3rd International Conference on Recent Development in Engineering Science, Humanities and Management
National Institute of Technical Teachers Training & Research, Chandigarh, India (ESHM-17)
26th March 2017, www.conferenceworld.in ISBN: 978-93-86171-35-1

A REVIEW OF VARIOUS TURBULENCE MODELS USED TO SIMULATE THE FLOW PAST AN ELLIPTIC CYLINDER

Anil Kumar Lor[1], N. K. Singh[2]

[1]*Department of Mechanical Engineering, NIT Kurukshetra, India*

[2]*Associate Professor, Department of Mechanical Engineering, NIT Kurukshetra, India*

ABSTRACT

Flow around bluff bodies such as elliptical, square and circular cylinders is part of the fundamental fluid mechanics. There are extensive literatures on the flow around a circular cylinder because of its practical importance in hydrodynamics and aerodynamics applications and the fundamental significance in the flow physics. Building on the understanding of the basic concepts of stationary circular cylinder, many researchers have studied multiple stationary or oscillating cylinders, making it the synonym for the external separated flows. Many engineering applications such as nuclear rods, heat exchangers, condensers, off shore structures, and bridges can be modelled as a circular cylinder but on the other hand, some applications involve flow past complex bodies like aero plane wings, rotor blades, submarines and missiles which can't be investigated as the flow over a circular cylinder. In such cases, other parameters such as angle-of-attack and the aspect ratio can significantly influence the boundary layer separation, forces such as drag and lift and characteristics of the wake region, the nature of the vortex street and the flow separation phenomenon. Flow with these phenomenon can be modelled assuming bluff body as elliptic cylinder. On the other hand, elliptic cylinders have the fundamental fluid dynamic features between those of a circular cylinder and a flat plate. This article is a review of various studies focused on modelling the flow past an elliptic cylinder. Most of the scientists and researchers uses CFD codes to analyse the models which is subjected to various conditions and compared these results with the experiments.

***Keywords*: Cross flow, Elliptical cylinder, Numerical Simulation, Turbulence modelling**

I. INTRODUCTION

When some placed are placed in a fluid stream, they generate separated flow over a considerable proportion of their surface and hence can be classified as bluff. On bluff bodies with sharp edges, separation is fixed at the trailing edges, whereas on bluff bodies having continuous surface curvature, the location of separation depends both on the nature of the boundary layer and the shape of the bluff body. For low Reynolds numbers when boundary layer separation begins, the flow around the body remains stable, but as the Reynolds number is increased and reached to a critical value, instabilities start occurring. These instabilities can lead to an unsteady wake motion which is organized, disorganized or a combination of both.

Numerous experimental and numerical studies has been done on the laminar and turbulent flow behind bluff bodies. Turbulent flow contains extremely complex physics which involves high adverse pressure gradient, flow separation, and unsteady and aperiodic vortex shedding. Due to these problems, numerical analysis is very difficult. The main objective of the present study is to find out some numerical schemes and concepts to model the flow around an elliptical cylinder which is in the supercritical and upper transition flow regimes. Due to absence of sufficient data in these regimes for an elliptical cylinder, investigation data of flow over a circular cylinder has been included in this study.

II. DIFFERENT MODELING APPROACHES

There are several questions due to which the researchers have taken interest in the turbulent flow around cylinders. The distribution of pressure and location of maximum pressure (force) on the surface of the bodies is required in the design process with regard to failure studies while values of the drag coefficients and lift coefficients on the cylinder is necessary for the assessment of fluctuating forces and heat generated on the surface. Research on this fluid dynamics problem is done by means of experiments and by numerical simulation. For numerical simulation, the three predominant approaches are the

1. Reynolds-averaged Navier-Stokes equations (RANS) models.
2. Large eddy simulation (LES)
3. Direct Numerical Simulation (DNS).

In RANS and LES, the averaged Navier–Stokes equations are solved, where the average is defined as a statistical or temporal average in RANS and as a spatial average over a small volume in LES. Due to the non-linearity of the Navier–Stokes equations, after averaging some additional terms are there for which modelling is needed. At present there are various advanced turbulence models which can be used to simulate such flows and researchers investigate these models to know their suitable applications. Both the above-discussed methods use turbulence models to simulate the flow; but in DNS, Navier-Stokes equations are solved without using any turbulence model. This means that no spatial and temporal averaging is done but their whole range of these scales are resolved which makes this method very costly.

