International Journal of Scientific & Engineering Research Volume 8, Issue 7, July-2017
ISSN 2229-5518

Experimental analysis of velocity distribution over a sphere placed in wind tunnel and its comparison by CFD

Suhaib Hasan, Talha Hasan, Sayyed Haider, Naimuddin

Abstract — a wind tunnel was used to analyze the flow past a sphere, which was kept in the test section of an open circuit subsonic flow wind tunnel. The objective of the study was to visualize, observe, and measure the flow around the sphere and to measure the drag force on the body. The study was performed at different stream velocities and the velocity and pressure distribution was observed at different strategic points on the sphere. The velocity and pressure distribution was recorded using a multi tube manometer connected with the sphere model with rubber tubes. A CFD simulation was performed using ANSYS FLUENT software and a comparison was performed with that of wind tunnel experimental values. The results showed very little deviation between the Experimental and the simulated values thereby concluding that a CFD analysis can be conveniently used in place of Experimental work if appropriate parameters are used during the simulation.

Index Terms— Wind tunnel, CFD, Sphere model, Coefficient of drag

1 INTRODUCTION

1.1 WIND TUNNEL

Wind tunnel is a specially designed and protected space into which air is drawn, or blownby mechanical means in order to achieve a specified speed and predetermined flow pattern at a given instant. The flow so achieved can be observed from outside the wind. Wind Tunnel through transparent windows that enclose the test section and flow characteristics is measurable using specialized instruments. An object, such as a model, or some full -scale engineering structure, sphere, air foil model, and cylinder can be immersed into the established flow, thereby disturbing it. The objectives of the immersion include being able to simulate, Visualize, observe, and/or measure how the flow around the immersed object affects the immersed object.

1.2 CFD

Computational fluid dynamics (CFD), a fast growing component in computer aided engineering, plays a very vital role in reducing costs and turn -around times in the design and development of aircraft.

CFD is an acronym that refers to "Computational Fluid Dynamics". CFD uses numerical methods to solve the fundamental nonlinear differential equations that describe fluid flow (the Navier -Stokes and allied equations) for predefined geometries and boundary conditions. The result is a wealth of predictions for flow velocity, temperature, density, and chemical concentrations for any region where flow occurs.

.

The CFD simulations and wind tunnel testing represent an brand new design concepts. These complex simulations or wind tunnel results show whether the aircraft aerodynamics behaviors are acceptable for the purpose of its design.

Two features of the CFD outshine wind tunnel testing and the element of cost is one of such advantages. During the preliminary aircraft design phase, wind tunnel models undergo multiple modifications. These modifications, which can lead to higher costs, are necessary in order to optimize design configuration or allow iteration changes. Fortunately, CFD simulations do not require these costly and time -consuming model modifications. There is no expensive model alteration to carry out or down time in the wind tunnel while the model is being fixed. These CFD simulations can apply changes to the virtual models as quickly as they can be modified in the computers to obtain new results. This time saving benefit is another edge that CFD simulation has over the traditional wind tunnel testing. In the same amount of time needed to conduct a wind tunnel testing, many simulations could be completed to produce far more extensive results and detailed flow field information that wind tunnel results are incapable of showing. For full configuration aircraft models, these extensive results can show the detail flow field interaction of the wing -fuselage interface whereas the wind tunnel results can only present the overall aerodynamics behaviours. In the design phase, especially in the preliminary stage, it would be impractical to study several major configuration changes without the use of CFD. The ability to obtain results with CFD in a short amount of time stands out against wind tunnel testing that requires time to create or modify a model.

- *Suhaib Hasan is currently working as an Assistant Professor in the department of mechanical Engineering, Al falah University, Faridabad India-121004 E-mail: suhaibhasan09@gmail.com*
- *Talha Hasan is currently working as an Assistant Professor in the department of mechanical Engineering, Al falah University, Faridabad India - 121004 E-mail: talhahasan1070@gmail.com*
- *Sayyed Haider is currently working as an Assistant Professor in the department of mechanical Engineering, Al falah University, Faridabad India-121004 E-mail: sayyedhaider.me@gmail.com*
- *Naimuddin is currently working as an Assistant Professor in the department of mechanical Engineering, Al falah University, Faridabad India - 121004 E-mail: naimlkc007@gmail.com*

