ISSN: 0974-5823 Vol. 7 No. 10 October, 2022

International Journal of Mechanical Engineering

CFD Simulation of Aero-Acoustic Flow Analysis Across Bluff Body

Onkar Pravinrao Ajegaonkar [1]

[1] PG student of M.Tech. Mech-Thermal Engineering, School of Mechanical Engineering,
Dr. Vishwanath Karad MIT World Peace University, Pune, India.

Prof. (Dr.) Atul M. Elgandelwar [2]

[2] Associate Professor School of Mechanical Engineering,
Dr. Vishwanath Karad MIT World Peace University, Pune, India.

Email : [1] onkarajegaonkar@gmail.com, [2] atul.elgandelwar@mitwpu.edu.in

***Abstract* -** For numerous years, scientists have focused on unsteady flow around bluff bodies such as circular cylinders, spheres etc. Turbulent flow around a circular cylinder is of tremendous importance for understanding and furthering research into a wide range of basic and practical fluid mechanics problems. To anticipate flow noise, a software-based simulation of flow around a circular cylinder was performed in this study. This paper's goal is to investigate these practically and methodologically actual problems: to develop, verify, and validate the numerical method based on Large Eddy Simulation (LES) and Ffocus – Williams and Hawking's (FWH) models, which will be able to predict unstable periodic transient flow parameters around the cylinder and simulate the sound generated primarily by the vortex shedding behind cylinder accurately. Using existing research papers, we have used the different velocities and simulated the acoustic results for the sound identification. Two types of major noises are present which are Structure born noise, Air born noise of which this paper focuses on the Air born noise. The simulation tool, Star CCM+ was used to evaluate the above mention parameters considering that the previous researchers have not used this software for the simulation. The fully 3D turbulent incompressible flow around a cylinder and in its wake at a Reynolds number (Re) in the range of Re = 2.3×10^3 to 4×10^5 based on the cylinder diameter and Mach number M = 0.1 is calculated using Large Eddy Simulations (LES). Encouraging results are found in comparison to experimental data for the fluctuating Drag and Lift coefficient..

INTRODUCTION

Aerodynamically bluff bodies are frequently employed in a variety of engineering applications and are well known to be sources of unwelcome noise when included into fluid streams. Examples include the external mirrors on automobiles, the landing gear on airplanes, the antennae on airplanes and submarines, the blockages in air ducts, and a wide range of architectural design elements. Given their significance to so many different industries, it is crucial to gain a fundamental understanding of the Mechanism that causes aerodynamic bluff body noise and, secondly, to provide engineers with a computationally efficient way to calculate this noise so they can quickly develop new, quiet technologies. The flow over a circular cylinder is analogous to bluffest body flows. Boundary layers form on the upstream surface of the cylinder and separate when they encounter adverse pressure gradients near the top and bottom of the cylinder, forming two free shear layers. The free shear layers form behind the cylinder, become unstable, and eventually form the von Karman vortex street, a constant stream of vortices of alternate rotation that persists many diameters downstream of the cylinder.

Turbulence around a circular cylinder is extremely interesting for studying and developing investigations into a wide range of fundamental and practical fluid mechanics issues. This flow is driven by extremely complex physical processes such as fluid velocity due to an undesirable differential pressure, turbulence conversion, or vortex shed due to contact. This interaction is classified by a von Karman vortex roadway, which is a chain of vortex sheds alternately from the top and bottom sides of the cylinder. The fluid process results in an unbalanced force acting on the cylinder. Aeolian sounds are produced, while turbulent wake flow produces broadband sound. Acoustic noise produced by a flow can be caused by a variety of mechanics, but it is ultimately caused by fluctuations in the flow. These fluctuations will cause acoustic sources to be distributed throughout the flow. Local stress fluctuations in the flow (e.g., Reynolds stresses, viscous stress effects, and non-isentropic effects all act as quadrupole sources); pressure fluctuations at walls (e.g., dipole sources at solid boundaries); mass and heat fluctuations (e.g., distributed monopole sources); and external fluctuating force fields all contribute to noise. In this study, noise is produced when the fluid passes over the cylinder on the flow field, and it is discovered using CAA Methods. First, the flow is validated with different velocities by calculating drag coefficients and then with the existing research paper. Following the validation of the work, we proceed to the acoustics. This paper explains how much sound (dB) is generated by different velocities. Various methods or turbulence models, such as the Reynolds transport equation, Navier-Stokes equation, Shear Stress Transport Turbulence Model, K-Omega Turbulence Model, and K-epsilon Turbulence Model, are used for acoustic simulation. For flow validation, the K-Omega Turbulence model and LES Models are used in this paper for acoustic simulation.