III. REVIEW OF FLOW PAST AN ELLIPTIC CYLINDER

H J Lugt et al. [1] numerically investigated the 3D unbounded laminar flow past an elliptic cylinder with infinite span at angle of attack by using various finite difference schemes. They found that there was no periodicity in the wake for Reynolds number up to 15, and at Re=200, vortex shedding starts thus developing a Karman vortex street. For 0 <, there were no inertial forces acting on the cylinder. Calculations were made using the Oberbeck's integrals for elliptic cylinders with different focal to length ratios and also showed streamlines and vortex patterns for these configurations.

H J Lugt [2] numerically studied the autorotation of an elliptic cylinder with its major axis perpendicular to the flow, for R. He found that due to the synchronisation between the vortex shedding and the rate of rotation of the cylinder, autorotation phenomenon take place. Lock-in happens over a range of the vortex shedding frequency. He gave different conditions for auto rotations to take place.

Jackson [3] by using the finite element simulations investigated the onset of periodicity in the flow which leads to oscillatory forces on the body, in 2D, steady state, laminar flow past an elliptic cylinder, and found that the transition corresponds to a 'Hopf bifurcation' which was located using extended systems of equations. He found the critical Reynolds number and critical Strouhal number for various configurations of the elliptic cylinder, for ex $Re_c = 76.794 \text{ and } St_c = 0.14644 \text{ for}$, elliptic cylinder oriented perpendicular the flow. Also found that the use of extended system techniques makes the computational cost much cheaper than time-dependent calculations for getting these critical quantities.

J K Park et al. [4] numerically investigated the effect of the angle of attack on the unsteady laminar flow past a thin elliptic cylinder by using the DuFort-Frankel scheme and the Buneman algorithm. They found five different regimes of flow for low Reynolds numbers ($25 \leq R$). The first two regimes were steady flow regimes. Regime one, was without any separation bubble whereas the regime two was with a separation bubble. The critical Reynolds number for these two regimes in case of circular cylinder was found to be 8.5. The remaining three regimes were unsteady ones where transition to the Karman vortex street took place. Also found that the vortex shedding frequency in regime three was found to be twice as that of regime five.

Yoshihiro Mochimaru [5] simulated a two dimensional (2D), steady and incompressible flow using the Fourier spectral method for high Reynolds number regime (10). For different minor to major axis ratios (1/12, 1/6, 1/3, 1/2), he evaluated coefficient of drag, streamlines and isobars. Notable reduction in the drag coefficient was observed for higher Reynolds numbers. He concluded that the steady state flow past an elliptic cylinder can be simulated using Fourier spectral method through double exponential transformation in the space.

Mittal et al. [6] investigated the effects of three dimensional effects on the flow at Reynolds numbers for which the flow becomes three-dimensional but was simulated as 2D. They compared the results from 2D and 3D simulations of flow over elliptic and circular cylinders and found that the drag coefficient computed from 2D simulations was notably higher than that from 3D simulations and experimental results. The over prediction of the mean drag prediction was due to the higher Reynolds stresses in the wake region and is more pronounced in bluffer bodies.

Again in 1996, *Mittal et al. [7]* studied the 3D flow past an elliptic cylinder by developing a spectral collocation technique to study the concept of unsteady flow separation and the wake region. They also discussed the inflow and boundary conditions and used discretization techniques. The effect of axis ratio and the angle-of-attack on flow field was investigated by direct numerical simulation for Reynolds numbers up to 1000. To validate their results, they compared the mean drag coefficients and the Strouhal numbers with the existing results from the flow past circular cylinders. They found that the three dimensionality affects the flow phenomenon and there was over prediction in the mean drag and lift amplitude.

M T Nair [8] numerically studied the 2D unsteady laminar flow past an elliptic cylinder with aspect ratio of 0.1. For a combination of two Reynolds numbers (200 and 10000) with two non-dimensional rotation rates (1 and 4), they studied the lift and drag coefficients. They concluded that the wake region was smaller for higher Reynolds number and the shedding was also increased.

Johnson et al [9] carried out a numerical investigation of the flow past an elliptic cylinder with different axis ratio and low Reynolds number by using the spectral-element method. The axis ratio was varied from 0.01 to 1 and Reynolds number from 30 to 200. They found that there was a peak in the Strouhal number for the transition in wake region from normal vortex street to a secondary vortex shedding street and a minima in the drag coefficient for an axis ratio. Also, the critical Reynolds number for which the vortex shedding starts, was decreasing with decrement in the axis ratio.

