

# An Estimation of the Hydrodynamic Flow Field (HDFF) around a Sphere through CFD and Experiment

Cvetelina Velkova

Department of Mechatronics

Naval Academy N. Vaptsarov, Varna, Bulgaria

Varna, Bulgaria

[cvvelkova@nvna.eu](mailto:cvvelkova@nvna.eu)

**Abstract.** In the presented study is made a numerical modelling of the hydrodynamic flow field (HDFF) around a sphere. Numerical modelling has been made using the software ANSYS Workbench and the calculation of the hydrodynamic flow field (HDFF) around a sphere was implemented using the solver ANSYS Fluent. The analysis of HDFF around the sphere has realized using Finite Element Method (FEM) available in the ANSYS Fluent environment. The HDFF has examined around the sphere at a given horizon using the real flow, the fluid is water. The obtained numerical results for velocities and pressures around the sphere have compared to actual experimental data, [1]. The aim of the presented study is to propose a numerical approach that can give adequate assessment of HDFF in a real turbulent flow around a sphere immersed in water using modern computational methods as CFD Fluent and validation of these results with real experimental data.

**Keywords:** Numerical modeling, free surface, sphere, HDFF, ANSYS Fluent

## I. INTRODUCTION

Referring to the methodology set out in item 3.3 in a source [1] for numerical determination of hydrodynamic flow field (HDFF) in a deeply immersed sphere, the present study has formulated an analytical approach to address the problem at hand. Referring to chapter 3 of source [1], in it the author presents the methodology for calculating the HDFF of the ship. Briefly, he explains that the calculation is carried out in the following sequence: 1. enter the coordinates of the corner points of the spatial quadrangular source elements describing the wetted surface of the ship's hull; 2. specify the locations of the hydrodynamic features, the streamlines of which, when the current flows around, describe the shape of the underwater part of the ship's hull; 3. a system of linear algebraic equations defining the intensities of the above-mentioned features is compiled and solved; 4 the velocities and pressures at points in the flow around the body are calculated.

In the cited source [1], the author states that in it he presents the algorithms of the programs with which he

implements the above-described methodology for numerical calculation of the HDFF of the ship's hull.

Based on the made formulation, a numerical model of turbulent flow around a sphere in ANSYS Fluent environment has developed by mixing two air and water fluids to calculate the HDFF. The main key element in the used analytical formulation of the problem according to the source [1] is the determination of the pressure field and flow velocities around a flowing sphere of infinitely ideal fluid, and in the numerical realization of the problem the liquid is real.

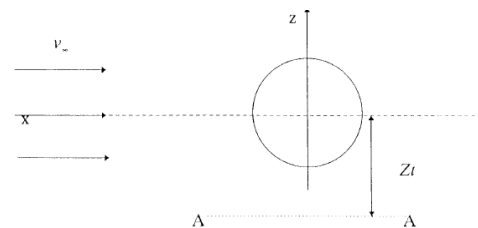


Fig. 1. Analytical formulation of the problem for determination of velocities field and pressures in points from the flow around a sphere, [1]

The assessment of the hydrodynamic flow field (HDFF) surrounding the sphere and the application of the numerical approach have conducted through an experiment [1], wherein velocities and pressures at various flow points have calculated for a sphere subjected to actual flow conditions. The experiment has conducted in accordance with the parameters outlined in the methodology section.

A different opinion on the issue related to the experimental evaluation of the HDFF around a sphere has presented in a source [6]. Quoting the author of source [1] in developing the methodology for numerical calculation of the ship's HDFF, the theoretical setup discussed in chapter two was applied. Methodology was built based on the method of Ness & Smith [88], established for bodies moving in an infinite ideal fluid, and further developed mathematical apparatus accounting for the influence of water boundaries and pressure from wave motions.

The accuracy of the model is also verified by calculating the velocities and pressures at points in the fluid

Print ISSN 1691-5402

Online ISSN 2256-070X

<https://doi.org/10.17770/etr2024vol3.8119>

© 2024 Cvetelina Velkova. Published by Rezekne Academy of Technologies.

