International Journal of Application or Innovation in Engineering & Management (IJAIEM)
Web Site: www.ijaiem.org Email: editor@ijaiem.org, editorijaiem@gmail.com
Volume 2, Issue 6, June 2013 ISSN 2319 - 4847

High Reynolds number flow over cylinders: A Review

Manish Rawat[1], Rajesh Gupta[2] and R.M.Sarviya[3]

[1]PG student, Department of Mechanical Engineering, MANIT, Bhopal
[2]Associate Professor, Department of Mechanical Engineering, MANIT, Bhopal
[3] Professor & HOD, Department of Mechanical Engineering, MANIT, Bhopal

Abstract

Flow around bluff bodies such as circular and elliptical cylinder is the fundamental fluid mechanics problem. Many engineering applications such as off shore structures, bridge piers and pipelines can be modeled as a circular cylinder but flow over complex bodies like wings, submarines, missiles, and rotor blades, in which the parameters such as axis ratio, angle of attack and initial velocity profile can influence the flow characteristics of the wake, nature of separation and forces (Drag and lift force). Such type of flow can be modeled as an elliptical cylinder. Furthermore, elliptic cylinders have the general fluid dynamic features between those of a circular cylinder and a flat plate. This article is a review of numerous analyses focused on modeling the flow over a cylinder at higher Reynolds number. This study contains more than 30 research papers on circular cylinder as well as on elliptical cylinder. Most of the scientists and researchers used CFD codes to analyze the models subjected to various conditions and compared results with measurements.

Keywords: Circular cylinder, Elliptical cylinder, Cross flow, High Reynolds number, Turbulence modeling.

1. INTRODUCTION

The laminar and turbulent flow behind such bluff bodies has been the subject of numerous experimental and numerical studies. Turbulent flow contains extremely complex physical features involving high adverse pressure gradient, flow separation, and vortex shedding, due to these numerical analyses is very difficult. The main objective of present study is to find out some numerical scheme and concept to model an elliptical cylinder in the supercritical and upper transition flow regimes. In the absence of sufficient data in these regimes for an elliptical cylinder, research paper over a circular cylinder has been included in this study.

2. FLUID FLOW MODELING APPROACHES OVER CYLINDERS

The interest of researchers in the turbulent flow around cylinders is due to several questions. The pressure distribution and location of maximum pressure (force) on the bodies is required in the design process with regard to failure while knowledge of the skin friction and drag force around the bodies is necessary for the assessment of required propulsive force and heat generated on the surface. Research on this fluid mechanical problem is conducted by means of experiments and, increasingly, by numerical simulation. For research the two predominant approaches of their numerical solution are the

1. Reynolds averaged simulation (RAS)
2. Large eddy simulation (LES).

Both methods solve the averaged Navier–Stokes equations, 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 their averaged forms contain additional terms for which models are needed. They are termed turbulence models in RANS and sub grid or sub filter scale models in LES. Presently there are various advanced turbulence models which can be used to simulate such flows but lack of the universal model still enforce the researchers to understand the best area of these models.

3. REVIEW OF FLUID FLOW OVER CIRCULAR CYLINDER

3.1. RANS based models

Muk Chen Ong et al. [1] investigate high Reynolds number flow (1×10^6, 2×10^6, 3.6×10^6) around a smooth circular cylinder by using 2D URANS equation with a standard K-epsilon turbulence model. Applicability of standard k -epsilon model for engineering applications in the supercritical and upper-transition flow regimes was examined in this research. The essential hydrodynamic quantities such as coefficient of drag, lift and strouhal number predictions shows acceptability of the data. The computed c_d and skin friction coefficient decrease slightly as the Reynolds number increase. The predicted mean pressure distribution at Re= 10^6 capture the trend of experimental data but the discrepancy is larger due to different implementation of the wall function.

***Muk Chen Ong et al.* [2]** present a computational study of high Reynolds number flow around a circular cylinder close to flat seabed by using standard K-epsilon turbulence model. The effect of gap to diameter ratio and seabed roughness for a given boundary layer thickness of the inlet flow on hydrodynamic quantities and resulting bed load have been predicted. In this research flow at Re = 3.6×10^6 and δ/D= 0.4 with seabed roughness ($Z_w = 1\times10^{-6}$, 2×10^{-5}) are investigated. The drag coefficient increases as G/D increase for G/D $\leq$ 0.3, reaches maximum then decrease approaching a constant which means that influence of seabed gap become negligible. The higher roughness reduces coefficient of drag for same Reynolds number. strouhal number increase up to G/D=0.6 then remain constant for both roughness. For a small gap, the pressure coefficient around the cylinder is asymmetric about the horizontal centerline of the cylinder results a net upward lift force on the cylinder. For an intermediate gap (G/D=0.25), the positive pressure zone becomes symmetric at the front of the cylinder, but the suction at the gap is large, and causes a negative mean lift force on the cylinder. For large gaps, Cp becomes symmetric, and the mean lift force approaches zero.

The effect of spin ratio on the load exerted by the fluid over a circular cylinder at high Reynolds number was examined by ***S.J. Karabelas et al.* [3**] using 2D RANS equations with modified K-epsilon turbulence model. This research investigates the effect of the flow over a rotating cylinder at high rotational rates. 12 rotational rates from 0 to 8 are examined at 3 Reynolds number, Re = 5×10^5,10^6 and 5×10^6. This study shows that the lift and drag force varies slightly in the Reynolds number range (less than 10%). Lift increases linearly with spin ratio (a) and the drag force increases up to a = 4, where it reaches a plateau and eventually decreases. Turbulence dissipation rate, kinetic energy and shear stress are found to increase nonlinearly with dimensionless rotational rate.

