NUMERICAL ANALYSIS OF FLOW INDUCED VIBRATIONS OF TUBES BANKS IN CROSS FLOW USING POROUS MEDIA

Rahul Dube[1], Akhilesh Parmar[2], Rahul Thakor[3], Surajsinh Vasadia[4], Chirag Mistri[5], Mayank Sompura[6]

[1,2,3,4,5]*Bachlor of Degree, Valia, Bharuch, Gujarat*
[6]*Professor, Mechanical Dept. Valia, Bharuch, Gujarat*

***

Abstract - *The Flow past a 2-D cylinder is one of the areas where extensive research has been carried out. Computational fluid dynamics (CFD) simulation has become a powerful tool for analysis of the Flow induced vibration characteristic cross flow over the circular Cylinder. The Flow induced Vibration is depends on to Overall Sound Pressure Level (OASPL), Strouhal Number, Pressure Drop and Frequency.*

The heat exchanger tubes are subjected to vibrations due to high pressure stream passing over it. The porous media is a material containing pores and generally characterized by the porosity. By using porous media in between cross flow tubes such induced vibrations and pressure drop can be reduced. In the present work, 2-D circular inline cylinder CFD models i.e. model- 1 (without porous media) and model-2 (porous media) were simulated and results were reported. The distance between two cylinders is (1.5D, 2.0D, and 2.5D) and thickness of the porous media was (0.25D, 0.5D, 0.75D and 1D).

In CFD model-2, significantly reduction in the velocity and OASPL then the model-1 is observed. The model-2 exhibits higher frequency then the model-1, thus Strouhal number increases with which the vibration over the tube significantly reduces. The pressure drop across the model-2 is less and more linear then model-1.

Key Words: **CFD, Cross Flow, porous media, OASPL, Strouhal Number,pressure drop**

1. INTRODUCTION

High Speed Compressible Flows past open cavities induced complex unsteady aerodynamic characteristics. The flow past a 2-D cylinder is one of the areas where extensive research has been carried out. It has numerous industrial and engineering applications such as in the design of speed trains, airplanes and road vehicles, cooling towers, oil and gas pipelines, tubular and compact heat exchangers, cooling of electronic components, and flow dividers in polymer processing applications and so on.

Boundary layer remains laminar from the stagnation point at the front of the cylinder to the point where it separates. The noise is generated due to various reasons such as vortices in the wake region, turbulent nature and so on. Most previous numerical investigations have focused only on the flow fields of the circular cylinders. So our interest turns towards. The technology based on CFD (computational Fluid Dynamics) Become capable of analyzing aerodynamic problems.

The prediction of acoustic fields on flow past a circular cylinder using CAA models in ANSYS Fluent 15. In this model, the acoustic source data is extracted from a transient Computational Fluid Dynamic (CFD) analysis based on an unsteady Shear stress transport (SST) turbulence model. Most of modern CAA models are based on this FW-H analogy [1]. In order to solve the flow over the cylinder more appropriately taking into account its 3D effects, a second three dimensional simulation was carried out using the filtered incompressible Navier-Stokes equations of the Large Eddy Simulation (LES) model.

An understanding of the physical mechanisms involved in sound generation is a crucial step for reducing or controlling the sound emission. The sound generated by a flow over a circular cylinder is simulated numerically. As a benchmark based on the experimental data of Revell *et al* [2], a flow with a Reynolds number $Re = 90{,}000$ and a free stream Mach number $M = 0.2$ was simulated. Most of noise is generated by the unsteady pressure fluctuations on the cylinder wall. Thus, the flow which is the source of noise can be computed separately from the acoustic field. Under this assumption, an acoustic analogy is employed to solve the acoustic field. The far-field sound is computed from integral solutions of the wave equation which uses as source term the flow field obtained from the CFD solution.

