSSION

1. Grid Generation

To facilitate the observation, air flow motion is simulated 3D space domain, thereby changing the shape of the 2D grid is not required.

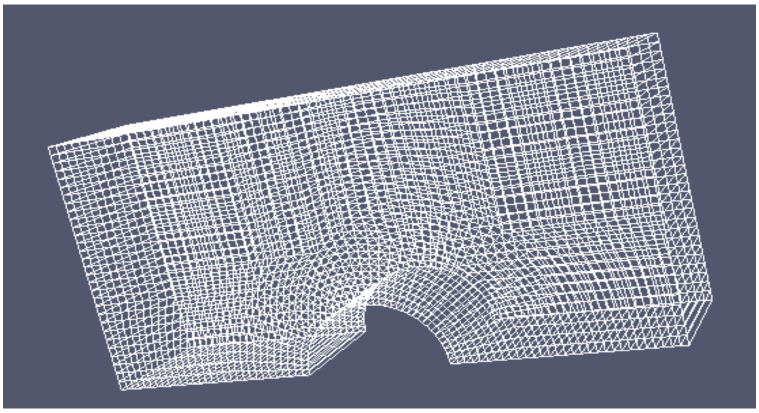

Figure no.4: Grid simulation on OpenFOAM

The resulting structure is presented on Figure no.4. The computational mesh required is created by using the OpenFOAM utility blockMesh after performance necessary edition to the file blockMeshDict. Thus, the geometry is formed based on corner points in a quadrilateral block which is meshed with hexahedral elements.

The number of cells in x-, y- and z- direction are defined in the entries. The boundary faces are defined under section patches. Names and types of patches are also defined. With the elevation of the number of mesh or grid resolution the solution fields for different variable pressure, velocity tends to converge to the exact analytical solution.

2. Velocity of FlowAround a Cylinder

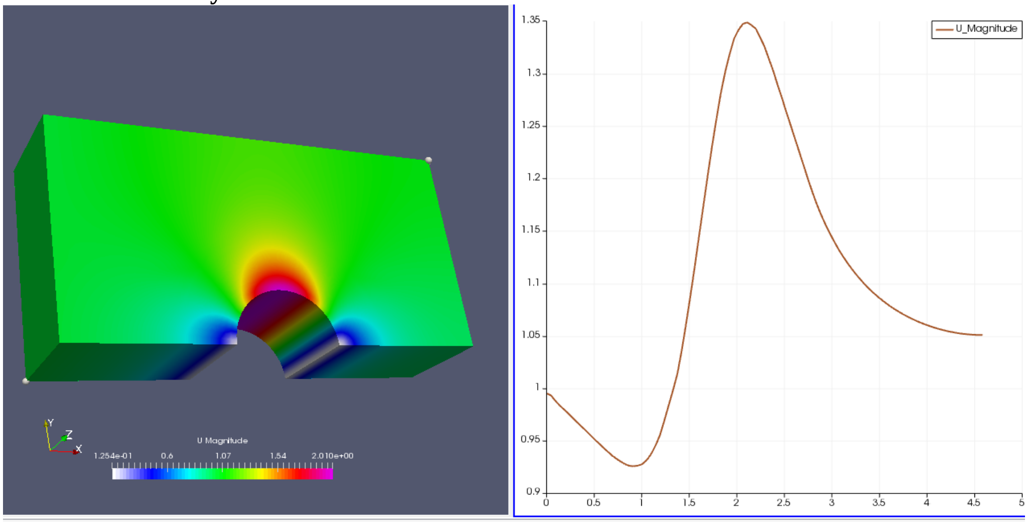

Figure no.5: Velocity of flow around a cylinder simulation

Velocity of flow achieves maximum value at vertex position on top of the cylinder (2 m/s) and reached the lowest value at the lower edge of cylindrical (0m/s). This is completely consistent with the theoretical basis of potential flow around a cylinder problem.

