

## Numerical Investigation Of The Fluid Flow Around And Past A Circular Cylinder By ANSYS Simulation

Mojtaba Daneshi<sup>1</sup>

<sup>1</sup>Department of Mechanical Engineering, Science and Research Branch, Islamic Azad University, Tehran, Iran. (E-mail: M.daneshi@srbiau.ac.ir)

---

### ABSTRACT

Analysis of the flow around objects is one of the basic problems in fluid dynamics, and is of the utmost importance. Reviews of field speed, pressure and fluid forces entered check of the object such as the force of lift and drag on the various regimes of flow is one of the goals in this area. In the present work is based on the cylinder diameter 100 Reynolds number for flow analysis slow and also based on the diameter of the cylindrical Reynolds number  $10^5$  as well as for the analysis of the flow around a cylindrical dirham has been considered. The aim of this research is to investigate the influence of Reynolds number on flow parameters and verify the correctness of the calculations carried out by the governing equations of fluid mechanics-flow. The average values of the coefficient of drag and lift coefficients, surface pressure was calculated and the validity of the results of the numerical solution of numerical solution of the results with other researchers and also with the results of laboratory work compared. Research on the hydraulic parameters of the flow around a cylinder help of ANSYS software. The results are completely compatible with the experiments. Eventually, the flow through two types of unconfined tube banks with different arrangements in low Reynolds numbers were simulated and analyzed. The mean pressure distribution is predicted reasonably well at  $Re_D = 10^5$ . However, the Reynolds number dependence is not captured, and the solution becomes less accurate at increased Reynolds numbers.

**Keywords:** Numerical, Circular Cylinders, Drag Coefficient, Fluid Flow, ANSYS.

---