The Reynolds number is within the range of the Subcritical flow regime that is characterized by a laminar boundary layer separation with turbulence transition occurring downstream in the wake. Unsteady Reynolds Average Navier-Stokes (URANS) equations are solved using the FLUENT CFD code. Despite the fact of the flow being inherently 3D for $Re = 90{,}000$, a two-dimensional numerical simulation can capture the main physical aspects of the flow dominated by an alternate and periodical vortex shedding.

Nomenclature

D	Diameter of the tubes
L	Distance Between The two tubes.
t	Thickness of the Porous Media.
V	Velocity of the fluid
μ	The Dynamic Viscosity of the fluid.
ρ	Density of the fluid

1.1. CFD Conservation Equation

Commercial CFD code Fluent solve appropriate volume averaged conservation equation that given fluid flow for flow field of complex definition like flow through micro porous media. The external body force term F in momentum conservation equation accounts for viscous and initial losses of the fluid within the porous media and is defined by fluent as

$$F = -\left(\sum_{j=1}^{3} D_{ij}\mu v_i + \sum_{j=1}^{3} \frac{C_{ij}\rho v_{mag} v_j}{2} \right) \tag{1}$$

Where F is External body force term for the momentum equation, D is the viscous Resistance matrix, C is the inertial resistance matrix as defined by fluent. This external body force terms is the drag force imposed by the pore wall and contributes to the pressure drop across the regenerator matrix. It is proportional to the physical velocity of fluid v. therefore, for a homogeneous matrix with steady flow without internal axial body force, the axial pressure drops becomes,

$$\frac{\partial(\phi\rho)}{\partial x} = \frac{\partial}{\partial x}(\phi\rho v_i v_i) + \frac{\partial}{\partial x}(\phi\tau_i) - \left[D\mu v_i + \frac{c\rho v_{mag} v_i}{2} \right] \tag{2}$$

1.2. Preliminary efforts

Preliminary work has been studied to understand the process methodology of CFD ANSYS. The problem deals with a circular cylinder in the airflow in a 2D simplification. The free Stream velocity U = 69.20 m/s and the cylinder diameter is 1.9 cm. The domain Extends 2.5D in upstream and 17.5D in downstream direction and 2.5D above as well as below the cylinder. The domain is sketched in figure 1. The near field unsteady flow result is important to obtain the sound source and be used as the input data to the wave equation in order to obtain the acoustic far field. Thus the noise prediction depends directly on the accuracy of the flow results. The turbulence models of two dimensional circular cylinder flow field is Large Eddy Simulation (LES) model to obtain the time history of the near field flow over a circular cylinder at R_e=90,000.

In this set of numerical simulations, the boundary conditions are (I) no-slip and no penetration conditions on the cylinder surface, (II) a uniform free stream velocity with no perturbations at the inlet boundary, (III) a constant free stream pressure on the outlet boundary (IV) free-slip conditions on the upper and lower boundaries of the computational on the unstructured grids. The numerical solver in Fluent CFD software is used for the present study.

The noise of the cylinder flow field is resulting from the unsteady pressure alternative and fluctuation and periodic vortex shedding. The accuracy of the numerical simulations is generally evaluated by comparing it with available experimental and computational results. Microphone has located at 35D and 128D away from the circular cylinder axis and at angle of 90 degree from the cylinder stagnation point [2].

1.3. Grid Independent Study

Table: - 1 Grid Independent Test (Without Porous Media)

Element size	Nodes	Elements	OASPL-1	% Deviation	OASPL-2	% Deviation
0.004	2871	2738	20.1886	----	14.6545	---
0.003	4323	4154	82.3562	3.079341807	71.5521	3.882602614
0.002	10082	9842	100.501	0.220320996	89.674	0.253268597
0.001	36590	36104	115.328	0.14753087	104.504	0.165376809
0.00098	42531	42065	114.337	-0.008592883	103.506	-0.009549874
0.00095	45247	44782	114.866	0.004626674	104.031	0.00507217
0.0009	51640	51149	114.988	0.001062107	104.167	0.001307303
0.00085	59398	58875	115.535	0.004757018	104.709	0.005203183

