Effects on Cross Section Shape of Cylinder on Drag and Sound

A. Sabik Nainar, A. Franklin Deenath, M. Godson, R. Gopivarma

***Abstract:** In this project work an attempt is made to predict the far-field aero acoustics noise and aerodynamic drag coefficient for square and circles in the by using the instable Reynolds be close to Navier Stokes equation coupled with the Fowes Williames and Hawkings acoustics analogy. The consider square and circle cylinder have a diameter of 0.016 m and air flow over the cylinder has a Reynolds number of 54800.To reduce thee computational cost the simulation are conducted in a two dimensional computational domain. The obtained numerical results such as aerodynamic lift and drag coefficients and the aero acoustics properties are also compact with experiments. Attempts are also made to study the effect of cross section shape of cylinder an attack as well.*

***Keywords:** aero acoustics sound, aerodynamic drag, CDF, RANS*

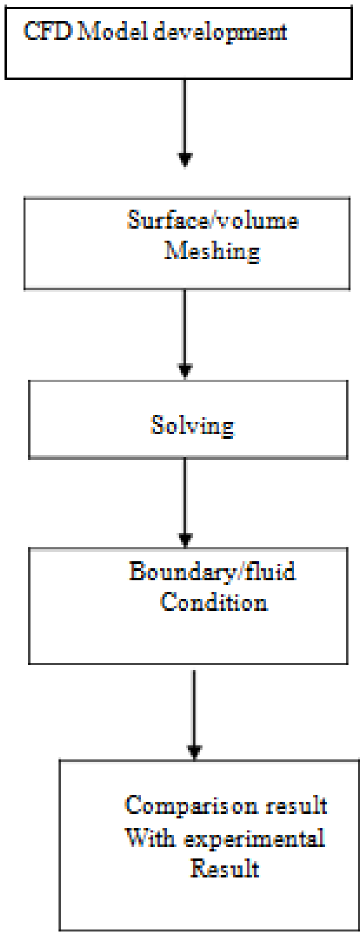
I. INTRODUCTION

Computational Fluid Dynamics (CFD) is a division of fluid mechanic's that uses mathematical study and information structure to analyze besides explain difficulties that contains un solidified flows. Computational are cast-offs to accomplish the designs vital to simulation the allowed stream tide to the fluid, and the collaboration of the fluid (liquid and gases) with the surface well defined through boundary condition.[2]The ANSYS is functional to extensive series of investigation and manufacturing glitches in several fields of study and production, with acoustics , aerodynamic, manufacturing classification projects and analyzed the fluid flows.[3] They builds are basicmechanism automobiles ,trains ,aircraft and manymethods of manufacturing apparatus.. The minimization of unkind and changing force stages created by build forms after placed in a fluid stream has the aids of dropping drag, vibration and discharged and sound. It is consequently actual significant that a good kind of the flow field about the build forms be found to complete these aims. This project explores how low

Reynolds number fluid forms waken noisiness can distract the generation of instable flow and force.

The situation selected for study is the collaboration of a square cylinder and an extremely tinny flat plate. In this paper, an attempt is made to predict the far field aero acoustics noise and aerodynamic drag coefficients for square cylinders by using the instable Reynolds to be closed to Navier Stokes equations coupled with the Fowcs Williams and Hawkings acoustic analogy.[5]The considered square cylinders have a diameter of 0.016 m, and the air flow over the cylinder has a Reynolds number of 54800. To decrease the cost, the reproductions are conducted in a two dimensional computational domain. The obtained numerical results for instance aerodynamic lift and drag factors, and these aero acoustic properties are also compared with experiments. Attempts are also made to propose a method to reduce the drag and sound as well.

II. METHODOLOGY



Revised Manuscript Received on December 05, 2019.

A.SabikNainar, Department of Aeronautical Engineering, Kalasalingam Academy of Research and Education, Krishnankoil 626 126, India. Email:sabiknainar.a@klu.ac.in

Franklin Deenath, Department of Mechanical Engineering, Kalasalingam Academy of Research and Education, Krishnankoil 626 126, India. Email: :franklindeenathff@gmail.com

M. Godson Department of Mechanical Engineering, Kalasalingam Academy of Research and Education, Krishnankoil 626 126, India. Email:ashicgodson18@gmail.com

R. Gopivarma Department of Mechanical Engineering, Kalasalingam Academy of Research and Education, Krishnankoil 626 126, India. Email:gopivarma777999@gmail.com

