

ABSTRACT

The fluid in motion exerts a force on the solid body immersed in it or the solid body moving in a fluid resulting the force exerted on the solid by the fluid such as flow around an airplane, the drag force acting on automobile, trees and underwater pipelines. The Drag Force is the function of drag co-efficient which is mainly depended on the flow velocity, surface body roughness, body orientation immersed in the fluid with the direction of fluid flow and the object configuration i.e. shape and size of the object. Literature review speaks about the different shape and size of the cylinder that had been taken for the study of drag force over the cylindrical bodies. The selected shape of the cylinder that had been used for experimentation are circular cylinder, wavy cylinder, and the square shape cylinder. These all investigation over the different shape of cylinder give us quite explanation about the dependency of drag force on the cylinder and its shape by the flowing fluid with its qualitative and quantitative conclusion. However the duration of carrying these experiment is more and consuming space for large wind tunnel with constant voltage power supply to maintain the flowing velocity. The change in the power supply causes change in the flow velocity resulting error in the drag force calculation. Some literature shows numerical approach use of software like ANSYS Fluent, CFX, etc. for the calculation of drag force over body in a flowing fluid with more effective and efficient. The application of software provides opportunity for desirable change in the shape and size, surface roughness of the test object or body without extra cost or no more of new model for the experiment.

The present work is the advancement of Researcher with their research to the growing digital world. The paper shows the importance of the software and its effective work in the field of research. Thesis involves the experimentation and numerical approach for calculation of drag force over the circular cylinder of different length, different diameter and different surface roughness for different range of flow velocity. The experimentation is carried out on the Air flow bench (AF12), with the application of direct weighing method, pressure distribution method, the co-efficient of drag is obtained and for the numerical approach the ANSYS Fluent software is used for the simulation. Numerical method involve the application of Computational Fluid Dynamics (CFD). CFD follows the computational code based on Navier-Stokes equation to solve the fluid flow problem, providing satisfactory result with significant cost reduction in comparison to the experiment model. The turbulence model, k- ϵ model and Finite Volume Method (FVM) with SIMPLE spatial discretization method of second order

correction is used for the calculation of drag force over the circular cylinder of varying diameter and varying length and varying surface roughness .

Drag force, co-efficient of drag resulted for varying diameter, surface roughness of the circular cylinder is noted and a comparison graph has been plotted between experimental and numerical result. The comparison shows the numerically predicted data to be within the acceptable error range of 15% which is comparatively less than the error range of 20% as per the literature. After validating the numerical results, the numerical model was run again taking the liquid water as the flowing fluid and results obtained are shown.

Keywords: *Computational Fluid Dynamics, Finite Volume Method, ANSYS Fluent, Discretization, Drag Force, Coefficient*