International Journal of Scientific & Engineering Research Volume 8, Issue 7, July-2017
ISSN 2229-5518

2 Literature Review

2.1 **D.M. Hargreaves, B. Kakimpa et al [2013]** :This paper examines the use of a coupled Computational Fluid Dynamics (CFD) – Rigid Body Dynamics (RBD) model to study the fixed-axis autorotation of a square flat plate. The calibration of the model against existing wind tunnel data is described. During the calibration, the CFD models were able to identify complex period autoration rates, which were a ttributable to a mass eccentricity in the experimental plate. The predicted flow fields around the autorotation plates are found to be consistent with existing observations. In addition, the pressure coefficients from the wind tunnel and computational work were found to be in good agreement. By comparing these pressure distributions and the vortex shedding patterns at various stages through an autorotation cycle, it was possible to gain important insights into the flow structures that evolve around the plat e. The CFD model is also compared against existing correlation functions that relate the mean tip speed ratio of the plate to the aspect ratio, thickness ratio and mass moment of inertia of the plate. Agreement is found to be good for aspect ratios of 1, b ut poor away from this value. However, other aspects of the numerical modelling are consistent with the correlations.

2.2 **Abdur Rahim, Mukhtar Ahmad [2014],** to enable proper design and construction of various fluid flow systems, the investigation of the actual nature of flow through them is necessary. The experimental investigation of full scale models is neither feasible nor economically viable. The small scale experimental investigation is an alternative for the requisite investigation but is a cumbersome process which requires considerable amounts of physical and monetary inputs. Further the results obtained have to be extrapolated to the actual scale of the equipment and thus a significant amount of error is generally visible due to this extrapolation. T hus, the best alternative is to use a Computational Fluid Dynamics (CFD) analysis of the system. A number of commercial CFD simulation packages are available nowadays. The best part of these packages is that they are generic in nature. If these packages gi ve sufficiently accurate simulation results, they will be a better option for solving a variety of fluid flow and heat transfer problems.

This paper is aimed at finding the suitability of the commercial CFD package in simulating internal flows in different 2D geometries viz. flow through diffuser and can-type gas turbine combustor and evaluating the performance of different turbulence models. Simulation results were validated from the respective experimental data for the geometries analyzed. The pe rformance of the different turbulence models used was also investigated for different Reynolds number values. A parametric study regarding the effect of variation of initial parameters for the Spalart-Allmaras model on the final simulation results was cond ucted. The divergence angle of combustor - annulus was also varied and its effect on the final flow characteristics at the outlet was also examined. Spalart - Allmaras model was found to be more efficient, both in terms of the number of iterations required for convergence and the closeness of the results to experimental data in case of the pipe flow. Since the k~ε model gave better results than the other models, it is used for simulating flow through the diffuser and 2D can -type gas turbine combustor.

Now in this project we quote an obstruction (sphere) between the wind tunnel and calculate that when that type of object is putted, then what affected the speed of flow. The velocity measured on the different point on the sphere then found varies and also pr essure head are varied.

Now in this project we will use the sphere and put it in the test section in wind tunnel. And we will calculate the pressure head and velocity at different point on the sphere. We will compare it with the CFD Fluent version . We will see that what will happen when we use the CFD, on our result, cost and also time.

3 Experimental Set Up And CFD Analysis

3.1 Wind Tunnel Testin g

3.1.1 Initial manometer head at different point on sphere model

h_{11}	h_{12}	h_{13}	h_{14}	h_{15}	h_{16}	h_{17}	h_{18}
Cm							
28.7	28.7	28.7	28.7	28.7	28.7	28.7	28.7

3.1.2 Final manometer head at different point on sphere model

N	h^*_{11}	h^*_{12}	h^*_{13}	h^*_{14}	h^*_{15}	h^*_{16}	h^*_{17}	h^*_{18}
%	Cm							
99.6	32.5	33.2	34	42.2	43.2	34.5	34	41.8