***Bijan Mohammadi et al.* [4]** implemented the classical K-ε model with wall law for unsteady turbulent flow past a circular cylinder in the range Re = 10^4 - 10^6 including subcritical, supercritical regimes and drag crisis phenomenon. The mean drag coefficient is under predicted by this method for a wide range and strouhal number is over predicted. The length of separation bubble predicted shown good agreement with two layer RSE model. The mean velocity distribution along the center line behind cylinder by this analysis yield fair agreement with experiment but pressure drop curve is unrealistic due to over prediction of the horizontal velocity at the boundary of the wake. So this classical K-ε model with wall law for unsteady turbulent flow is not satisfactory.

***A.C. Benim et al.* [5]** investigate the turbulent flow past a circular cylinder computationally by means of RANS equation with Shear Stress Transport (SST) model by resolving the near wall layer. A wide range of Reynolds numbers is investigated (Re=1×10^4 to 5×10^6), encompassing the critical regime. The boundary layer transition and the drag coefficient reduction can qualitatively be predicted by this SST model resolving the boundary layer. Quantitatively, the predicted drag coefficients underestimate the measurements throughout. This is expected to be due to the influence of the organized transient motion, which could not be taken into account by the RANS-SST formulation. This study suggested applying a transient, three-dimensional model such as LES for such type of flows.

***Ismail celik et al.* [6]** used K-ε model for numerical prediction of long time average flow over circular cylinder at high Reynolds number (10^4 – 10^7). In subcritical region, the transitional model is used and all predictions such as pressure distribution, wall shear stress, and velocity field shows fair agreement with the other results up to the separation point but thin boundary layer should have fine grid. After separation point, the predictions are not satisfactory due to presence of vortex shedding and adverse pressure gradient. In supercritical region, the predicted separation angle (118^o) and coefficient of pressure shows acceptability of this model in the range but wall shear stress prediction is more than other results. This research suggests to use fine grid near the cylinder to obtain fine results. So K-ε model with transition works well for this type of flow.

***B.N. Rajan et al.* [7]** examined the 2D URANS with eddy-viscosity based turbulence models over a circular cylinder at Reynolds number varying from 10^4 - 10^7 to predict the numerous flow characteristics. The models used are K-ε, SST, SA and V2F for this study. Coefficient of drag and lift coefficient shows periodic fluctuation due to vortex shedding for all models at Re= 10^5. The maximum lift coefficient and mean drag coefficient predictions by V2F model agree well with previous measurement. The drag coefficient and strouhal number predicted by all model is in qualitative sense but magnitude is over predicted. All model show a decrement in C_d up to the Re = 10^7 and increasing trend is not observed in transcritical region. Surface pressure and skin friction prediction on cylinder surface shows discrepancies after 10^5 and location of separation point predicted downstream as compared to measurement data. So URANS predictions are in qualitative sense but accurate prediction required more sophisticated model.

***Ugur Oral Unal et al.*[8]** presented a computational study to compare the effect of turbulence modeling in predicting some flow field variable of the near wake region of circular cylinder at Re= 41300. The turbulence models used are S-A, RKE, Wilcox K-ω and SST model. The comparison with experimental results shows that S-A and RKE turbulence model is not suited for the flow with a high level of separation while the WKO and SST model predictions are in good agreement. WKO predicted the recirculation bubble thickness more accurately than SST. On the other hand, SST model shows better predictive capability for the contour of velocity, vorticity and shear stress. WKO over predicted the stream wise length scale of the flow field also it under predicts the extreme value. SST model captured the thickness of shear layer more accurately than WKO. Overall in this flow range the use of SST turbulence model with a fine mesh near the wall appears to predict more realistic and accurate result.

M. E. Young et al. **[9]** examined the effect of variation in inlet turbulence length scale on the flow properties by Kω-SST model at Re = 1.4×10^5 and investigate the acceptability of modified time limit K-ω model. The modification in turbulent viscosity term prevents production of turbulent kinetic energy in highly strained area. As turbulent length scale is increased from 0.001D to 0.1D, C_D and C_L have periodic variation. The variation in C_D is -7.8 to +8.5% from the mean value and in strouhal number is -5.6 to +3.1%. Coefficient of pressure is increased with length scale but under predicted in comparison to experiment. The performance of CFD is poor in capturing the effect of boundary condition variations, and it is evident that improvements in eddy-viscosity modeling are required. The time limit model is examined at Re = 10^3 to 3×10^6 and the result shows that this model predict more closely than standard model upto 5×10^5.

3.2. LES model

Michael Breuer **[10]** examined the applicability of LES for high Reynolds number (Re = 140,000) subcritical flow over circular cylinder and investigate the effect of dynamic subgrid scale (SGS) modeling and Smagorinsky model at different grid on the predicted result. This study shows that dynamic model works well for complex flow at higher Re and grid refinement does not improve the prediction quality due to dependency of the filter width from the grid resolution. This research shows acceptability of LES results for high Reynolds number flows, especially in the near wake region.