Aeroacoustic Flow : 1] Acoustics, which includes all multiphasic fields concerned with the production, transmission, and detection of sound signals, is merely the physics of sound. 2] The term "sound “includes the transmission of sound over a different medium, such as air. This could be the propagation of mixed waves in porous materials, pressure waves in liquids, or elasticity waves in solids (vibrations).

1.2 the significance of aeroacoustics 1] Understanding atmospheric and weather events requires an understanding of aerodynamics. Aeroacoustics is the study of noise brought on by turbulent fluid motion or by aerodynamics interacting with a surface, and it is important to lessen noise and vibration. "Numerical Investigation of Flow Control Using Vertex Generator for Landing Gear Noise Reduction,"by A. P. Hao et al (2016). [1] The goal of this work is to investigate flow management methods using vortex generators (VGs) in order to lessen the noise produced by high Reynolds number flow over bluff bodies. In order to compute the flow field and supply surface pressure samples to an Acoustic FEM solver,a three-dimensional delayed detached eddy simulation method is used. A basic model of a circular cylinder and a configuration of a vortex generator using the basic cylinder are two geometries that are examined. Aeroacoustics Analysis for a Flow over 2D Cylinder by Suhas A. Kwoshik (2018). [2] For lower Mach numbers (0.17 - 0.35), a two-dimensional (2-D) unsteady flow past a circular cylinder has been numerically explored. The major goal of this study is to employ Computational Aero-Acoustic (CAA) models to capture the flow and acoustic fields for flow past a circular cylinder in a domain. The Ffowcs- Williams & Hawking (FW-H) model's flow field and acoustic field results are contrasted with those from the Broadband noise source (BNS) model. Along with the sound pressure level (SPL), flow information including velocity distribution, turbulent intensity, and results for the acoustic power level are reviewed. A numerical investigation of transient flow around a cylinder and aerodynamic sound emission was conducted by J.-C. Cai et al (2017).[4] It is exceedingly challenging to forecast the sound output of fluid flows because of the nonlinearity of the governing equations. In high-speed flows, when the nonlinear inertial terms in the equation of motion are significantly bigger than the viscous terms, this

Sound production frequently happens (high Reynolds numbers). One of the sources of the sound is found to be the variation between the real flow and the reference flow. Lighthill put up this notion and referred to it as an analogy. The formal solution of integral equations is a second crucial aspect of Lighthill's theory. "Two and Three-Dimensional Simulation of Sound Generated by Flow around a Circular Cylinder," by Reinaldo Orselli et al (2009). [5] This work uses the Lighthill acoustic analogy to compute the far-field sound produced by low Mach number flow around a two-dimensional and three-dimensional circular cylinder in the subcritical regime. Unsteady Reynolds-averaged Navier-Stokes models are used to estimate the time-dependent incompressible flow in the case of the two-dimensional example. Solving the filtered Navier-Stokes equations of the Large Eddy Simulation model yielded the flow for the three- dimensional scenario.

GOVERNING EQUATIONS AND BOUNDARY CONDITIONS

Flow past a circular cylinder has been simulated by solving numerically the unsteady Navier-Stokes equations for an incompressible fluid in 3D geometry. The equations for continuity and momentum may be expressed as follows:

Continuity: $$\frac{\partial\rho}{\partial t}+\nabla.(\rho V)=0$$

Momentum: $$\frac{\partial(\rho v)}{\partial t}+\nabla.(\rho uV)=-\frac{\partial\rho}{\partial x}+\frac{\partial\tau_{xx}}{\partial x}+\frac{\partial\tau_{yx}}{\partial y}+\frac{\partial\tau_{zx}}{\partial z}+\rho f_x$$

$$\frac{\partial(\rho v)}{\partial t}+\nabla.(\rho vV)=-\frac{\partial\rho}{\partial z}+\frac{\partial\tau_{xy}}{\partial x}+\frac{\partial\tau_{yy}}{\partial y}+\frac{\partial\tau_{zy}}{\partial z}+\rho f_y$$

Fig Shows the Boundary conditions.