Zhihua Li et al. [10] numerically simulated the flow over an elliptic cylinder using $2D\ URANS\ SS$

turbulent model with different axis ratio and Reynolds number up to . Coefficient of pressure, rms lift

coefficient, mean drag and the local and surface averaged Nusselt number were predicted for different axis ratios. As slenderness of the cylinders is increased, a reduction in the pressure drop and the drag forces is reported when compared to the circular cylinder. But Nusselt number also follows the same trend. They also compared the results of lenticular and elliptic cylinders which were found to be more or less comparable to each other.

M S Kim et al. [11] studied the unsteady flow past an elliptic cylinder using SIMPLER algorithm of Patanker (1980) along with Crank-Nicolson temporal integration method for various thickness-to-cord ratios (t/c=0.6, 0.8, 1.0 and 1.2) and different Reynolds numbers (Re=200, 400 and 1000). They found that the pressure drag was dominating in total drag and increases with increment in the t/c. Also the mean friction drag depends upon the Reynolds number. The amplitude of drag and lift forces were found to be increasing with the Reynolds number and t/c but rate of increment is more with t/c.

Hu Ruifeng [12] performed a computational study on the 3D flow past a rotating elliptic cylinder with different aspect ratios and rotational speeds. For Re=200, they found that the periodic vortex shedding get suppressed or clustered with increasing the rotational speeds. He also presented iso-surfaces of wake structures for different AR and rotating speeds. At low axis ratio, the streamwise vortices are found to be comparable to the spanwise vortices, while increasing the AR, the strength of spanwise vortices were reported to be more.

S N Naik et al. [13] presented numerical solutions for the uniform flow over an elliptic cylinder for different rotational speed using the immersed boundary method (IBM). They studied the periodic flow for a Re=100 and a fixed AR=0.1. Vortex shedding was reported for all the rotation rates and 'Hovering Vortex' were also reported for $\alpha(non\ dimensional\ rotation\ r$ and also studied the factors affecting this phenomenon.

With increasing the rotation rate, the lift amplitude was also found to be increasing.

IV. VARIOUS TURBULENT MODELS

Spalart-Allmaras turbulence model is one of the most popular one-equation model which is based on eddy-viscosity turbulence models. An additional transport equation needs to be solved for a term which is the modified form of the turbulent kinetic viscosity for using this model. The model is extensively used in the aerodynamics as it gives good results for these applications. In its original form, the production term is

ISBN: 978-93-86171-35-1

dependent upon rotation tensor while in the modified version, the measures of both rotation and strain tensors for the production term are combined. It gives good results for low-Reynolds-numbers. It includes damping functions for turbulent viscosity, turbulent production and dissipation. To fulfil these requirements, the meshing has to be very fine in the near-wall region.

K-epsilon model is most common turbulence model which is used in CFD, focuses on the mechanism that affects the turbulent kinetic energy. It is a two equations turbulence model, one equation is for turbulent kinetic energy and other one is for dissipation rate. The two equations are solved to calculate the turbulent viscosity, hence Reynolds stress by using Boussinesq relationship. The standard model along with the Boussinesq equation, works well for a broad range of engineering problems. But, when the problem include intensive isotopes of flow or unbalanced effects, this model finally reaches to responses which are over-diffused, i.e., the values predicted by this model will be large.

The Realizable turbulence model is comparatively a recent development and an improved version of the standard model. It contains a new formulation and a new transport equation for the turbulent viscosity and the dissipation rate respectively. The constant in RKE model, which is present at the turbulence viscosity term as a closure coefficient, arranged in such a manner that it includes the effect of the strain and rotation tensors. Furthermore, for the turbulence dissipation with functional expression, a new transport equation was developed. This model is a high-Reynolds-number model which requires very fine mesh near the wall and is suitable for fully turbulent flows only.

The RNG turbulence model was developed using a mathematical technique called "renormalization group" (RNG) in which renormalization of the instantaneous Navier-Stokes equations is done to take account for the effects of smaller eddies. The analytical derivation results in a model with constants different from those in the standard k-ε model. Model also have some additional terms and functions in the transport equations. For flows with separation and recirculating regions, this model along with the modified coefficients provides results which are accurate and less diffusive than standard model.

Wilcox turbulence model (WKO) is an improved version of the standard model. Like the standard version, two additional transport equations for the turbulence kinetic energy, k and the dissipation per unit kinetic energy, are required to be solved. The constants which are present in both k and ω transport equations, are expressed as functions in the *WKO* model that improves the performance of the standard model for the free shear flows without affecting the boundary layer flow.