Pietro Catalano et al. **[11]** examined the accuracy of LES with wall modeling for high Reynolds number (0.5×10^6, 1×10^6, 2×10^6) flow around a circular cylinder. Boundary condition provides by simple wall stress model. The results for drag coefficient, base pressure coefficient, Strouhal number and mean pressure distribution on the cylinder surface are within the experimental and URANS result range at Re up to 1×10^6. LES over predict the skin friction coefficient due to strong pressure gradient and vortex- shedding pattern was not clear in LES. Also it shows Reynolds number insensitiveness in contrast with experiments. The present research advised to use a more systematic approach to capture the grid resolution and the wall-modeling effects.

Tian cheng Liu et al. **[12]** implemented Lattice Boltzmann method to solve turbulent flow with high Reynolds number up to 140000 around a circular cylinder with sub-grid turbulence model. This method relates the turbulence relaxation time with particle distribution. Pressure lines and streamlines shows that as Reynolds number increases, there are more and more small-scale vortices appearing on the backward face, and the flow turned into turbulent flow. The prediction of drag coefficient and strouhal number showed very good agreement with others. So the extended Lattice Boltzmann method is suitable for unsteady turbulent flow.

Ryan Merrick et al. [13] conducted an experimental and numerical simulation to study the effect of surface roughness on the separation point and wake intensity. The aim of this study is to control the flow by using artificial roughness and correlates it with Reynolds number so that transition in subcritical region occur similar to supercritical region with artificial roughness. A number of roughness pattern are investigated in BLWT at Reynolds number from $1\text{x}10^4$ to $2\text{x}10^5$. The simulations performed by using a large eddy simulation (LES) at high Re ($1.5\text{x}10^6$) and medium Re ($1\text{x}10^5$) and low Re ($3.0\text{x}10^4$). The pressure coefficient is compared for both the case and it shows that artificial roughness pattern can simulate super critical flow properties in sub critical region. The suitability of roughness pattern for definite flow characteristics is remains for research.

S.J. Karabelas **[14]** implemented large eddy simulation to predict the effect of spin ratio varying from 0 to 2 on the flow parameters of a rotating cylinder at Re = 1.4×10^5. The streamline of velocity at different spin ratio shows that the first stagnation point on cylinder surface is moving forward to a greater angle and lower vortex diminishes and finally eliminate as the spin ratio increase in case of counter clockwise rotation of cylinder. The results for coefficient of lift and drag are in good agreement with others. Lift coefficient increases with spin ratio while the drag coefficient reduces. The negative mean pressure coefficient reduces with spin ratio and its position moves toward lower surface. For spin ratio greater than 1.3, the load stabilized after a transition period and variation in lift coefficient reaches its minimum value. All predictions by LES model show well agreement with other results.

R. Panner Selvam **[15]** simulated the drag crisis phenomena over a circular cylinder by using 2 dimensional LES and FEM at high Reynolds number in subcritical and supercritical regions (Re = 10^4, 10^5 , 5×10^5, 10^6). The magnitude of drag coefficient and reciprocal of strouhal number decreases, as the Reynolds number increases. Both shows similar trend as experiment but the value are over predicted. By using special wall function the drag crisis phenomena can be observed. 3d modeling and fine grid can be used for the drag crisis phenomena.

Meng Wang et al. **[16]** examined the validity and accuracy of LES with wall modeling at high Reynolds number (Re = 5×10^5, 1×10^6, and 2×10^6). LES with wall model based on turbulent boundary layer, models the near wall effect and put the boundary conditions in terms of wall shear stress. The mean drag coefficient, the base pressure coefficient, and the Strouhal number predictions are in well agreement up to Re = 1×10^6 and after that the predictions shows Reynolds number insensitiveness. Drag coefficient doesn't show increasing trend up to 2×10^6. The skin friction coefficient are over predicted on the front half but acceptable on the back half at all Reynolds number. The inaccuracy at high Reynolds number is due to poor grid resolution. So a more sophisticated study is required for wall modeling and grid resolution.

Angelo A. Mustto et al. **[17]** proposed a new numerical model for the prediction of flow characteristics over a circular cylinder at the Reynolds number ranging from10^4 to 6×10^5. In this new method, LES is used with mesh free vortex method to simulate the larger motion. The mean coefficient of drag predictions show qualitative trend with over predicted values. The higher drag coefficient is due to the absence of three dimension effect in 2D numerical simulation. The pressure coefficient predictions are acceptable from the front face to the point of separation but at the back face, results are under predicted. All results show the acceptability of mesh free vortex method to simulate complex flow with acceptable accuracy.

3.3. Hybrid and miscellaneous model

Stephen Wornom et al. **[18]** conducted a computational study on Variational multiscale large-eddy simulations (VMS–LES) of the flow over a circular cylinder at the subcritical Reynolds number up to 2×10^5. A combination of FEM and FVM discretization is used on unstructured grid with eddy viscosity scale model which separates the grid into the large resolved scale and small resolve scale. The predicted variation of strouhal number, drag coefficient and rms coefficient of lift with Reynolds number show similar trend qualitatively and quantitatively. The overall agreement is good for the coefficient of pressure at the surface but overestimation at stagnation point is predicted.