4 Formula's used

$$H = \left(\frac{h_1 \sim h^*_1}{100} \sin\theta_M\right) \left(\frac{\rho_m}{\rho_a} - 1\right) m$$

$$V = C_v\sqrt{(2gH)} \quad m/s$$

5 5 Calculation

5.1 Wind tunnel calculation

At point 1, on the sphere

$$H_1 = \left(\frac{28.7 \sim 32.5}{100} sin90\right)\left(\frac{804}{1.204} - 1\right)$$

$$= 25.21m$$

$$V_1 = 0.98\sqrt{(2x9.81x25.21)}$$

$$= 21.79m/s$$

At point 2, on the sphere

$$H_2 = \left(\frac{28.7 \sim 33.2}{100} sin90\right)\left(\frac{804}{1.204} - 1\right)$$

$$= 29.85\ m$$

$$V_2 = 0.98\sqrt{(2x9.81x29.85)}$$

$$= 23.72\ m/s$$

At point 3, on the sphere

$$H_3 = \left(\frac{28.7 \sim 34}{100} sin90\right)\left(\frac{804}{1.204} - 1\right)$$

$$= 35.16\ m$$

$$V_3 = 0.98\sqrt{(2x9.81x35.16)}$$

$$= 25.74\ m/s$$

At point 4, on the sphere

$$H_4 = \left(\frac{28.7 \sim 42.2}{100} sin90\right)\left(\frac{804}{1.204} - 1\right)$$

$$= 89.56m$$

$$V_4 = 0.98\sqrt{(2x9.81x89.56)}$$

$$= 41.08m/s$$

$$H_5 = \left(\frac{28.7 \sim 43.2}{100} sin90\right)\left(\frac{804}{1.204} - 1\right)$$ ERE

$$H_5 = \left(\frac{28.7 \sim 43.2}{100} sin90\right)\left(\frac{804}{1.204} - 1\right)$$

$$= 96.20\ m$$

$$V_5 = 0.98\sqrt{(2x9.81x96.20)}$$

$$= 42.27m/s$$

At point 6, on the sphere

$$H_6 = \left(\frac{28.7 \sim 34.5}{100} sin90\right)\left(\frac{804}{1.204} - 1\right)$$

$$= 38.48\ m$$

$$V_6 = 0.98\sqrt{(2x9.81x38.48)}$$

$$= 26.9m/s$$

At point 7, on the sphere

$$H_7 = \left(\frac{28.7 \sim 34}{100} sin90\right)\left(\frac{804}{1.204} - 1\right)$$

$$= 35.16\ m$$

$$V_7 = 0.98\sqrt{(2x9.81x35.16)}$$

$$= 25.73m/s$$

At point 8, on the sphere

$$H_8 = \left(\frac{28.7 \sim 41.8}{100} sin90\right)\left(\frac{804}{1.204} - 1\right)$$

$$= 86.91\ m$$

$$V_8 = 0.98\sqrt{(2x9.81x86.91)}$$

$$= 40.44m/s$$

5.2 Project work steps on CFD

5.2.1 SETUP AND SOLUTION the following sections d e-scribe the setup and solution steps for the sphere
Step 1: Creating a FLUENT Fluid Flow Analysis System in ANSYS Workbench

Step 2: Creating the Geometry in ANSYS Design Modeler

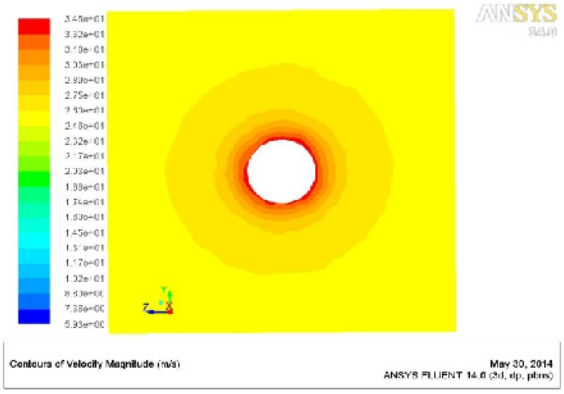
Step 3: Meshing the Geometry in the ANSYS Meshing Appl i-cation