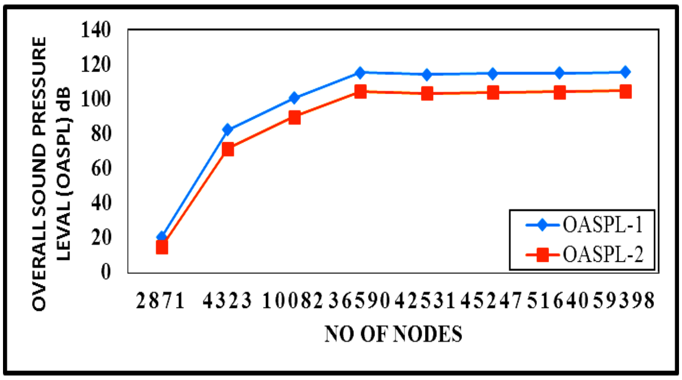

Figure 1 Compare OASPL vs Number of Nodes

The grid independent study was carried out for the test section. The different meshing schemes were created and simulations were done on each grid for velocity to determine how mesh schemes affects variables. For each case, the test section was created using unstructured tetrahedral cell. Evaluation of unsteady flow simulation focused on how mesh discretization altered the OASPL of the test section. The table (1) shows the comparison between the present OASPL and previously published OASPL results for each mesh at the velocity of 69.2 m/s.

The figure represents the grid independency study for the test section. It was found that mesh model with 42000 nodes performed mesh faster to its out significant less in solution accuracy.

1.4. CFD Modeling and simulation Procedure :Model-1 (Without Porous Media) and Model-2 (With Porous Media)

To study effect of porous media on vibration and pressure drop for cross flow in-line tube, we have attached porous media between two cylinders. A two dimensional circular cylinder CFD models i.e. model- 1 (without porous media) and model-2 (with porous media) were created in ANSYS 15. Free stream velocity (U) was applied as the inlet boundary condition over a cylinder diameter (D) is 19 mm and other boundary conditions are adopted as reported by Revell et al. [2].

The distance between the two circular cylinders in both the models i.e. model-1 and model-2 are L (1.5D, 2.0D, and 2.5D). CFD simulations were conducted for four different size of the porous media attached between the tubes i.e. thickness, t (0.25D, 0.5D, 0.75D and 1D). The hydrodynamic parameters viz. viscous resistance (D) 1.531 X 10^9 m^{-2} and inertial resistance values are C= 12600 m^{-1} and porosity 0.662 were taken from the previously published paper [16] and kept constant for all the cases. The domain extends 2.5D in upstream and 17.5D in downstream direction and 2.5D above as well as below the cylinder as shown in figure 2.