Ju-Yeol You et al. **[19]** investigate the turbulent flow around a circular cylinder at Re = 3.6×10^6 by using an unstructured mesh technique, and compared URANS(S-A, SST) and hybrid RANS/LES (DES, SAS) models. Hybrid RANS/LES means that computational region is divided by URANS near the cylinder and LES for the remaining field. Unstructured mesh technique is used to model a volume easily and meshed the dense cells forcibly in the wake region. Vortex shedding pattern obtained from URANS is 2D and continues while hybrid model gives 3D irregular pattern. The Strouhal number (0.25) shows good agreement while coefficient of pressure was under predicted for all models used. The magnitude of recirculation length and bubbles size predicted by hybrid model is larger than URANS. The average turbulent kinetic energy is mostly contributed by transverse component and its magnitude is over predicted by Hybrid model than URANS. This difference can be reduced by changing turbulent length scale or source term makes the hybrid model more realistic.

Kyle D. Squires et al. **[20]** examined the delayed detached eddy simulation to predict the super critical flow around a circular cylinder against the results of base line version of the method. The flow over the cylinder is at Re =8×10^6. Three grids with coarsest mesh having 1.47×10^6 cells and finest mesh having 9.83×10^6 cells are used for prediction of pressure and force coefficient. The averaged drag coefficient, Strouhal number, coefficient of skin friction and separation angle for each of the grids using DES97 and DDES are in good agreement. The pressure coefficient for both models and each grid are similar.

Andrei travin et al. **[21]** examined the validity of detached eddy simulation predictions with laminar and turbulent separation. The Reynolds number used for laminar separation is 50,000, 140,000, while 140,000 and 3×10^6 is used for turbulent separation. The coefficient of pressure and drag predictions with medium grid refinement shows good agreement with experimental result than finer grid. Skin friction coefficient on cylinder surface reveals a poor agreement with experimental results after point of separation for laminar separation but for turbulent separation, poor agreement is shown over the complete surface. Wake statistics about velocity, mean recirculation bubble length shows fair result at lower Reynolds number and Reynolds stress in the wake shows qualitative agreement with over predicted magnitude. Overall the predictions of DES meet with partial success.

M. Saghafian et al. [22] implemented the nonlinear eddy-viscosity models to the circular cylinder through a range of Reynolds numbers, from 10^3 to 10^7, containing subcritical to supercritical flow. The model of Craft et al. (CLS model) with some adjustment in the coefficients of the 'cubic' terms is applied to steady ambient flows. In steady flow, the coefficient of drag, lift and strouhal number variation with Reynolds number show good agreement if cubic terms are reduced by 60%. The ratio of friction drag to total drag shows good agreement in the subcritical regime.

Alaa Elmiligui et al. **[23]** implemented two multiscale turbulence model, first model consist of K-ε model with hybrid RANS/LES transitions function depends upon grid spacing and length scale while the other model is modified version of the partially averaged Navier-Stokes (PANS) model, in which unresolved kinetic energy parameter (f_k) varying with grid spacing and the turbulence length scale. Flow characteristics are predicted for stationary and rotating cylinder in laminar and turbulent separated region. The predictions for coefficient of drag, coefficient of pressure and strouhal number by these models compared well with the experimental results for stationary cylinder. Cylinder rotates at different spin ratio (0.0, 0.3, 0.6, and 0.9), the free stream Mach number is 0.3 and Re = 50000 is used for the predictions on rotating cylinder. The coefficient of drag decreases while coefficient of lift and strauhal number increases with increasing spin ratio. Pressure distribution becomes asymmetric with increasing speed produces the lift. So the RANS/LES and the PANS approaches gives much better predictions than others.

Masahisa Tabata et al. **[24]** applied finite-element scheme to a problem of high Reynolds number flows varies from at Re = 10^3 to lo^6 past a circular cylinder and investigate the effect of boundary layer subdivision on the flow characteristics. In different subdivisions, the number of nodal points varies in the boundary layer. The predictions with fine subdivision of the boundary layer shows decrement and the recovery of the drag coefficient at high Reynolds numbers. For higher Reynolds numbers than Re = $1o^4$, finer subdivisions will be needed for the quantitative analysis and for lower Reynolds number the predictions are same for coarser and finer subdivision of boundary layer. All predictions show that a fine subdivision of the boundary layer was required in order to capture the behavior of the flow at a high Reynolds number.

H. Lubcke et al. **[25]** compared the LES and RANS predictions for the turbulent flow around the bluff bodies up to Reynolds number 140000. The aim of the study is to assess the acceptability of explicit algebraic stress models (EASM) and compare the results with LES and other studies. The LES prediction for coefficient of drag is under predicted by 50% while the prediction of EASM is in well agreement. Strouhal number, recirculation length and the centerline velocity predictions by both models shows acceptability in the subcritical region. All predictions show the capability of the EASM to capture the important flow features.

Shuyang Cao et al. **[26]** presented a combined experimental and computational study of Vortex shedding and aerodynamic forces around a circular cylinder in linear shear flow at subcritical Reynolds number varies from 1.0×10^4 to 3.6×10^4. Smagorinsky subgrid scale model is used to study the effect of shear parameter β on flow characteristics. The shear parameters depends upon velocity gradient, diameter and upstream mean velocity at the center plane of the cylinder, varies from 0 to 0.27. The result shows that the shear parameter has no significant influence on the vortex shedding frequency at subcritical Reynolds number. Base pressure increased with shear parameter due to vortex shedding suppression in the shear flow. Drag force decreased with increasing shear parameter due to the recovery in the base pressure. Lift force produced in the shear flow is due to the asymmetry of pressure distribution around the cylinder and difference in velocity. The separation point in the shear flow shifts downstream on the high-velocity side, and moves upstream on the low-velocity side.