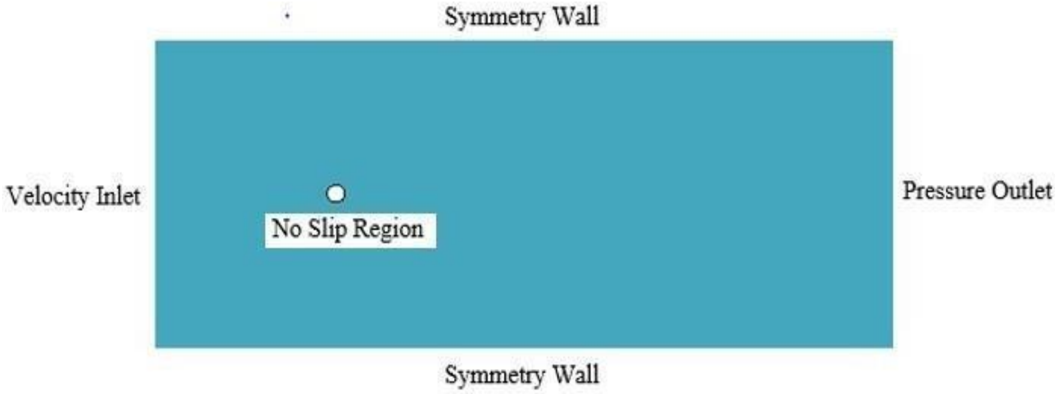


Fig. 2: Rectangular Computational Domain and Boundary Conditions

Entry into the computing area is considered as a Velocity Inlet. Outlet of the computing area is considered as a Pressure Outlet. Upper and Lower Sides of the walls are considered as a symmetry walls. And the cylinder placed in the middle of the rectangular computational domain is consider as a no slip Region (u=v=0).

The dimensions considered for this geometry are:

- cylinder diameter (D) = 0.1 m (100 mm)
- Length (L) = 70 D

NUMERICAL DETAILS

The grid used the cylinder configuration is shown in fig.

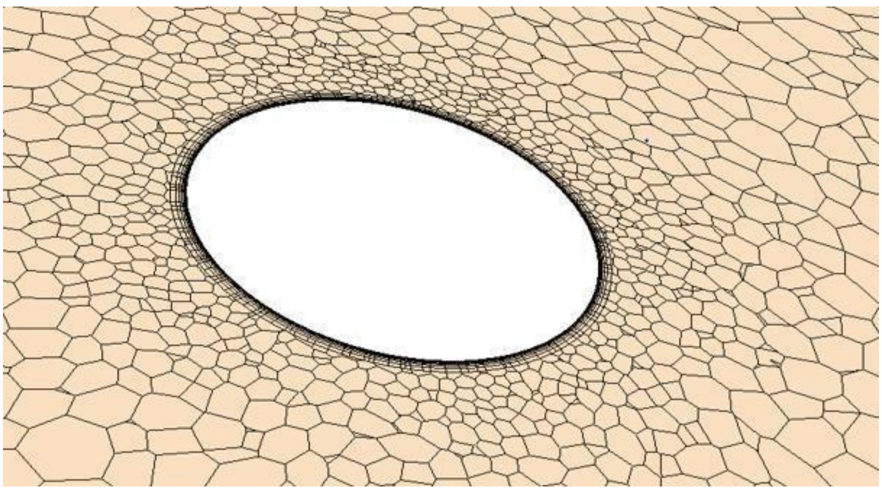


Fig. 3: Mesh used for computation

A high resolution unstructured polyhedral mesh is used with 928993 cell count is used for the simulation purpose using Star CCM+ Software. The major benefit of polyhedral mesh is that, each individual cells has many neighbors, so gradient can be well approximated. Polyhedrons are more sensitive to stretching that tetrahedrons which result in better mesh quality leading to improved numerical stability of the model. For the better capture of the Wall boundary layer effects 20 Prism layers are created around the cylinder. High performance computing is preferred for faster execution (HPC). 3Dimensional and unsteady flow is considered for the analysis. An Incompressible SIMPLE Finite volume code is used. Second order upwind scheme is used for discretization of space and time. A constant time step of t = 0.005 is used for Calculation.

In Star CCM+ Software, Lighthills Acoustic Analogy is applied for CAA Model to find the flow induced noise. Star CCM+ Computes unsteady pressure fluctuations and accordingly generated noise is resulted. FWH Ffocus – Williams and Hawkings acoustic integral formulation is preferred for Far Field Noise prediction such as overhead aircraft noise. The FWH Ffocus – Williams and Hawkings model is used only to predict the propagation of sound in free space. It does not include any effects such as reflection, refraction of sound. This model is used for the transient simulation. BNS Broadband Noise Source model allows acoustic source to be estimated based on the results of steady state simulations. In this paper both the BNS and FWH acoustic models are used to study the acoustic field.

RESULT AND DISCUSSION

Flow induced noise is becoming a major issue in a different industry such as automotive, aerospace. In this paper the Implementation of the CAA Computational Aero-acoustic Approach is used to investigate

The flow induced noise. For this purpose, various turbulence models are used for simulation purpose.so there are three types of sources of the flow noise namely Monopole, Dipole and Quadrupole.

Monopole noise source is occurring when there is fluctuations in the mass flow. Dipole noise source is occurring on the cylinder where the flow causes fluctuations in the pressure. Quadrupole noise source is occurring due to turbulent wakes.