This is an open access article under the [Creative Commons Attribution 4.0 International License](https://creativecommons.org/licenses/by/4.0/).

flow, for a clouded sphere, of an infinite ideal fluid. The theoretical computation was implemented with the help of the program "SFERA", with the problem setup shown in Fig. 1, source [1].

where into Fig. 1 from source [1]:

$V_{\infty}$  - flow velocity of flowing sphere

$Z_t$  - distance from  $x$  axis to the line A-A (per  $z$  axis)

$D$  - diameter of sphere – 320 mm

When estimating the numerically obtained values of the velocities and pressures around the sphere with the experiment, the induced velocities, and pressures around the sphere from infinite flow to an ideal fluid are determined at 25 points on the line A-A equidistant to the  $x$  axis, [1]. The pressures at the points obtained by the realized numerical approach and comparison with experimental ones were calculated when flowing the sphere from flow with velocities in infinity.

$$V_{\infty} = 2m/s; V_{\infty} = 1.5m/s; V_{\infty} = 1m/s; \\ V_{\infty} = 0.5m/s$$

The pressure coefficient for the points on A-A is determined by expression, [1]:

$$C_p = 1 - \left(\frac{V}{V_{\infty}}\right)^2 = 1 - \frac{9}{4} \sin^2 \alpha, (1)$$

a hydrodynamic pressure for the same points is determined by the following expression [1]:

$$p - p_0 = \frac{\rho V_{\infty}^2}{2} \left[1 - \left(\frac{V}{V_{\infty}}\right)^2\right] = \frac{\rho V_{\infty}^2}{2} c_p (2)$$

Building on [1], this numerical problem also incorporates insights from [7], [8], [9].

## II. METHODS

### A. Numerical Model

Geometry and dimensions of the object- sphere. The geometry of the constructed numerical 3D model of the streamlined sphere in the ANSYS Fluent environment is shown in Figure 2.

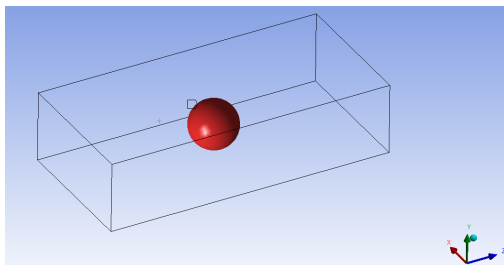


Fig. 2. The geometry of the constructed 3D model of the streamlined sphere in ANSYS Workbench environment.

The computational area (CA) of the real flow model around a streamlined sphere realized in ANSYS Fluent is constructed with the following computational dimensions shown in Fig. 3:

$R = 160mm$  - radius of a sphere

$L6 = 250mm$  - height from the center of the sphere to the free surface

$L7 = 480mm$  - height of computational area (CA)

$L8 = 460mm$  - distance from the center of the sphere to the edges of the wall of the computational area relative to the  $x$  axis.

$L9 = 920mm$  - distance from the center of a sphere to the inlet and outlet of computational area relative to  $z$  axis

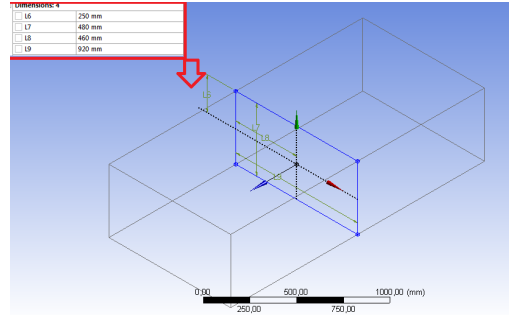


Fig. 3. Dimensions of created computational domain around the sphere.

### B. Numerical Mesh

The idea of estimating hydrodynamic flow field (HDFF) in a streamlined area using numerical methods such as ANSYS Fluent is based on the analytical and experimental approach presented in the source [1], [5]. The numerical model of the flow around the sphere uses as an idea the method proposed in source, [2], [4]. In the present study, the geometry of the streamlined sphere was modeled using the CAD software package ANSYS Workbench. The numerical simulation of the flow around a sphere is realized in ANSYS Fluent environment.

One of the main steps in developing the numerical model of the streamlined sphere is the construction of an appropriate computational mesh around the sphere. Proper estimation of hydrodynamic flow field (HDFF) of the flow around the sphere and obtaining the values of the pressure and the velocity field around it includes appropriate and accurate numerical mesh. The type of computational mesh constructed around the streamlined sphere, as well as the number of cells and nodes are shown in Fig. 4. A quarter shape of cell elements was used to accurately consider the characteristics of the real flow. The density of the computational mesh in this study are extremely important to accurately approximate and solve the equation of the fluid particle's motion, as well as Navier Stokes equations in each cell of the fluid volume describing the actual flow of water around a moving sphere.

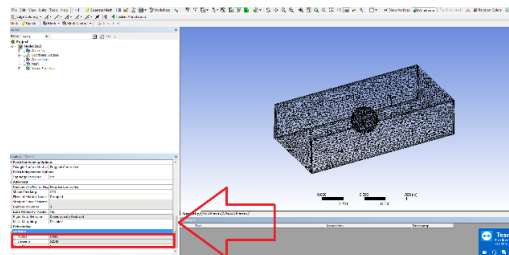


Fig. 4. Numerical model of the streamlined sphere with constructed computational mesh.