S. Dong et al. **[27]** investigates the turbulent flows past a stationary circular cylinder and past a rigid cylinder undergoing forced harmonic oscillations at Reynolds number Re = 10^4 By direct numerical simulations (DNS). Multilevel **-** type parallel algorithm with combined spectral-element/Fourier discretization on unstructured grids is used in the simulations. The drag coefficients, lift coefficients and the strouhal number are in good agreement and mean pressure distribution on the cylinder surface with high-resolution mesh agree well with the experimental results. Flow statistics (mean, r.m.s. velocities and Reynolds stress), Reynolds stress contour and unsymmetrical lobes in Reynolds stress distribution shows the acceptability of present DNS for the stationary cylinder. In case of oscillating cylinder, frequency varies from 0.14 to 0.25. In this range, the drag coefficient changes slightly while the lift Coefficient increases dramatically as the oscillation frequency increases. The predictions for lift force verses phase angles and the lift coefficients in phase with velocity shows reasonable agreement with experimental results.

A new hybrid RANS/LES (Fluctuation Correction Model, FCM) model was proposed by ***A. Belme et al.*** **[28]** to simulate the flow around a circular cylinder at Reynolds number 140000. In this approach the flow variables are decomposed in a RANS part (i.e. the averaged flow field), a correction part for the turbulent large-scale fluctuations, and a third part made of the unresolved or 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. The hybrid model involves a blending parameter which allows a smooth passing from RANS to LES. The predictions by FCM for pressure distribution over the surface and velocity field are in good agreement with experimental and other studies.

Carine Moussaed et al. **[29]** implemented a VMS-LES model with dynamic subgrid scale (SGS) models for the prediction of flows around a circular cylinder in subcritical regime at Re = 2×10^4. VMS-LES decompose the resolved scales into the largest and smallest ones and to add the SGS model only to the smallest ones. The mean drag coefficient and strouhal number predictions from VMS-LES model well agrees with experimental and other results. Mean pressure coefficient distribution at the cylinder show less discrepancy with this approach than other, especially at the rear part of the cylinder. The quantitative agreements for other parameters like mean recirculation length and r.m.s of the lift coefficient is good with this approach.

A. Belme et al. [30] implemented VMS-LES and a hybrid RANS/VMS-LES model to simulate Turbulent flows around circular cylinder at sub and supercritical Reynolds numbers up to 2×10^6. The VMS-LES is used for subcritical cases while hybrid RANS/VMS-LES is used for supercritical region predictions. In subcritical region Strouhal number, mean recirculation bubble length decreases and r.m.s. of the lift coefficient increases as the Reynolds number increases. Coefficient of drag increases slowly to a quasi-plateau attained around Reynolds 20000. All predictions in subcritical region show the acceptability of VMS-LES in subcritical region. In supercritical region, forced supercritical at Re = 140000 through the use of RANS close to the wall and Re= 500000, 1 million and 2 million is used for simulation. Forced supercritical is used for the validation of the approach. A good agreement is observed for main bulk parameters like coefficient of drag, lift and pressure at back side and recirculation length.

Maria-Vittoria Salvetti et al. **[31]** investigates the prediction capabilities of hybrid RANS/VMS-LES model with unstructured mesh in the simulation of the flow around a circular cylinder at Re = 140000. The coarse and fine mesh with different values of blending function is used for the predictions. The prediction shows that the flow characteristics are insensitive to blending function definition and sensitive to mesh refinement. With finer mesh, the coefficient of drag, lift and recirculation bubble length decrease while strouhal number and coefficient of pressure increases. The main flow characteristics predictions are in acceptable limit with such hybrid model.

Stephen Wornom et al. **[32]** studied the impact of the dynamic LES modeling and VMS-LES, when used separately or in combination inside a hybrid model, in the high Reynolds number flow ranging between 500K and 3M, around a cylinder. The various flow characteristics were predicted for different meshes. The variation with Reynolds number for main flow characteristics like coefficient of drag, lift, base pressure and strouhal number shows less over predictions than other models. The simulations performed with this hybrid model show a rather good agreement with coarse meshes.

M. Tutar et al. **[33]** implemented a random flow generation (RFG) algorithm with LES model over a two dimensional circular cylinder at Reynolds number of 140,000. The numerical simulations are performed in conjunction with the RFG algorithm for different turbulent inflow boundary conditions and mesh refinement. Influence of mesh refinement is examined at inflow turbulence level of 0.6 percent. The deviations from the experimental data decrease with the increase in mesh resolution. As the mesh resolution is increased, the vortex structures near the Separation point become larger and the location of the separation point moves downstream. The influence of turbulence level is examined in the range from 0.6 to 6%. The position of the separation points shift rearwards with increasing inflow turbulence level. The base pressure increases and the minimum pressure, time averaged and fluctuating drag and lift coefficients decrease with increasing inflow turbulence level.

Heng Xiao et al. [34] implement hybrid LES/RANS approach with dual mesh framework for turbulent flow around a circular cylinder at Re= 3.6×10^6. For better predictions with this model, the wall-normal mesh resolution is very fine than in other directions. The hybrid method with dual frame work produced much finer turbulent structures as well as narrower wake width. The better resolved wake contributes well to predict the drag force applied on the cylinder and drag crisis phenomena due to the correct prediction of flow separations behind the cylinder. The hybrid method also showed better performance for the pressure distribution around the wall surface.