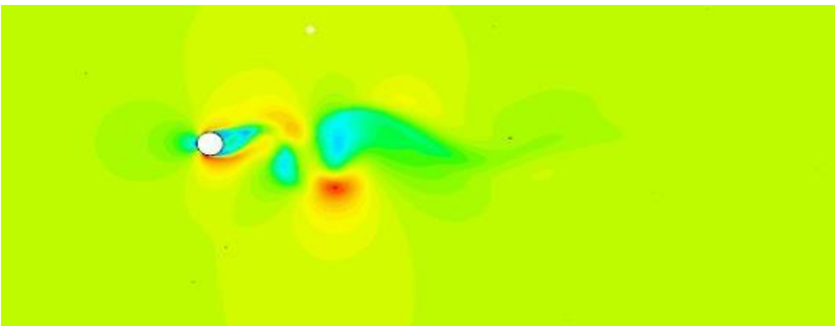
For flow over cylinder noise is generated is due to following reasons such as,

1) Turbulent Boundary Layer.
2) Vortices form in the wake region.
3) Flow separation behind the cylinder. etc.

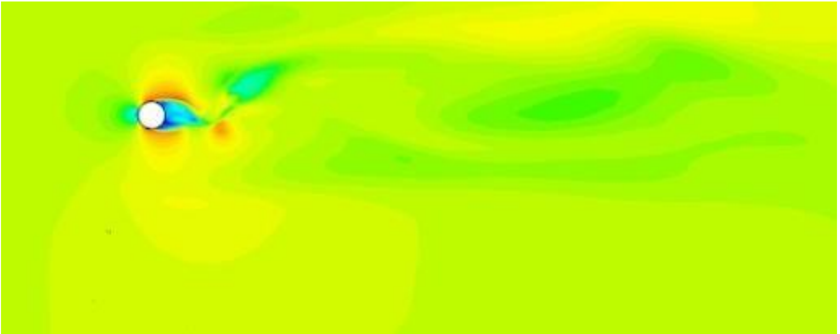
Two approach theories are used for the simulation purpose to evaluate the acoustic parameters.

Firstly, flow is validated with respect to the drag and lift coefficient values with the help of existing research paper.

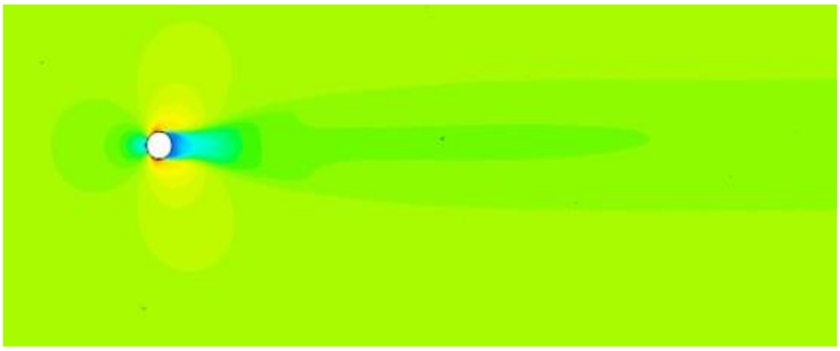
Secondly: different velocities are taken from existing research paper which is used for the acoustic analysis. The figure below illustrates how the flow changes with speed at a different Mach and Reynolds number.



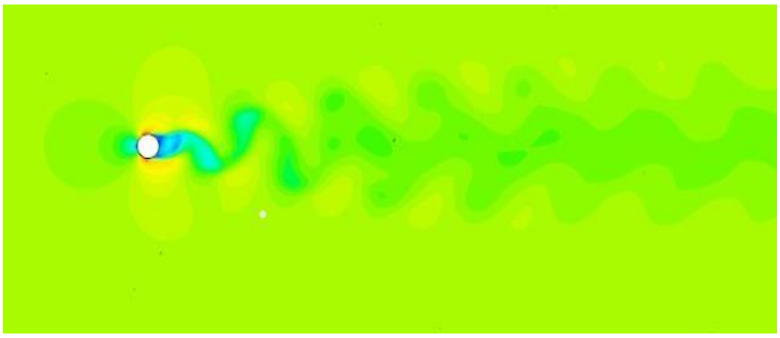
V = 0.35 m/s



V = 3 m/s



V = 21 m/s



V = 45 m/s

Table 4: Experimental and CFD Results validation table

Sr. no	Experimental Values of Drag (Cd)	Velocity (m/s)	Reynolds Number (Re)	CFD Results of Drag (Cd)
1	1.05	0.35	2.3 x 10^3	1.04
2	1.02	3	2 x 10^4	0.96
3	0.90	21	1.4 x 10^5	0.87
4	0.45	45	3 x 10^5	0.40

In this table, the computational values match the experimental values as calculated by previous researchers within a margin of error. Hence, flow can be considered to be validated. As acoustic is related to the flow fluctuations and we have validated readings regarding the desired parameters, hence we can proceed with the acoustic analysis for sound identification.