III. COMPUTATIONAL METHODOLOGY

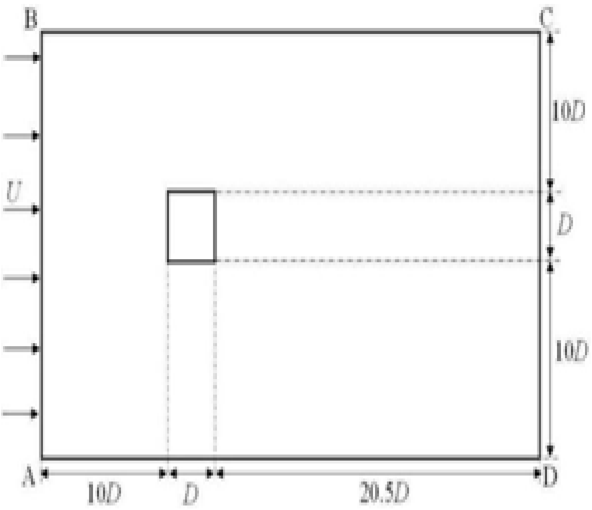
The turbulent air flow field over the cylinder is computed by utilizing the two dimensional unsteady Reynolds be closed to Navier Stokes technique. Analysing sound by using Ffowcs William and Hawkings and analysing drag by using Navier Stokes equation. The *k-ω* SST typical stands active to run closing toward the arrangement of calculations designed by URANS. The fixed volume system through the QUICK outline is applied for 3-D discretization, although a second - order implicit scheme is used for time based discretization. The unassuming algorithm is applied for the pressure-velocity connection. An adaptive time-stepping technique is used in our study, where the time steps are used mechanically with admiration to the error. The time-step is acceptable to revolution in among 10^{-8} s and 10^{-4} s. At what time the enduring are completely variables reduction fair under 10^{-5}, the solution has been observed as united.

IV. NUMERICAL METHODOLOGY

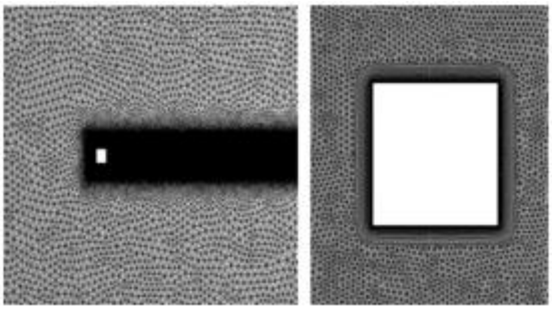
The aerodynamic send intention flinches through pretending low Mach number is 0.09 turbulent flow with an incompressible URANS method. By means of this flow reproduction, the sound basis is gotten. At that time, the sound basis is gotten, the sound circulation is before planned Curle's equation. Far along a length sensible modification is cast-off to additional recover the accurate of the sound calculation in view of the length sensible properties of flow.

V.COMPUTATIONAL DOMAIN& BOUNDARY CONDITIONS

The uniform fluid stream is normal to the inlet boundary 'AB', i.e. *x*-axis is parallel to the incoming velocity *U*. The origin (0, 0) of the coordinate system is positioned at the midpoint of the cylinder. The boundaries 'BC' and 'AD' have zero shear stress and normal velocity. The outlet boundary 'CD' has a zero value of gauge pressure. The wall state is allocated on the plate and cylinder surface.



Domain



Full view **Near view (cylinder)**

VI.VALIDATION OF AERODYNAMIC AND AEROACOUSTIC PARAMETERS:

For numerical calculations, the turbulent airflow stream should attain a statistically steady state and this takes place at approximately 0.1 s. In all computations, the entire simulation time is 1.1 s and the final 1 s (i.e. from 0.1 s to 1.1 s) of time history data is used in calculating the aerodynamic and acoustic results. In other words, flow sampling occurred over a non-dimensional time of $t\ u/D$ = 3125, which equates to approximately 368 vortex shedding cycles. The precision of the SPL calculations primarily relies on the accuracy of the aerodynamic solution. To validate this solution, the mean aerodynamic drag coefficient. ($C_{D,mean}$), lift coefficient ($C_{L,rms}$) and Strouhal number (*St*) corresponding to the dominant vortex shedding frequency, computed numerically, are compared with previous studies (Table 2). The aerodynamic forces on the cylinder consist of two main parts: pressure and shear force acting in directions normal and tangential to the

cylinder surface, respectively. The pressure and shear forces exerting on the cylinder surface are integrated, and resolved in the *x*- and *y*- directions to obtain the drag and liftforces, respectively. The *CD,mean*, *CL,rms* and *S t* of the cylinder obtained from simulations are in good agreement with the previous studies(Bosch and Rodi, 1998; Murakami and Mochida, 1995; Park, 1995; Samion et al., 2016; Shimada and Ishihara, 2002; Sohankar, 2006; Tian et al., 2013; Vickery, 1966).The *St* corresponding to the dominant vortex shedding frequency is computed by fast Fourier transform (FFT) of the lift force time history.