3. Pressure of Flow Around a Cylinder

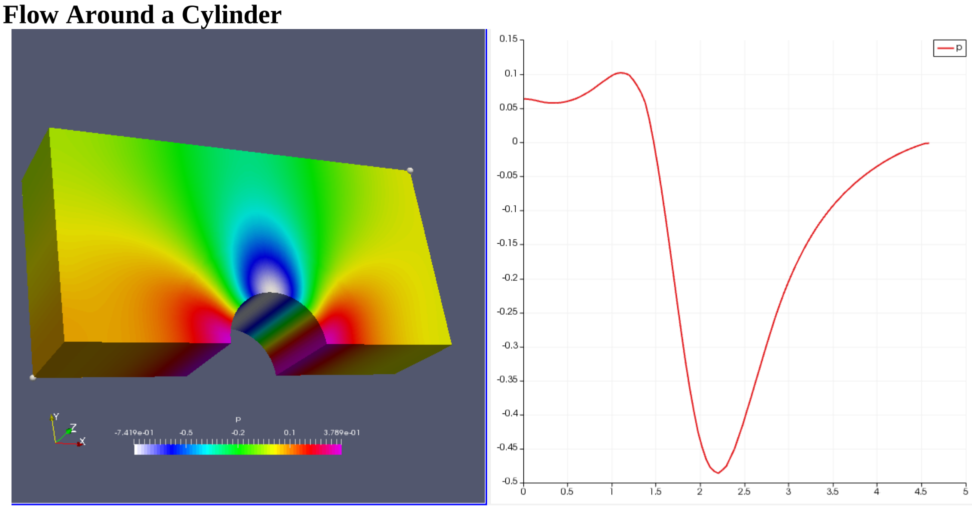

Figure no.6: Pressure of flow around a cylinder simulation

Contrary to the flow velocity, pressure of flow achieves minimum value at vertex position on top of the cylinder (0Pa) andreached the maximum value at the lower edge of cylindrical(1Pa). This is completely consistent with the theoretical basis of potential flow around a cylinder problem.

4. Velocity Potential Around a Cylinder

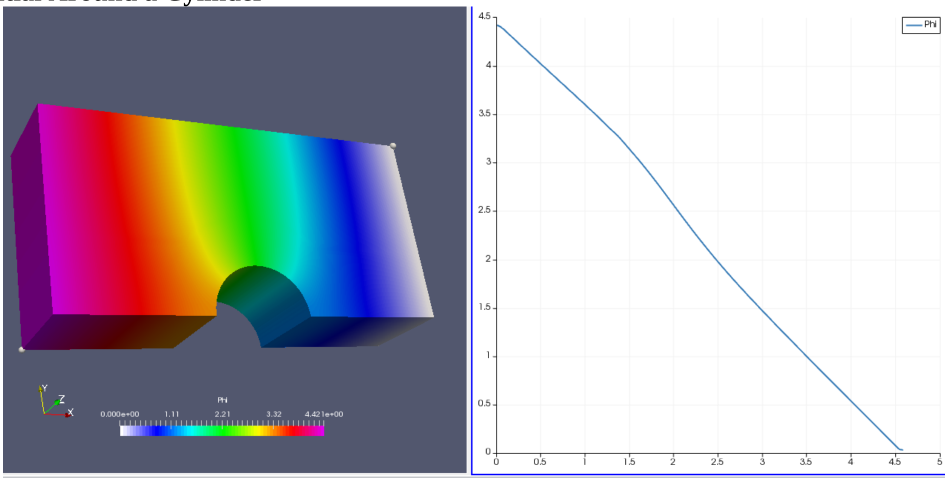

Figure no.7:Velocity potential around a cylinder simulation

IV. CONCLUSION

In this paper, the numerical simulation of one basic problem that the potential flow around the cylinder was conducted. The main goals are to predict flow fields, for instance: the outer flow field for aerofoils, water waves, electroosmotic flow, and groundwater flow. Simulation results also indicate that the distribution of velocity and pressure of the potential flow through the cylinder that is not steadily. The pressure will be lower on the wake side of the cylinder, than on the upstream side, resulting in a drag force in the downstream direction.

OpenFOAM package offers a suitable alternative to commercial software, but for daily use it is necessary to prepare some templates for the standard cases and to have some useful pre-processing tool. A few tests with the meshing tool blockMesh. This seems to be one of the pros of the OpenFOAM package. Although the basic and advanced problems of external arrodynamics can be solved easily, there are some task to be done in future. About all it is the endless work finding a suitable potential flow theory and the improvement of the settings of the mesh generation.

Acknowledgements

The authors are grateful to College of Engineering, The University of Danang, Viet Nam and Chinese Culture University, Taiwan for having made this work possible. The authors also appreciate the technical support of Prof.Tsing-Tshih Tsung, Department of Mechanical Engineering, Chinese Culture University, Taiwan.

REFERENCES

[1] Gomez, Sebastian, Verification of Statistical Turbulence Models in Aerodynamic Flows, *Doctoral dissertation*,2014

[2] Adam Kosík, The CFD simulation of the flow around the aircraft using OpenFOAM and ANSA, *Proceedings of 5th ANSA &μETA International Conference*, Greece,2013

[3] UCSMHD Millewa , PM Senathilaka, WPAC Dayarathna, SA Samarasingha and SLMD Rangajeeva, Validation of OpenFOAM as Computational Fluid Dynamics (CFD) Software, *Proceedings of 8th International Research Conference*, KDU, 2015.

[4] Hrvoje Jasak, Dynamic Mesh Handling in OpenFOAM, *Proceeding of the 47th Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition*, Orlando, 2009

[5] M.J Churchfield and P.J. Moriarty, G. Vijayakumar and J.G. Brasseur, Wind Energy-Related Atmospheric Boundary Layer Large-Eddy Simulation Using OpenFOAM, *National Renewable Energy Laboratory*, Golden, CO, Report No. NREL/CP-500-48905,2010

[6] OpenFOAM Foundation. Features of OpenFOAM. Retrieved from http://www.openfoam.org/features/

[7] OpenFOAM Foundation.The open source CFD toolbox. Retrieved from http://www.openfoam.com/about/

[8] https://en.wikipedia.org/wiki/Potential flow around a circular cylinder