In the steady state approach BNS model is used for the identification of the noise source across the cylinder. For this purpose, in the BNS model there are two sub models are used namely Curle noise source acoustic model and the Proudman noise source acoustic model is used.

4.1 Noise Source Model

Curle Noise Source Model evaluates the noise from a turbulent boundary layer flow over a solid body assuming isotropic turbulence and low Mach number. This model is typically used for air handling subsystems (AHS) application including rotating parts such as blower's, heat exchangers, mufflers, Ducts. The curle model represents dipole source of noise coming from the fluctuating surface pressure that results from the solid boundaries acting on the fluid. Specifically.

This model calculates the local contribution to the acoustic power per unit area of the body surface. The Curle model can be activated for steady and unsteady simulations with Reynolds-Averaged Navier-Stokes (RANS) turbulence models, which provide turbulence kinetic energy and wall shear stress. The acoustic power per unit surface can be reported in dimensional units (W/m^2) and in dB:

Where, Pref is the reference acoustic power.

$$SAP\ (dB) = 10 \log (SAP / P_{ref})$$

4.2 Proudman Noise source model

Proudman Noise source model evaluates the acoustic power per unit volume and the sound is from quadrupole such as areas around blower blades. The Proudman model assumes isotropic turbulence. Proudman considers the generation of noise by isotropic turbulence. In Proudman's model, high Reynolds model for turbulence in

$$AP = \alpha \rho_o (u3 / l)(u5 / co^5)$$

near incompressible flow, the acoustic power per unit volume is,

Where:

- α Is a constant related to the shape of the longitudinal velocity correlation.
- u Is the root mean square of one of the velocity components.Is the
- l longitudinal integral length scale of the velocity.
- ρ_0 is the far-field density.
- c_0 is the far-field sound speed

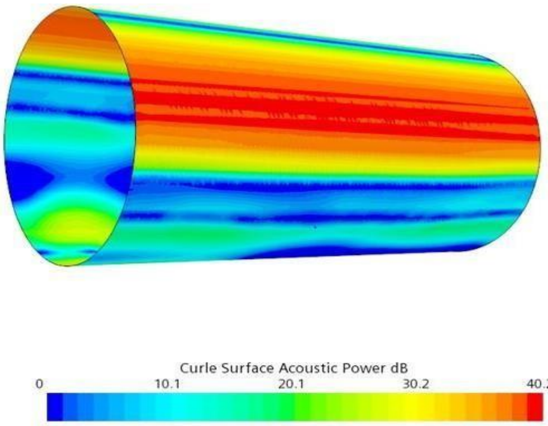


Fig 4.1 Curle Model (V=0.35 m/s)

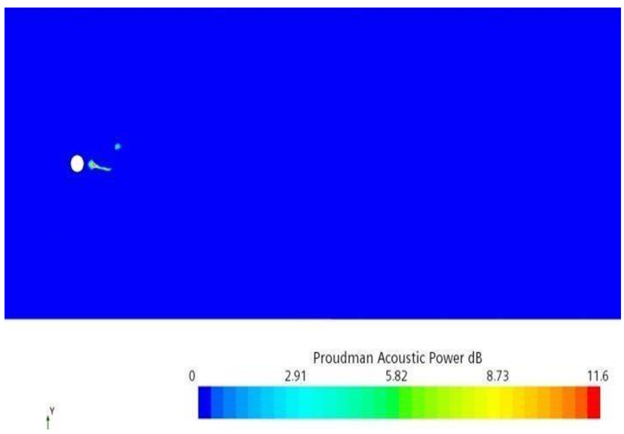


Fig 4.2. Proudman Model (V=0.35 m/s)

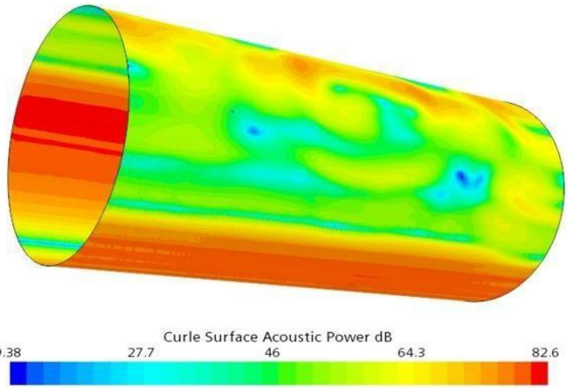


Fig 4.1 Curle Model (V = 21 m/s)