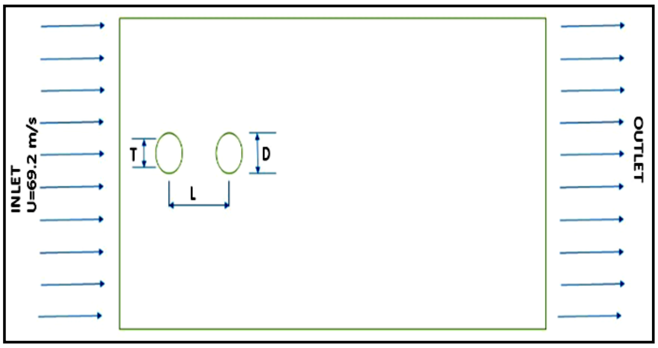

Figure – 2 Computational Geometry (A) Without Porous Media

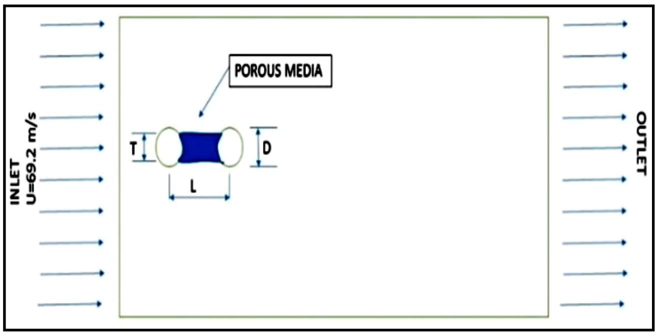

Figure – 2 Computational Geometry (B) With Porous Media

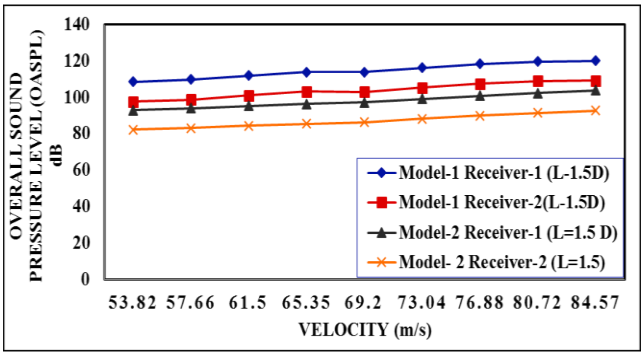

Figure 3 (A) Velocity Vs Overall Sound Pressure level (OASPL) for L=1.5 D

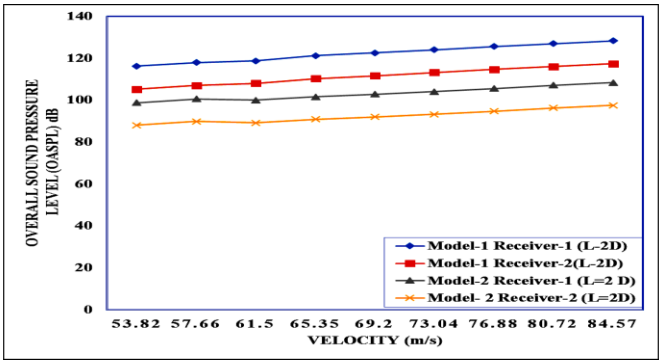

Figure 3 (B) Velocity Vs Overall Sound Pressure level (OASPL) for L=2.0 D

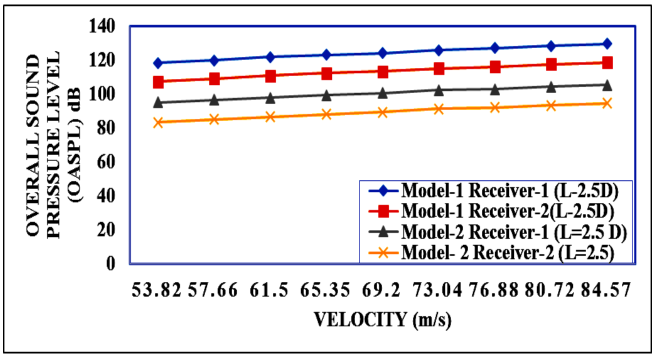

Figure 3 (C) Velocity Vs Overall Sound Pressure level (OASPL) for L=2.5 D

Figure 3.a through figure 3.c display the OASPL over a range of velocity across two test section i.e. model-1(without porous media) and model-2 (porous media) at receiver-1 and receiver-2. Both models, represents the gradually increase in OASPL at receiver-1 and receiver-2 with increase in velocities. The receiver-2 shows lower value of OASPL then the receiver-1.From the graph, it is clearly observed that model-1(with porous media), OASPL at both receiver-1 and receiver-2 is less than the model-2, which is due to the porous media, attached between the test sections.