### C. Boundary Conditions and set input data

Firstly, the geometry of the sphere and the constructed fluid volume limiting the flow around it are implemented using the CAD program ANSYS Workbench, [2], [3], [4].

The computational domain around the sphere is drawn with corresponding dimensions first 2D then through the function “extrude” is obtained the 3D shape of the fluid volume limiting the flow field around the sphere. According to what is stated in source [1] and the above formulation of the problem, the sphere should be placed in the center of fluid volume, after which the boundary conditions should be applied, as shown in Figure 5. The set boundary conditions, which are required by the fact that a flow around the sphere is real flow are described mathematically as follow:

*Inlet:* Input velocity in all four cases, as follows:  $V = 0,5; 1; 1,5; 2\text{ m/s}$

*Outlet:* pressure that changes

*Sphere:* wall

*Bottom (Inlet Bottom):* wall.

*Free surface:* Pressure of 1 atm

The free surface is set as a boundary condition, i.e. it expresses the contact of the water from the fluid volume with the air, [3]. The constructed geometry of the fluid volume around streamlined sphere is in accordance with the conducted experiment and the experimental formulation of the problem set it the source [1], which was used for the comparative analysis with the obtained numerical results.

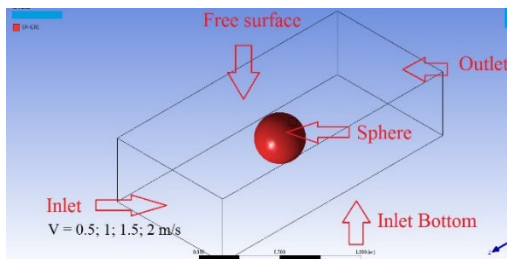


Fig. 5. Fluid domain with set boundary conditions in ANSYS Fluent environment.

The solution process, i.e. calculation of the parameters of flow around the moving sphere streamlined with real flow – water is calculated by four numerical simulations realized with ANSYS Fluent at set velocity range. A turbulence model was used to calculate the real flow around the sphere.

### III. RESULTS AND DISCUSSION

#### A. Pressure field around the sphere

The realized numerical simulation of the flowing the sphere with real liquid by means of ANSYS Fluent makes it possible to obtain the results of the values of the total and static pressure at flow points around streamlined sphere. The nature of the total pressure field curves around the sphere is shown in Fig. 7a, Fig. 7c, Fig. 7e and Fig. 7g for the four cases of the streamlined sphere i.e. RS 11,  $v=0,5$  m/s; RS 21,  $v=1$  m/s, RS 31,  $v=1,5$  m/s and for sphere RS 41,  $v=2$  m/s. The type of static pressure curves around the sphere are shown in Fig. 7b), Fig. 7d), Fig. 7f) and Fig. 7h) again for the four cases of the streamlined sphere i.e. RS 11,  $v=0,5$  m/s; RS 21,  $v=1\text{m/s}$ , RS 31,  $v=1,5$  m/s and for sphere RS 41,  $v=2$  m/s.

Velocities and velocity field around streamlined sphere. The values of the numerically calculated velocities at flow

points around the sphere, as well as the velocity field around it are shown at Fig. 6a – Fig 6c for three cases of the streamlined sphere: RS 21,  $v=1$  m/s, RS 31,  $v=1,5$  m/s and for sphere RS 41,  $v=2$  m/s in figures respectively:

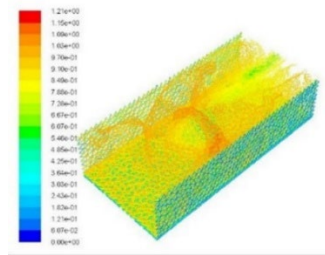


Fig. 6a. Velocities and velocity field of the liquid around the sphere RS 21  $V = 1\text{ m/s}$

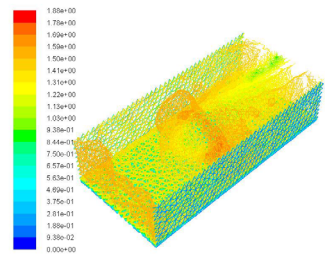


Fig. 6b. Velocities and velocity field of the liquid around the sphere RS 31,  $v=1,5$  m/s.