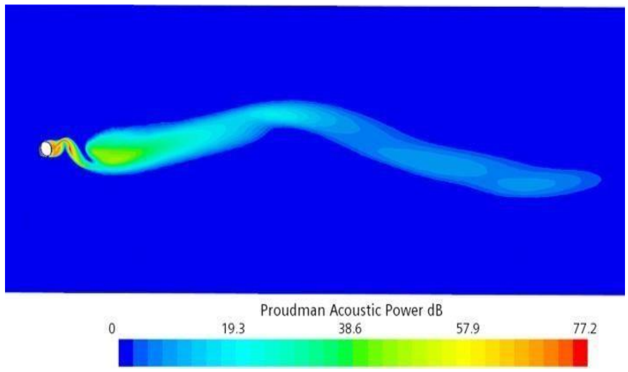


Fig 4.2. Proudman Model (V = 21 m/s)

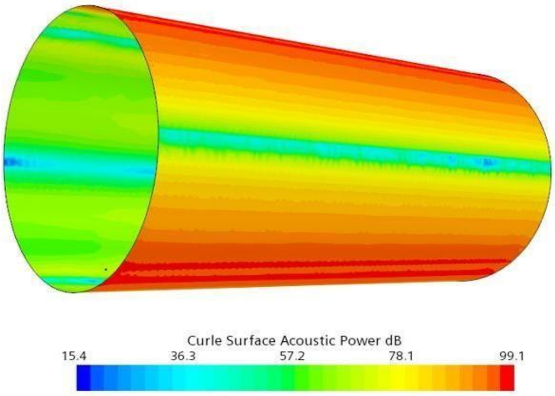


Fig 4.1 Curle Model (V= 45 m/s)

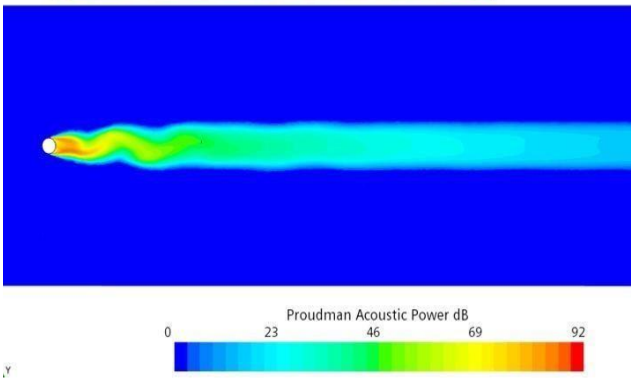


Fig 4.2. Proudman Model (V =45 m/s)

Above mentioned Fig.4.1 and 4.2 shows that the Curle and Proudman noise source models, for the acoustic simulation we have considered four different velocities with the help of existing research papers: the velocities are V= 0.35 m/s, V= 3 m/s, V= 21 m/s, V= 45 m/s.

For this purpose, we have to consider the physics model for the further analysis. So the Time Domain is considered as Implicit Unsteady. Turbulent viscous regime is used. For the calculation purpose Large Eddy Simulation (LES) or detached eddy simulation model (DES) is used. K-Omega Wall Treatment Is used for the further wall treatment.

4.3 Ffocus Williams and Hawking's (FWH) Acoustic Model

In this study, for the acoustic analysis FWH acoustic model is used. FWH acoustic integral formulation is the preferred strategy for far field noise prediction such as overhead aircraft noise. This model calculates the far field sound signal that is radiated from the near field flow data from a CFD solution. The goal is to predict small amplitude acoustic pressure fluctuations at the location of receiver. The FWH acousticmodel is the transient phenomenon is used only to predict the propagation of sound in free space.

In FWH analysis the data is in Time domain initially. For the spectral Analysis we have to convert the time domain data into frequency domain data i.e., Fast Fourier Transform (FFT). So, the obtained results will be in Sound pressure level (dB) with respect to the Frequency (Hz). (SPL Vs Frequency).