The SST turbulence model comes under the two-equation eddy-viscosity model which is more improved version of the standard . Due to the known weakness of the standard model, which is its sensitivity to free stream boundary conditions for free shear flows, a cross-diffusion term in the equation along with a blending function is included. It uses the formulation in the near-wall region and for the outer free stream conditions. The SST model is considered good in prediction capability for the flows with adverse pressure gradients and flow separations.

Large eddy simulation (LES) is a space filtering method in CFD. Large eddy simulation (LES) falls between DNS and RANS in terms of the fraction of the resolved scales. LES directly computes the large-scale turbulent structures which are responsible for the transfer of energy and momentum in a flow while modeling the smaller scale of dissipative and more isotropic structures. A filter function is used in the LES to differentiate between the small and large scale eddies. A filter function dictates which eddies are large by introducing a length scale, usually denoted as in LES, the characteristic filter cut-off width of the simulation. All eddies which are larger than the defined length scale are resolved directly, while the smaller one are approximated.

Detached eddy simulation is a 3D unsteady numerical solution, which acts as a sub-grid-scale model in regions where the grid density is fine enough, and as a *RANS* model in regions of coarse grid density. The model senses the grid density and adjusts itself to a lower level of mixing, in order to unlock the larger-scale instabilities of the flow and to let the energy cascade extend to length scales close to the grid spacing. Therefore, the meshing need not be that fine as in case of *Large-Eddy Simulation* (*LES)* and thus reduces the computation cost. The new length scale in DES depends upon grid spacing. It is selected as the minimum of *RANS* length scale and , where is the maximum dimension of the grid cell.

Dynamic LES model is one in which the Smagorinsky model constant , is dynamically computed based on the information provided by the resolved scales of motion making the model self- tuning. The dynamic procedure thus obviates the need for users to specify the model constant in advance which is a function of space and time. The dynamic model provides a systematic way of adjusting or allowing it to be a function of position which is desirable for inhomogeneous flows. The dynamic method allows adapting the filter size in the spatial direction. It shows improvement in complex flows as for example drag crisis flow around a cylinder.

Partially averaged Navier-Stokes (PANS) model in which unresolved kinetic energy parameter varies with grid spacing and the turbulence length scale. The two-equation turbulence model is used for the unresolved kinetic energy and the dissipation where a constant is replaced by unresolved kinetic energy parameter ( ). The parameter varies between zero and one and has the magnitude equal to one in the viscous sub layer, and when the *RANS* turbulent viscosity becomes smaller than the *LES* viscosity.

In the *Mesh free vortex method,* the vorticity field is modeled using a cloud of point's vortices. The total velocity field can be constructed by adding up the contribution of the vortices in the cloud and their image, the uniform flow and the dipole term that models the circular cylinder. Complex velocity field induced at a point can be obtained by using the circle theorem. The vortex method uses discrete points to model the vortices, whose transport at each time step is carried out in a sequence.

VMS-LES hybrid model consist in splitting between the large resolved scales *(LRS)* i.e. those resolved on a virtual coarser grid, and the small resolved ones *(SRS).* The *VMS-LES* method does not compute the *SGS* component of the solution, but modelizes its effect on the small resolved scales which corresponds to the highest level of discretization, and preserves the Navier-Stokes model for the large resolved scales. The *SGS* model is used to modelizes the dissipation effect of the unresolved scales on the resolved scales. To distinguish the resolved scales in to the largest and smallest ones is the main idea of this method.

In the *Hybrid RANS/VMS-LES model*, VMS-LES and RANS approaches are combined by the use of *NLDE (Non-Linear Disturbance Equations)* technique in which the solution of Navier- Stokes equations is decomposed into three parts. First part is mean (*RANS*), the second one is that takes into account the turbulent large-scale fluctuations and a third part consist of the unresolved *(SGS)* fluctuations. The basic idea is to solve the *RANS* equations and to correct the obtained averaged flow field by adding the resolved fluctuations in a hybrid mode. A blending function is introduced to the suitability of the grid used for the LES. The blending function varies between 0 and 1. When Blending function is less than one, additional resolve fluctuations are computed while blending function is 1 when the *RANS* approach is recovered. In this model, RANS approach is used in the viscous sub layer and in the separating shear layers, while for the wake region of the bluff body, LES model is used making it a hybrid model.