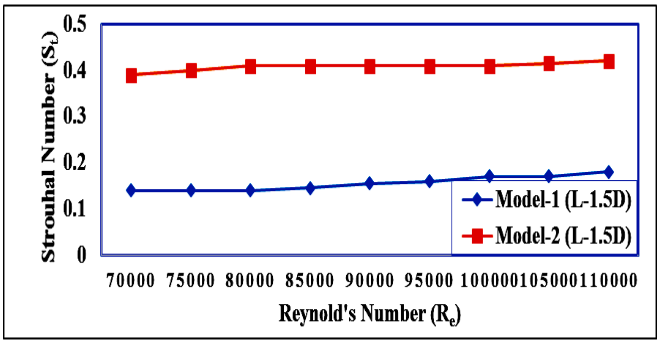

Figure – 4 (A) Reynold's Number Vs Strouhal Number for L=1.5 D

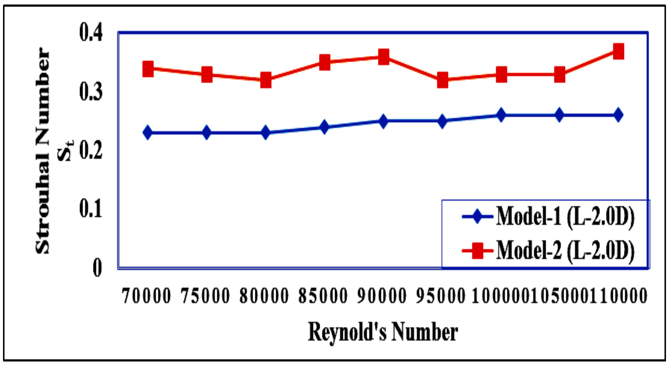

Figure – 4 (B) Reynold's Number Vs Strouhal Number for L= 2.0D

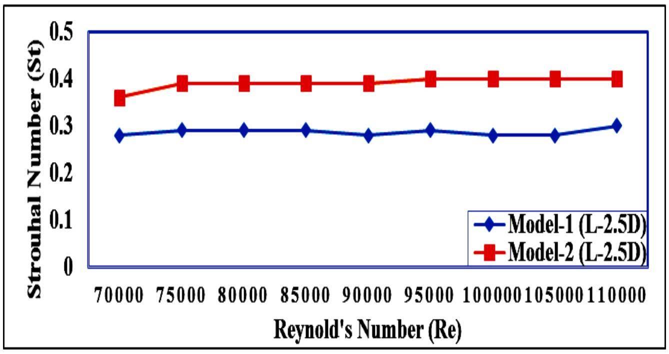

Figure – 4 (C) Reynold's Number Vs Strouhal Number for L= 2.5 D

Figure 4.a through figure 4.c displays the Reynold's number vs Strouhal number for model-1 and model-2. Both model-1 and model-2, shows that, Strouhal number gradually increases with the Reynolds number. It is clearly observed that the model-2 represents larger Strouhal number then model-1. The Model-2 exhibits higher frequency and thus Strouhal number increases with which the vibration over the tube significantly reduces.