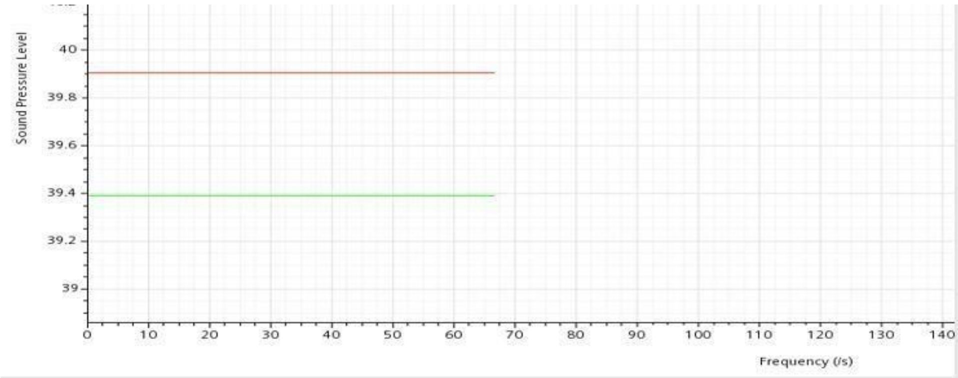


Fig.4.3 SPL Vs Frequency (V= 0.35 m/s)

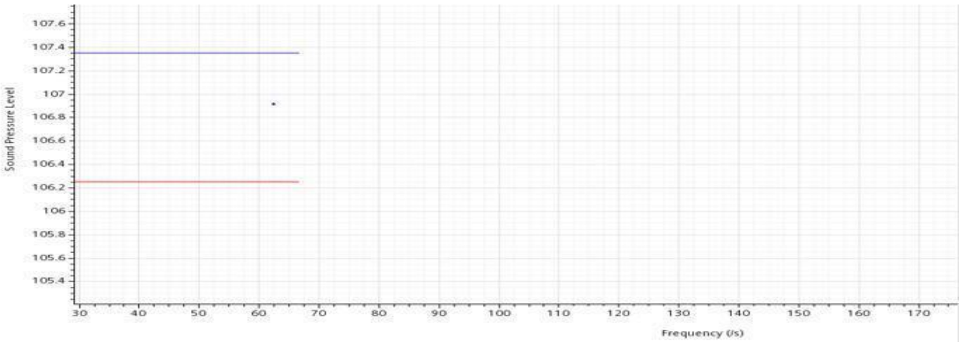


Fig.4.3 SPL Vs Frequency (V= 21 m/s)

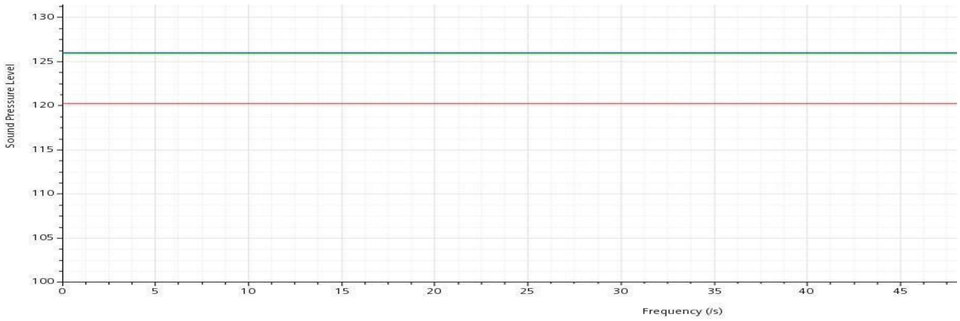


Fig. 4.3: SPL Vs Frequency (V= 45 m/s)

Table 4.3: CFD Acoustic Result table

Sr. No	Reynolds Number (Re)	Velocity (m/s)	Mach Number	Sound Generated (dB)
1	2.3 x 10^3	0.35	0.001	40
2	2 x 10^4	3	0.008	80
3	1.4 x 10^5	21	0.061	106
4	3 x 10^5	45	0.131	125

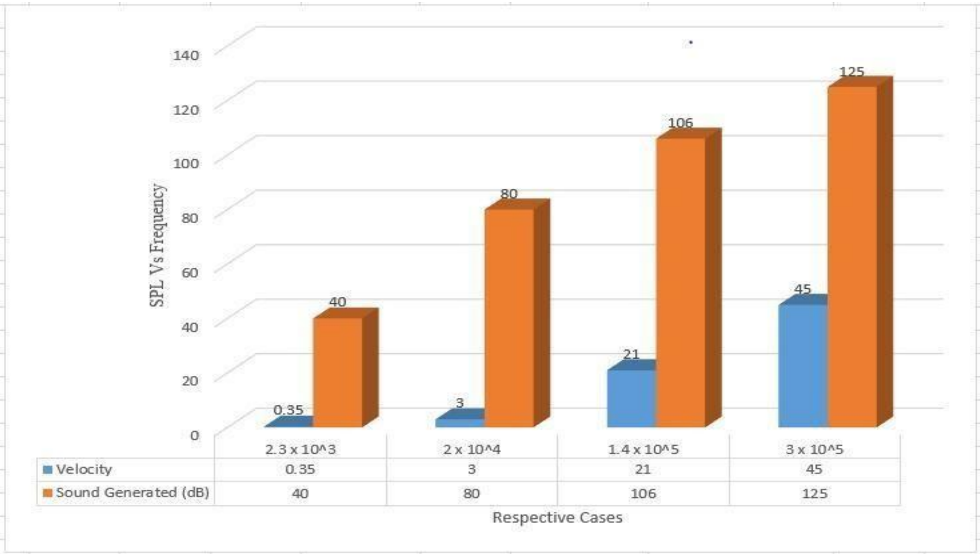


Fig. 4.3: SPL (dB) Vs Frequency (HZ) with respect to the velocities and Reynolds number