4. REVIEW OF FLUID FLOW OVER AN ELLIPTICAL CYLINDER

The effect of minor to major axis ratio on the coefficient of drag and convective heat transfer coefficient over an elliptical cylinder was investigated by **Zhihua Li et al. [35]** , using 2D URANS with kω-SST model and energy equation upto Re = 10^4. Coefficient of pressure, friction, drag and average Nusselt number was predicted at axis ratio of 0.3, 0.5 and 0.8 and compared with published data for circular cylinder. This study focuses on the use of streamlined profile cylinder rather than circular cylinders to reduce the drag force and Nusselt number. The streamlined tubes result in a delay of separation and reductions in drag force as the cylinders are made more slender. The Nusselt number also shows the same trend as drag force.

Yoshihiro Mochimaru [36] implemented a fourier spectral method to analize the 2D flow past an elliptical cylinder in higher Reynolds number regimes. Coefficient of drag, streamlines and isobars are evaluated at various axis ratios (1/12, 1/6, 1/3, 1/2). The study shows a remarkable reduction in coefficient of drag in higher Reynolds number region as it was expected experimentally. All results of this study shows that moderately higher Reynolds number flow overran elliptical cylinder can be simulated by fourier spectral method through stantially double exponential transformation in space.

G. Heidarinejad et al. [37] applied the direct numerical simulation to study the effect of the aspect ratios and the angles of attacks on the wake flow over an elliptical cylinder, using a Random vortex method is presented. The suitability of the present model was checked by comparing with the experimental data of an ellipse with aspect ratio = 1 (circle) and the results agree with the experiment. The variation of the geometrical parameters has an important effect on the structure of flow. Among all parameters the effects of the angle of attack is more pronounced.

R. Mittal et al. [38] studied the phenomenon of unsteady separation and the structure of wake in the flow over an elliptical cylinder. The effect of thickness and angle-of-attack on the flow field was evaluated using direct numerical simulation at moderate Reynolds number. Strouhal number and drag coefficient for flow over a circular cylinder was compared in order to validate the results. Research contains two- and three-dimensional simulations for a range of flow and geometric parameters and the effect of inflow and outflow boundary conditions was investigated. The results shows that three-dimensionality affects the lift and drag forces of the cylinders; 2D numerical models at high Reynolds numbers, tends to over predict the average level of drag and the amplitude of lift coefficient.

Moon-Sang Kim et al. [39] studied the effects of Reynolds number and elliptical cylinder thickness on the drag and lift force. Thickness-to-chord(t/c) ratios of 0.6, 0.8, 1.0, and 1.2 are simulated at different Reynolds numbers of 200, 400 and 1,000 using 2D Navier stokes equation with SIMPLER algorithm of Patankar (1980) in conjunction with Crank-Nicolson time integration method. This study shows the relation among drag force, Reynolds number and t/c ratio. The pressure drag increases with Re and t/c but rate of increment is more with thickness to chord ratio than Reynolds number. The amplitudes of lift and drag force oscillations increases with Re and t/c but the amplitude of drag force is much less than that of lift force. The strauhal number increases as either Reynolds number increases or thickness decreases.

5. TURBULENCE MODELS

S–A turbulence model is one of the most popular one-equation eddy-viscosity-based turbulence models. The model requires the solution of an additional transport equation for a quantity that is a modified form of the turbulent kinematic viscosity. The model produces good results especially for the aerofoil and wing applications. In its original form, the production term is dependent upon rotation tensor while in the modified version, the measures of both rotation and strain tensors for the production term are combined. S–A is a low-Reynolds-number model and includes damping functions for turbulent viscosity, turbulent production and dissipation. Hence it requires a very fine mesh distribution in the near-wall region.

K-epsilon model focuses on the mechanism that affects the turbulent kinetic energy. It has two model equations, one for turbulent kinetic energy and other one 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 ν_t values predicted by this model will be large.

The RKE turbulence model is an improved version of the standard k–ε model. The constant Cμ 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, a new transport equation was developed for the turbulence dissipation with functional expression. RKE is a high-Reynolds-number model which is suitable for fully turbulent flows only.

The *RNG* $k-\varepsilon$ *model* is derived from the instantaneous Navier-Stokes equations, using a mathematical technique called “renormalization group" (RNG) methods. The analytical derivation results in a model with constants different from those in the standard k-ε model, and 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.

WKO turbulence model is an improved version of the standard k–ω model. Like the standard version, two additional transport equations for the turbulence kinetic energy, k and the dissipation per unit kinetic energy, ω 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 k–ω model for the free shear flows without affecting the boundary layer flow.

The SST turbulence model is a further improved version of the standard k–ω model. Due to the known weakness of the standard k–ω model, which is its sensitivity to free stream boundary conditions for free shear flows, a cross-diffusion term in the ω equation along with a blending function is included. The standard k-ω model and the transformed k-ε model are both multiplied by blending function and both models are added together. The blending function is designed to be one in the near-wall region, which activates the standard k-ω model, and zero away from the surface, which activates the transformed k-ε model. The SST model also includes a new definition of the eddy-viscosity improving the prediction capability of the standard k–ω model for boundary layer flows with adverse pressure gradients.