In the *LES with RGF algorithm,* random flow generation algorithm is incorporated into a finite element code with which the LES code is applied to generate a realistic inflow field. The algorithm takes correlation tensor of the original flow field, length and time scales of turbulence as the input. These quantities can be obtained from a steady state RANS simulations or experimental data. The outcome of the procedure is a time dependent flow field and it is divergence free inhomogeneous anisotropic flow field.

Direct numerical simulation (DNS) is the most accurate approach to simulate the turbulenc. This approach involves the numerical solution of the Navier-Stokes equations that govern fluid flow without modeling with its accuracy is only bounded by the accuracy of numerical scheme adopted, boundary conditions and discretizations. The main problem is that, the large number of grid points and the small size of time steps required to capture the so small time and space scale of turbulent motion make the advanced turbulent computations cumbersome.

V. CONCLUSIONS

Significant advances were made in the last decade in understanding the fundamentals of fluid flow over a circular cylinder with improved computational mesh generation techniques and efficiency of solvers but in the case of an elliptical cylinder, absence of comprehensive data is observed in the supercritical and upper transition regimes. Many engineering application, which operates at high speed or in a highly turbulent regime, are very difficult to model. So some numerical simulation in these regimes should be conducted to avoid such circumstances.

The circular cylinder that is a special and ideal case of an elliptical cylinder (minor to major axis ratio = 1) can be used for the validation of different approaches to turbulence modeling. A wide number of research papers are available in the literature. LES and different hybrid models are able to predict more accurately in supercritical and transcritical regimes but still, most researchers use the two-equation models, mainly the SST to

simulate the flow past a circular cylinder as this model is relatively simple and reasonably accurate. If higher accuracy is the main requirement, one should adopt models other than the two-equation models.

Overall the present study helps someone in finding the suitable method to model an elliptical cylinder in high Reynolds number regimes.

REFERENCES

[1.] Lugt, H.J. and Haussling, H.J., 1974. Laminar flow past an abruptly accelerated elliptic cylinder at 45 incidence. Journal of Fluid Mechanics, 65(04), pp.711-734.

[2.] Lugt, H.J., 1980. Autorotation of an elliptic cylinder about an axis perpendicular to the flow. Journal of Fluid Mechanics, 99(04), pp.817-840.

[3.] Jackson, C.P., 1987. A finite-element study of the onset of vortex shedding in flow past variously shaped bodies. Journal of Fluid Mechanics, 182, pp.23-45.

[4.] Park, J.K., Park, S.O. and Hyun, J.M., 1989. Flow regimes of unsteady laminar flow past a slender elliptic cylinder at incidence. International Journal of Heat and Fluid Flow, 10(4), pp.311-317.

[5.] Mochimaru, Y., "Numerical simulation of flow past an elliptical cylinder at moderate and high Reynolds number using spectral method," 11th Australian fluid mechanics conference, 1992.

[6.] Mittal, R. and Balachandar, S., 1995. Effect of three- dimensionality on the lift and drag of nominally two- dimensional cylinders. Physics of Fluids, 7(8), pp.1841-1865.

[7.] Mittal, R. and Balachandar, S., 1996. Direct numerical simulation of flow past elliptic cylinders. Journal of Computational Physics, 124(2), pp.351-367.

[8.] Sengupta, M.T.N.T.K., FLOW PAST ROTATING ELLIPTIC CYLINDERS.

[9.] Johnson, S.A., Thompson, M.C. and Hourigan, K., 2001, December. Flow past elliptical cylinders at low Reynolds numbers. In Proc. 14th Australasian Fluid Mechanics Conference, Adelaide University, South Australia, Dec (pp. 9-14).

[10.] Li, Z., Davidson, J.H. and Mantell, S.C., 2006. Numerical simulation of flow field and heat transfer of streamlined cylinders in cross flow. Journal of heat transfer, 128(6), pp.564-570.

[11.] Kim, M.S. and Sengupta, A., 2005. Unsteady viscous flow over elliptic cylinders at various thickness with different reynolds numbers. Journal of mechanical science and technology, 19(3), pp.877-886.

[12.] Ruifeng, H., 2015. Three-dimensional flow past rotating wing at low Reynolds number: a computational study. Fluid Dynamics Research, 47(4), p.045503.

[13.] Naik, S.N., Vengadesan, S. and Arul, K., 2016. Flow past rotating low axis ratio elliptic cylinder. In 46th AIAA Fluid Dynamics Conference (p. 4348).