CONCLUSION

This simulation study helps to understand the noise source identification and the propagation of sound generated around a circular cylinder. As discussed in the article at 45 m/s velocity, 125 dB of sound is generated around the cylinder at Mach no 0.1. For this approach BNS acoustic model is used to identify the sound. And for the propagation of sound FWH acoustic model is used.

This study is carried out by following two different steps. In the first step, flow is validated with the help of existing research paper. In those experimental values are taken from the existing research paper and these values are validated with the CFD results with the help of Star CCM+ Software. In the second step, different velocities were selected from the existing research paper and based on this aero-acoustic flow analysis was done on the cylinder to identify the noise.

Finally, this investigation concludes that by properly capturing the flow physics with the help of respective turbulence model and by resolving these flow physics models we can identify the noise source and its propagation.

REFERENCES

[1] A. P. Hao et.al "Numerical Investigation of Flow Control Using Vertex Generator for Landing Gear Noise Reduction" (2016).

[2] Suhas A. kwoshik "Aeroacoustics Analysis for a Flow over 2D Cylinder" (2018). DOI: 10.13140/ RG.2.2.23689.67681.

[3] J.-C. Cai1 , J. Pan2 , A. Kryzhanovskyi1. A numerical study of transient flow around a cylinder and aerodynamic sound radiation* Thermophysics and Aeromechanics, 2018, Vol. 25, No. 3.

[4] A. Hirschberg∗ and S.W. Rienstra : An introduction to aeroacoustics : 18 Jul 2004

[5] .O. Reinaldo, M. Julio, and S. Fabio, Two and three-dimensional simulation of sound generated by flow around a circular cylinder, in: 15th AIAA/CEAS Aeroacoustics Conf. (30th AIAA Aeroacoustics Conference), AIAA, 2009−1097.

[6] C. Wagner, P. Sagaut, and T. Hüttl, Large-eddy Simulation for Acoustics: Introduction, Cambridge University Press, London, 2012.

[7] M. Lighthill, On sound generated aerodynamically. I. General theory, Proc. Roy. Soc. London. Series A, Mathematical and Physical Sci., 1952, Vol. 211, No. 1107, P. 564−587.

[8] J.D. Revell, R.A. Prydz, A.P. Hays, Experimental study of aerodynamic noise vs drag relationships for circular cylinders, AIAA J., 1978, Vol.16, No.9, P. 889–897.

[9] Pal Pandian and Anil Raj ‘Flow Noise Investigation for an Unsteady Flow Past a Circular Cylinder’ 2014, ISSN (ONLINE): 2321-3051

[10] Ivan Kološ, Lenka Lausová: Numerical Analysis of Flow Around a Cylinder in Critical and Subcritical Regime : Sustainability 2021, 13, 2048. https://doi.org/10.3390/su13042048

[11] K M Sowoud1 , A A AL-Filfily1 and B H. Abed1 : Numerical Investigation of 2D Turbulent Flow past a Circular Cylinder at Lower Subcritical Reynolds Number : 3rd International Conference onSustainable Engineering Techniques (ICSET 2020).

[12] W.K. Blake. Mechanics of Flow-induced Sound and Vibration, Volume I. Academic Press, Orlando, 1986.

[13] A.M. Cargill. Low-frequency sound radiation and generation due to the interaction of unsteady flow with a jet pipe. Journal of Fluid Mechanics, 121:59–105, 1982.

[14] A.P. Dowling and J.E. Ffowcs Williams. Sound and Sources of Sound. Ellis Horwood Publishers, Chichester, UK, 1983.

[15] J.E. Ffowcs Williams and D.L. Hawkings. Sound generated by turbulence and surfaces in arbitrary motion. Philosophical Transactions of the Royal Society of London, A264:321–342, 1969.

[16] H.H. Hubbard, editor. Aeroacoustics of Flight Vehicles: Theory and Practice. Volume 1 Noise Sources; Volume 2 Noise Control (Nasa Reference Publication 1258). Acoustical Society of America, 1995.

[17] A. Powell. Theory of vortex sound. Journal of Acoustical Society of America, 36:177–195, 1964.