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. There is a single velocity and model field, and no issue of smoothness between regions. In DES, the new length scale substitutes the RANS length scale. The new length scale depends upon grid spacing. The new length scale is selected as the minimum of RANS length scale and 0.65×δ where δ is the maximum dimension of the grid cell.

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. In order to distinguish between the large scales and small scales, a filter function is used in LES. A filter function dictates which eddies are large by introducing a length scale, usually denoted as Δ in LES, the characteristic filter cutoff width of the simulation. All eddies larger than Δ are resolved directly, while those smaller than Δ are approximated.

Dynamic LES model is one in which the Smagorinsky model constant, Cs, 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 Cs in advance which is a function of space and time. The dynamic model provides a systematic way of adjusting Cs or Cw allowing it to be a function of position which is desirable for inhomogeneous flows. The dynamic method allows adapting the filter size in 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 (f_k). The parameter f_k varies between zero and one and has 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 use discrete points to model the vorticity, 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. The main idea of VMS-LES is to decompose the resolved scales into the largest and smallest ones and to add the SGS model only to the smallest ones.

In *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 a mean part (RANS), a perturbed/corrected part that takes into account the turbulent large-scale fluctuations and a third part made by 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. The closure terms provided by a RANS and a SGS eddy-viscosity model are blended together through the introduction of a blending function to identifying whether the grid is suitable to LES or not. 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. The model works in RANS mode in the boundary layer and in the shear layers detaching from the cylinder while complete LES fluctuations are added in the wake, passing through a layer in which the model works in hybrid way.

LES with RGF algorithm In this method 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 turbulence simulation. 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.

6. 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 case of 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 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 of the computational works on circular cylinder employ two-equation turbulence models, predominantly the standard k-ε model for modeling turbulence phenomena due to its simplicity and reasonable accuracy. In case of higher accuracy required one should adopt other models rather than 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] M.C. Ong, T. Utnes, L.E. Holmedal, D. Myrhaug, B. Pettersen, “Numerical simulation of flow around a smooth circular cylinder at very high Reynolds numbers,” Marine Structures 22, pp. 142–153, 2009.

[2] M.C. Ong, T. Utnes, L.E. Holmedal, D. Myrhaug, B. Pettersen, “Numerical simulation of flow around a circular cylinder close to a flat seabed at high Reynolds numbers using a k–ε model,” Coastal Engineering 57, pp. 931–947, 2010.

[3] S.J. Karabelas, B.C. Koumroglou, C.D. Argyropoulos, N.C. Markatos, “High Reynolds number turbulent flow past a rotating cylinder,” Applied Mathematical Modelling 36, pp. 379–398, 2012.

[4] B. Mohammadi and G. Medic, “A Critical Evaluation of the Classical k-ε " Model and Wall-Laws for Unsteady Flows over Bluff Bodies”.

[5] A. C. Benim, M. Cagan, A. Nahavandi, E. Pasqualotto, “RANS Predictions of Turbulent Flow Past a Circular Cylinder over the Critical Regime,” 5th IASME / WSEAS international Conference on Fluid Mechanics and Aerodynamics, Athens, Greece, August 25-27, 2007.

[6] I. Celik , F.D. Shaffer, “Long time averaged solutions of turbulent flow past a circular cylinder,” Journal of Wind Engineering and Industrial Aerodynamics 56, pp. 185–212, 1995.

[7] B.N. Rajani, A. Kandasamy and S. Majumdar, “On the Reliability of Eddy Viscosity Based Turbulence Models in Predicting Turbulent Flow past a Circular Cylinder Using URANS Approach,” Journal of Applied Fluid Mechanics, Vol. 5, No. 1, pp. 67-79, 2012.

[8] U.O. Unal, M. Atlar, O. Goren, "Effect of turbulence modelling on the computation of the near-wake flow of a circular cylinder, "Ocean Engineering 37, pp. 387–399, 2010.

[9] M. E. Young and A. Ooi, "Turbulence Models and Boundary Conditions for Bluff Body Flow," 15th Australasian Fluid Mechanics Conference The University of Sydney, Sydney, Australia 13-17 December 2004.

[10]M. Breuer, "A challenging test case for large eddy simulation: high Reynolds number circular cylinder flow," International Journal of Heat and Fluid Flow 21, pp. 648-654, 2000.

[11]P. Catalano, M. Wang, G. Iaccarino, p. moin, "Numerical simulation of the flow around a circular cylinder at high Reynolds numbers," International Journal of Heat and Fluid Flow 24, pp. 463–469, 2003.

[12]T. Liu, G. Liu, Y. Ge, H. Wu, W. Wu, "extended lattice boltzmann equation for simulation of flows around bluff bodies in high reynolds number," BBAA VI International Colloquium on bluff bodies aerodynamics & applications Milano, Italy, July, 20-24 2008.

[13]R. Merrick and G. Bitsuamlak, "Control of flow around a circular cylinder by the use of surface roughness: A computational and experimental approach".

[14]S.J. Karabelas, "Large Eddy Simulation of high-Reynolds number flow past a rotating cylinder," International Journal of Heat and Fluid Flow 31, pp. 518–527, 2010.

[15]R.P. Selvam, "Finite element modeling of flow around a circular cylinder using LES," Journal of Wind Engineering and Industrial Aerodynamics 67 & 68, pp. 129–139, 1997.