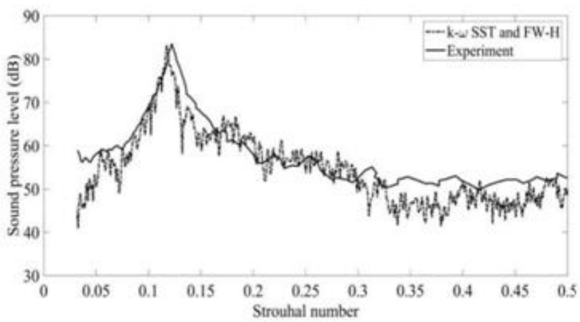
VII.VALIDATION OF THE AERODYNAMIC PARAMETERS:

Method (Author)	$C_{D,mean}$	$C_{L,rms}$	**St**
k-ω SST (Present)	2.151	1.446	0.118

k-ω SST (Tian et al., 2013)	2.06	1.492	0.138
k-ω SST (Samion et al., 2016)	2.1	1.43	0.126
k-ε (Bosch and Rodi, 1998)	2.108	1.012	0.146
k-ε (Shimada and Ishihara, 2002)	2.05	1.43	0.141
LES (Murakami And Mochida, 1995)	2.09	1.6	0.132

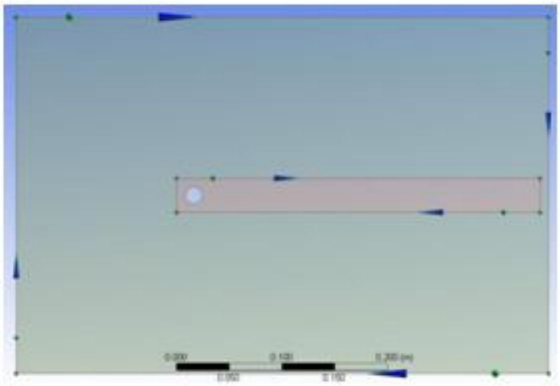
VIII.VALIDATION OF THE ACOUSTIC RESULT:

Acoustic parameters	Exp.	Num.
St_T	0.122	0.118
SPL_T (dB)	83.48	83.06



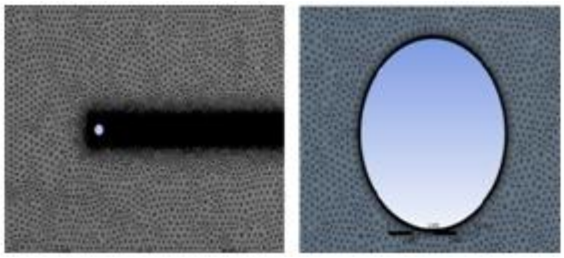
IX.DRAG AND SOUND AT CROSS SECTION SHAPE OF CIRCULAR CYLINDER :

A problem is formulated to analyse the drag and sound as well, at various cross section shape of cylinder of attack. Attempts are made to find the optimal circle at which drag and sound minimizes simultaneously. Commercial CFD solver fluent and the mat lab are used for CFD simulation and signal processingrespectively



X.ANALYSIS OF CIRCLE MESH:

Mesh generation is the practice of creating a mesh, a sub division of a circle cylinder plate geometric space into distinct geometric and topological cells.



Full view Near view

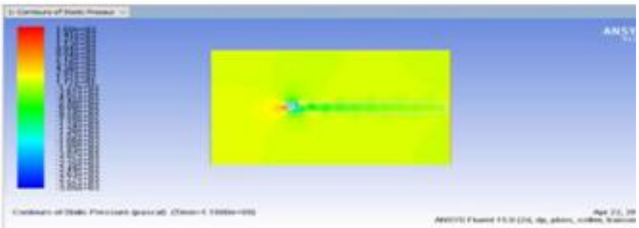
Mesh cells are used as distinct local approximation of the larger domain. while a mesh may be a triangulation. The process of meshing is notable from point set triangulation. The circular cylinder of cross section is applied on triangle meshing, size of edges and inflation of cylinder in mesh process. If the values are applied to generate the mesh .The sizing of edges(1) are selected to hide the inner domain and select the edges ,to hide the outer domain and select the another sizing of edges ,the facing of cylinder is also select .The circular boundary layer value of inflation is 20 .The name selection selected the inlet, outlet, cylinder, upper and lower edges are marked. To generate the mesh respectively.