Step 4: Setting Up the CFD Simulation in ANSYS FLUENT

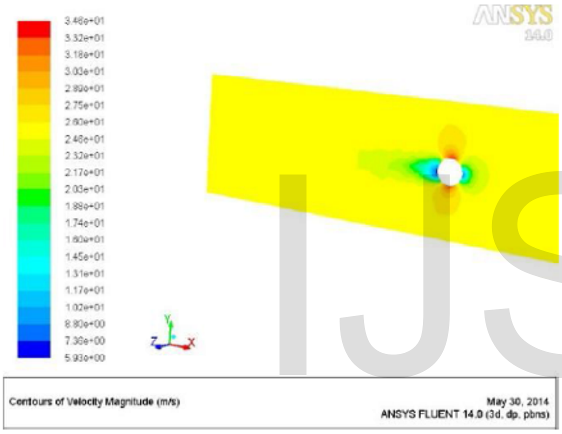
Step 5: Displaying Results in ANSYS FLUENT and CFD-Post

5.2.2 Velocity Contours by CFD

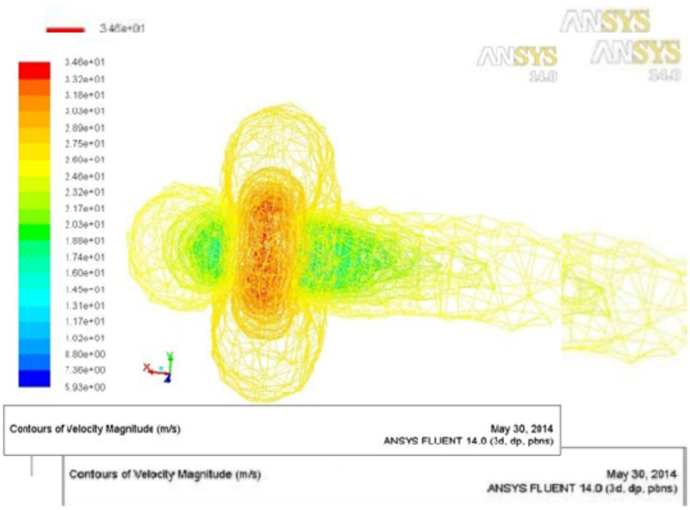
(i) Contour Plot of x Componentof V elocity



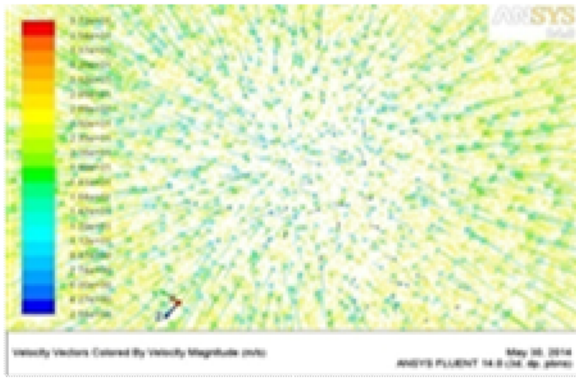
(ii) Component of Velocity along z axis



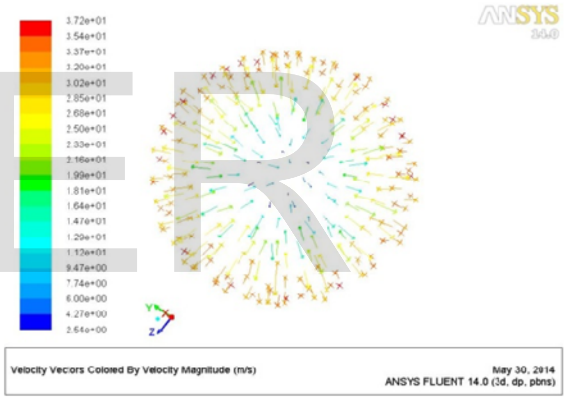
(iii)Contour Plot of with inlet sphere and 3D