[16]M.Wang, P. Catalano and G. Iaccarina, "Prediction of high Reynolds number flow over a circular cylinder using LES with wall modeling," Center for Turbulence Research Annual Research Briefs 2001. pp. 45-50, 2001.

[17]A.A. Mustto and G.C.R. Bodstien, "Subgrid scale modeling of turbulent flow around circular cylinder by mesh free vortex method," Engineering applications of computational fluid mechanics Vol. 5, No. 2.pp. 259-275, 2011.

[18]S. Wornom, H. Ouvrard, M.V. Salvetti, B. Koobus, A. Dervieux, "Variational multiscale large-eddy simulations of the flow past a circular cylinder: Reynolds number effects," Computers & Fluids 47, pp. 44–50, 2011.

[19]J.Y. You and O.J. Kwon, "Numerical Comparisons Between URANS and Hybrid RANS/LES at a High Reynolds Number Flow Using Unstructured Meshes," Int. J. Of Aeronautical & Space Sciences, Vol. 11, No. 1, March 2010.

[20]K.D. Squires, V. Krishnan, J.R. Forsythe, "Prediction of the flow over a circular cylinder at high Reynolds number using detached-eddy simulation,"Journal of Wind Engineering and Industrial Aerodynamics 96, pp. 1528–1536, 2008.

[21]A. Travin, M. Shur, M. Strelets and P. Spalart, "Detached-Eddy Simulations Past a Circular Cylinder," Flow, Turbulence and Combustion 63: pp. 293–313, 1999.

[22]M. Saghafian, P.K. Stansby, M.S. Saidi, D.D. Apsley, "Simulation of turbulent flows around a circular cylinder using nonlinear eddy-viscosity modelling: steady and oscillatory ambient flows," Journal of Fluids and Structures 17, pp. 1213–1236, 2003.

[23]A. Elmiligui, K.S.A. Hamid, S.J. Massey, S.P. Pao, "Numerical Study of Flow Past a Circular Cylinder Using RANS, Hybrid RANS/LES and PANS Formulations".

[24]M. Tabata and S. Fujima, "Finite-element analysis of high Reynolds number flows past a circular cylinder," Journal of computational and Applied Mathematics 38, pp. 411-424 North-Holland, 1991.

[25]H. Lubcke, St. Schmidt, T. Rung, F. Thiele, "Comparison of LES and RANS in bluff-body flows," Journal of Wind Engineering and Industrial Aerodynamics 89, pp. 1471–1485, 2001.

[26]S. Cao, Y. Tamura, "Flow around a circular cylinder in linear shear flows at subcritical Reynolds number," Journal of Wind Engineering and Industrial Aerodynamics 96, 1961–1973, 2008.

[27]S. Dong, G.E. Karniadakis, "DNS of flow past a stationary and oscillating cylinder at Re = 10000," Journal of Fluids and Structures 20, pp. 519–531, 2005.

[28]A. Belme, A. Dervieux, B. Koobus, S. Wornom , M.V. Salvetti, application of hybrid and vms-les turbulent models to aerodynamic simulations, 27th international congress of the aeronautical sciences, 2010.

[29]C. Moussaed, A. Belme, S. Wornom, B. Koobus, A. Dervieux, "A dynamic variational multiscale LES model for the simulation of bluff body flows on unstructured grids".

[30]A. Belme, S. Wornom, A. Dervieux, B. Koobus, M.V. Salvetti, "Assessment of VMS-LES and hybrid RANS VMS-LES models".

[31]M.V. Salvetti, H. Ouvrard, B. Koobus, S. Camarri, A. Dervieux, "A Hybrid Model Based on Continuous Fluctuation Corrections".

[32]S. Wornom, C. Moussaed, A. Belme, M.V. Salvetti, B. Koobus, A. Dervieux, "The impact of dynamic and variational multiscale LES modelling in hybrid calculations".

[33]M. Tutar, I. Celik and I. Yavuz, "Modelling of effect of inflow turbulence on large eddy simulation of bluff body flows," Mathematical and Computational Applications, Vol. 11, No. 3, pp. 225-234, 2006.

[34]H. Xiao, Y. Sakai and P. Jenny, "A Dual Mesh Framework for Turbulence Modeling: Application to Flow Around a Circular Cylinder," Institute of Fluid Dynamics, ETH Zurich, Switzerland

[35]Z. Li, J. Davidson and S. Mantell, "Numerical simulation of flow field and heat transfer of streamlined cylinders in cross flow," 2005 ASME Summer Heat Transfer Conference, USA

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

[37]G. Heidarinejad and S. Delfani, "Numerical simulation of the wake flow behind an ellipse using random vortex method," IJE Transactions A: Basics, Vol. 16, No. 2, pp. 183-192, June 2003.

[38]R. Mittal, S. Balachandar, "Direct Numerical Simulation of Flow Past Elliptic Cylinders," Journal of computational physics 124, pp. 351–367, 1996.

[39]M.S. Kim and A.S. Gupta, "Unsteady Viscous Flow over Elliptic Cylinders At Various Thickness with Different Reynolds Numbers," Journal of Mechanical Science and Technology, Vol 19, No 3, pp 877-886, 2005

AUTHOR

Manish Rawat received the M.TECH degrees in Theraml Engineering from Moulana Azad National Institute of Technology, Bhopal in 2013. He is having an experience of at least 03 years in teaching and 02 years in industries. he is doing his research work in the field of fluid flow and heat transfer. He now with a well known private university.