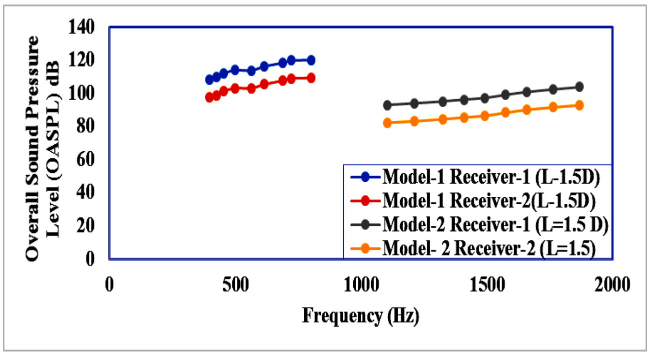

Figure – 5 (A) Frequency Vs Overall Sound Pressure Level (OASPL) for L= 1.5D

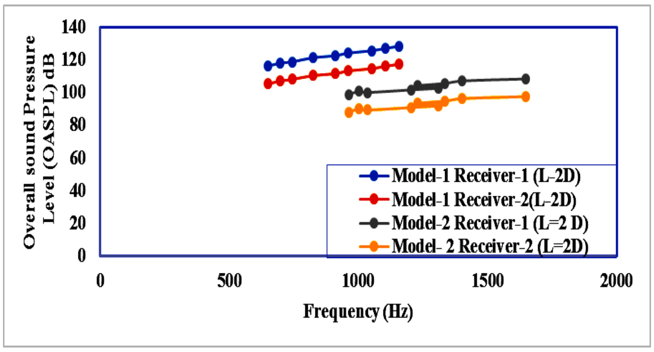

Figure – 5 (B) Frequency Vs Overall Sound Pressure Level (OASPL) for L= 2.0D

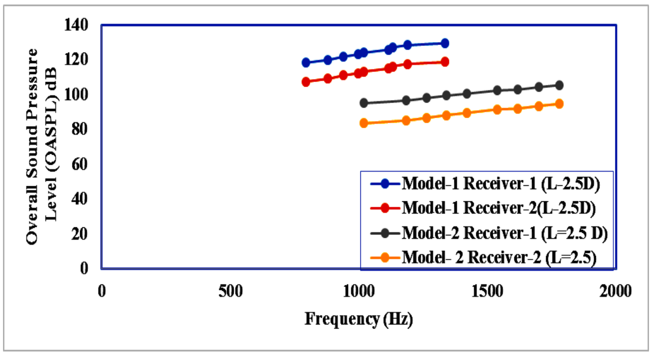

Figure – 5 (C) Frequency Vs Overall Sound Pressure Level (OASPL) for L = 2.5D

The figure 5 (A) through Figure 5 (C), represents the Overall Sound pressure level (OASPL) over a range of frequency across two section i.e. model- 1 (without porous media) and model-2 (porous media) at receiver-1 and receiver-2. Both the Models, shows gradually increase in OASPL at receiver-1 and receiver-2 with increase in frequency. The receiver-2 shows lower value of OASPL then the Receiver-1. From the graph, it is clearly observed that model- 1 (without porous media), OASPL is less than the model-2, which is due to the porous media attached between the test sections.

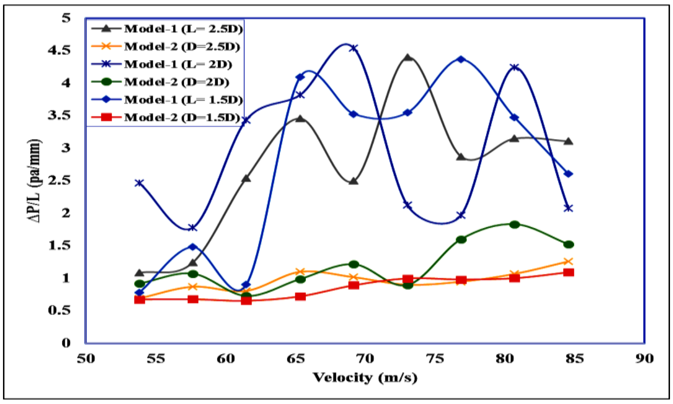

Figure – 6 Velocity Vs Pressure drop per length

From the Figure 6, shows the pressure drop per unit length over a range of velocity. It is observed that the pressure drop per unit length increases for both the model with the increase in the velocity. In model-2 (with porous media) shows gradually increase in pressure drop over a range of velocity, while the model-1 shows more fluctuation in pressure drop per unit length. Model-2 (with porous media) is simulated with varying thickness shows more pressure drop among all.

CONCLUSION

CFD has emerged as a powerful tool for analysis and predicting the unsteady turbulent flow fields and the generated aerodynamic noise and vibration. To study the effect of porous media on pressure drop and vibration of circular cylinder, two- dimensional CFD Models i.e. model-1(without porous media) and model-2 (with porous media) were created using tetrahedral mesh. The CFD model-2 was simulated with changing the size of porous media attached between the tubes i.e. 0.25D, 0.5D, 0.75D and 1D. The grid independent study shows that the medium mesh grid model i.e. with 42000 nodes provides good agreement between present OASPL and previously published result of OASPL. The model-1, OASPL at both receiver-1 and receiver-2 is less than the model-2, which is due to the porous media attached between the test sections. When porous media is attached between the cylinders, the pressure drop across the test section reduces over a range of velocity. The model-2 also represents larger Strouhal number then compared to model-1 and thus increases the frequency; eventually reduces the vibration induced in the tube bank.