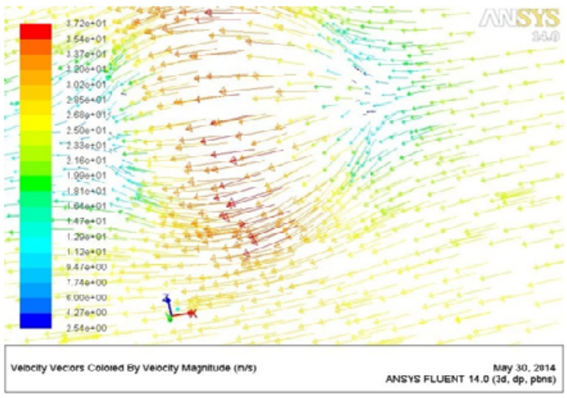
(iv)Velocity vector colored



(v)Velocity vector colored by velocity magnitude with sphere



(vi)Velocity vector colored with sphere model meshing



6. Result and Discussion

Velocity comparison at different point by wind tunnel and CFD:

The velocity was compared at different point on the sphere in wind tunnel and CFD, there was small difference between them as tabulated below

After measuring the various parameter by wind tunnel and compare it with ANSYS CFD FLUENT ,We found that there has come very little deviation between experimental and CFD calculation. The error occurs in pressure head at different point on the sphere are 0.79%, 0.83%, 0.42%, 0.36%, 0.31%,0.72%, 0.42%, 0.62% and also the velocity

1.69%, 2.42%, 0.34%, 0.55%, 0.61%, 1.67%, 1.28% and 0.84%.

From above result we can analysis that in aerodynamic system we can use the ANSYS CFD software for calculating the various parameters. If we will use this software for calculating the parameter in flow system then we can save the time and cost. CFD costs much less than experiments because physical modifications are not necessary.

S.N.	Head	Velocity by wind tunnel at various Point (m)	Velocity by CFD at various Point	[illegible] x 100 (%) Error
1	V_1	21.79	21.42	1.69
2	V_2	23.72	23.15	2.42
3	V_3	25.74	25.65	0.34
4	V_4	41.08	40.85	0.55
5	V_5	42.27	42.01	0.61
6	V_6	26.9	26.45	1.67
7	V_7	25.73	25.40	1.28
8	V_8	40.44	40.10	0.84

Table 1

7 Conclusion

Based on testing result and discussion following points are concluded

Error in result between experimental set up and CFD give small deviation .Small difference occurs by the CFD.

1. Pressure head measured by the CFD give the more close to the wind tunnel
2. Velocity also measured by the CFD give the more close to the wind tunnel.

There are more several other loss point which are responsible for creating the deviation between result such as (a) Pirated version of CFD (b) Number of iteration

8 References

[1] J.N. Abras, Enhancement of aero elastic rotor airload prediction methods, Ph.D. thesis, Georgia Institute of Technology, 2009.

[2] W.K. Anderson, R.D. Rausch, D.L. Bonhaus, Implicit/multigrid algorithms for incompressible turbulent flows on unstructured grids, Journal of Computational Physics 128 (2) (1996) 391–408.

[3] A. Antoniadis, D. Drikakis, B. Zhong, G. Barakos, R. Steijl, M. Biava, L. Vigevano, A. Brocklehurst, O. Boelens, M. Dietz, Assessment of CFD methods against experimental flow measurements for helicopter flows, Aerospace Science and Technology 19 (1) (2012) 86–100.

[4] D.L. Bonhaus, An upwind multigrid method for solving viscous flows on unstructured triangular meshes, Ph.D. thesis, University of Cincinnati, 1993.

[5] J.O. Bridgeman, G.T. Lancaster, Physics -based analysis methodology for hub drag prediction, in: Proceedings of the 66th Annual Forum of the American Helicopter Society, Phoenix, AZ, 2010.

[6] F. Champagne, C. Sleicher, Turbulence measurements with inclined hot-wires. Part 2. Hot-wire response equations, Journal of Fluid Mechanics 28 (1) (1967)177–182.

[7] F. De Gregorio, Flow field characterization and interactional aerodynamics analysis of a complete helicopter, Aerospace Science and Technology 19 (1) (2012)19–36.

[8] P. de Waard, M. Trouve, Tail shake vibration in flight – objective comparison of aerodynamic configurations in a subjective environment, in: Proceedings of the 55th Annual Forum of the American Helicopter Society, Montreal, Canada,1999, pp. 2306–2316.

[9] R.E. Gormont, Some important practical design constraints affecting drag reduction, in: Proceedings of the 31st Annual Forum of the American Helicopter Society, Washington, DC, 1975.