XI.VALIDATION OF THE ACOUSTIC RESULT IN CIRCLE:

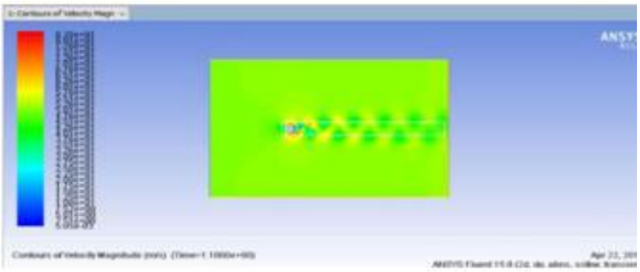
Acoustic parameters	Exp.	Num.
St_T	0.118	0.109
SPL_T (dB)	84.59	84.36

XII.VALIDATION OF THE AERODYNAMIC PARAMETERS IN CIRCLE

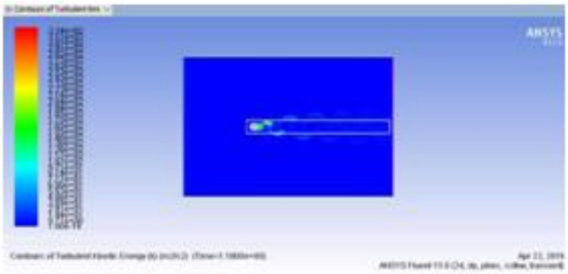
Method (Author)	$C_{D,mean}$	$C_{L,rms}$	St
k-ω SST (Present) square	2.151	1.446	0.118
k-ω sST (present)circle	1.43	0.48	0.19



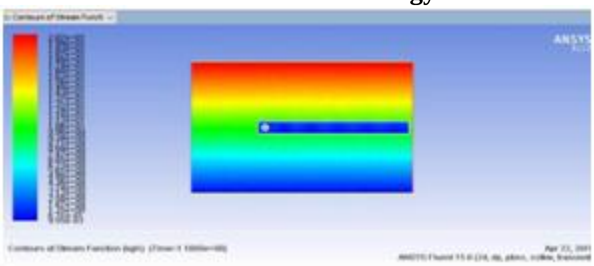
The mean static pressure distribution around circle cylinder



Velocity



Turbulent kinetic energy



Velocity stream function

XIII.CONCLUSION

Two dimension numerical solutions are carried out for the circular cylinder, we have analyzed the aerodynamic drag and sound for the square plate with the circle which is present inside. The one regime was carried out in the flow structure. The results have shown that the aerodynamic drag for circular cylinder with sound is reduced in needful manner.

REFERENCES

1. Abdi, R., Rezazadeh, N., Abdi, M. Reduction of fluid forces and vortex shedding frequency of a circular cylinder using rigid splitter plates. Eur. J. Comput. Mech. 26, 225–244,, 2017
2. Akilli, H., Karakus, C., Akar, A., Sahin, B., Tumen, N.F.,. Control of Vortex Shedding of Circular Cylinder in Shallow Water Flow Using an Attached Splitter Plate. J. Fluids Eng. 2008, 130, 041401.
3. Ali, M.S.M., Doolan, C.J., Wheatley, V.,. Low Reynolds number flow over a square cylinder with a detached flat plate. Int. J. Heat Fluid Flow 2012.36 133-144.
4. Amiraslanpour, M., Ghazanfarian, J., Razavi, S.E.,. Drag suppression for 2D oscillating cylinder with various arrangement of splitters at Re=100: A highamplitude study with OpenFOAM. J. Wind Eng. Ind. Aerodyn. 2017,164, 128–137..
5. Ansys, I.,. ANSYS FLUENT User ' s Guide. 2011, 15317, 2498
6. Apelt, C.J., West, G.S., Szewczyk, A.A.,. The effect of wake splitter plates on the flow past a circular cylinder in the range of 10000<R<50000. Fluid Mech. 1973,61, 187–198..
7. Baron, A., Mossi, M., Sibilla, S.,. The alleviation of the aerodynamic drag and wave effects of high-speed trains in very long tunnels. J. Wind Eng. Ind. Aerodyn, 2001
8. Bosch, G., Rodi, W.,. Simulation of vortex shedding past a square cylinder with different turbulence models. Int. J. Numer. Methods Fluids. 1998.
9. Casalino, D., Jacob, M.,. Prediction of aerodynamic sound from circular rods via span wise. statistical modelling. J. Sound Vib. 2003,262, 815–844.
10. Chauhan, M.K., Dutta, S., More, B.S., Gandhi, .K.,Experimental investigation of a flow over a square cylinder with an attached splitter plate at intermediate Reynolds number. J.Fluids struts.

AUTHORS PROFILE

A. Sabik Nainar, ME, Assistant Professor, Ex Indian Air Force, Life membership In Aeronautical society of India(Graduate) G 11027 Life membership In Indian Society for Technical Education (ISTE) LM 83606.

Franklin Deenath, Student, Department of Mechanical Engineering, Kalasalingam Academy of Research and Education, Krishnankoil 626 126, India.

M. Godson Student, Department of Mechanical Engineering, Kalasalingam Academy of Research and Education, Krishnankoil 626 126, India.

R. Gopivarma Student, Department of Mechanical Engineering, Kalasalingam Academy of Research and Education, Krishnankoil 626 126, India.