REFERENCE

1) G Kato, C., Numerical Prediction of Aerodynamic Sound by Large Eddy Simulation, Trans. Jpn. Soc. Mech.Eng. (In Japanese), Vol. 60 No.569, B, 1994, 126-132.

2) Revell, J. D., Prydz, R. A. and Hays, A.P., “Experimental Study of Airframe Noise vs. Drag Relationship for Circular Cylinders”, Lockheed Report 28074, Final Report NASA Contract NAS1-14403, 1977.

3) Abduljalil et al., “Selection and experimental evaluation of low-cost porous materials for regenerator applications in thermoacoustic engines”, Materials and Design 32 (2011) 217–228.

4) Antoine Corbeil,” Study of Small Hydraulic Diameter Media for Improved Heat Exchanger Compactness”, Master thesis, University of Ottawa, Canada January 2011.

5) Ling Li, Peiqing Liu, “Two Dimensional Circular Cylinder ANDNACA0012 Benchmark Problems of Aero-Acoustics Computation.” ICSV 21, 13-17 July, 2014, Beijing/China.

6) Patrick. N. Okolo, Kun Zhao, Eleonora Neri, John Kennedy and Gareth. J. Bennett, “CAA noise Reduction Parametric Study of Mesh Screens Applied to Landing Gears.” ICSV22, Florence, Italy, 12-16 July 2015.

7) Markus P. Rumpfkeil, Darrel K. Robertson, Miguel R. Visbal, “Comparison of Aerodynamic Noise Propagation Techniques.” National Harbour, Maryland, 52nd Aerospace Sciences, DOI: 10.2514/6.2014-0021.

8) Y. P. Wang, J. Chen, H. C. Lee, K. M. Li, “Accurate simulations of surface pressure fluctuations and flow-induced noise near bluff body at low Mach numbers.” The Seventh International Colloquium on Bluff Body Aerodynamics and Applications (BBAA7) Shanghai, China; September 2-6, 2012.

9) Reinaldo M. Orselli1, Julio R. Meneghini2 and Fabio Saltara, “Two and Three-Dimensional Simulation of Sound Generated by Flow around a Circular Cylinder.” 15th AIAA/CEAS Aeroacoustics Conference (30th AIAA Aeroacoustics Conference) 11 - 13 May 2009, Miami, Florida.

10) Jared S. Cox, Christofer L. Rumsey, Bassam A. Younis, “Computation of sound generated by viscous flow over a circular cylinder.” NASA Technical Memorandum 110339, National Aeronautics and Space Administration, Langley Research Center, Hampton, Virginia 23681-0001.

11) Pal Pandian P, Arun Raj S, “Flow Noise Investigation for an Unsteady Flow past a Circular Cylinder.” IJRAME ISSSN (ONLINE): 2321-3051, Vol. 2 issue.1, January 2014. Page: 84-91.

12) Bruno de Castro Braz, "SIMULATIONS OF AEROACUSTIC OF THE FLOWS AROUND A CILINDER AND AIRFOIL WITH SLAT."

13) Dangguo Yang, Jianqiang Li, Jun Liu, Yi Zhang, Yaohua Li, "Analysis on Physical Mechanism of Sound Generation inside Cavities Based on Acoustic Analogy Method" Open Journal of Fluid Dynamics, 2013, 3, 23-31.

14) Wu et al., "Measurement and correlation of hydraulic resistance of flow through woven metal screens", International Journal of Heat and Mass Transfer 48 (2005) 3008-3017.

15) C.O. Yadav et al.," CFD assisted Prediction of Hydrodynamic Parameters for Regenerator of Cryocooler", Procedia Technology 14 (2014) 328 – 335.

16) Tutorial: Modeling Flow-Induced (Aeroacoustics) Noise, Fluent Inc. May 11, (2005).

BOOKS:

1) John.D.Anderson, "COMPUTATIONAL FLUID DYNAMICS", McGraw-Hill publications.