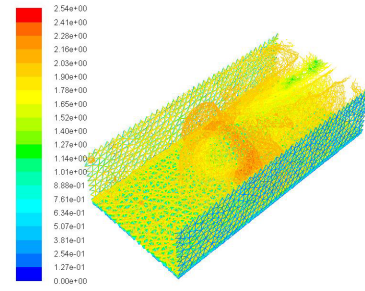


Fig. 6c. Velocities and velocity field of the liquid around the sphere RS 41,  $v=2\text{m/s}$ .

#### B. Pressure values at flow points around the streamlined sphere obtained by a real experiment.

The data on the values of the pressure at points of the flow, when the sphere moves in real liquid, according to the source [1], are given in Fig. 8.

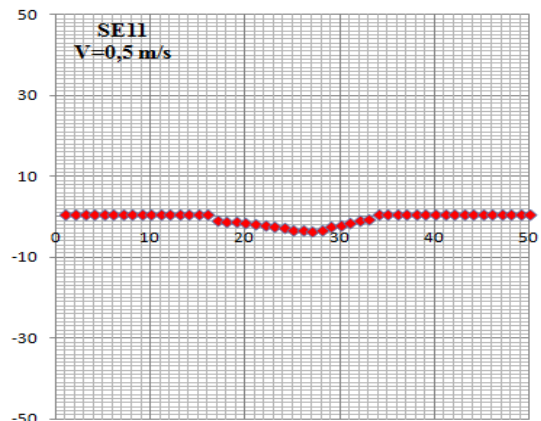


Fig. 8a. Statistical characteristics of the results obtained for the pressure by a real experiment around the sphere SE11,  $v=0,5$  m/s.

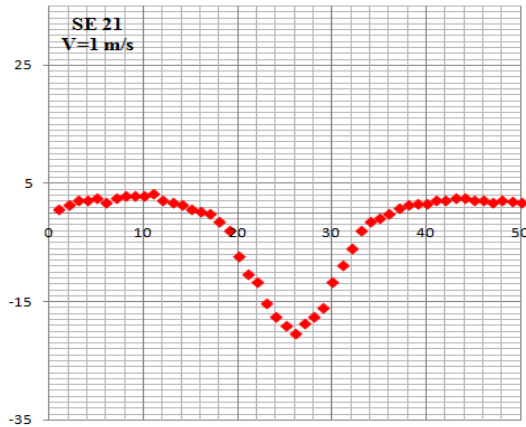


Fig. 8b. Statistical characteristics of the results obtained for the pressure by a real experiment around the sphere SE21,  $v=1$  m/s.

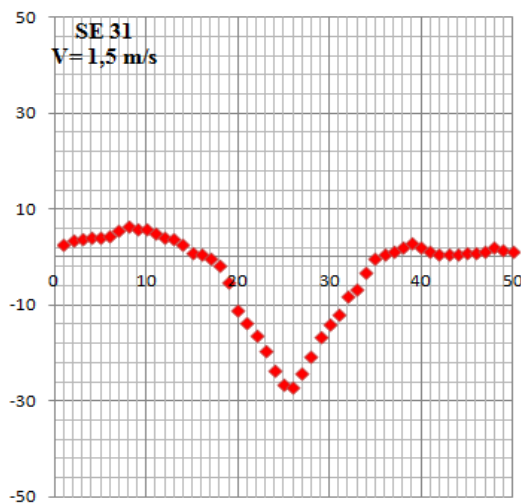


Fig. 8c. Statistical characteristics of the results obtained for the pressure by a real experiment around the sphere SE31,  $V = 1,5m / s$  .

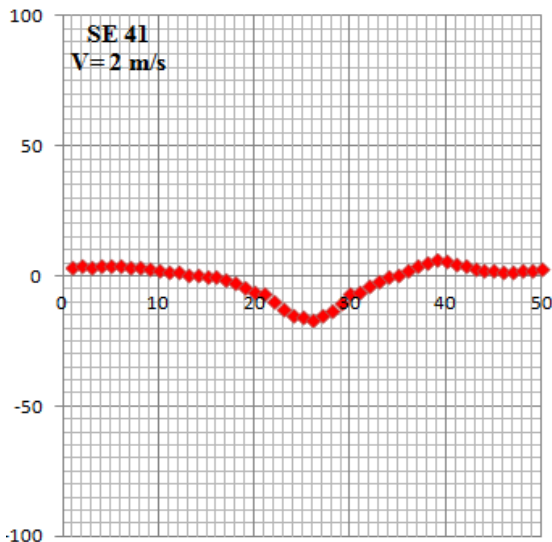


Fig. 8d Statistical characteristics of the results obtained for the pressure by a real experiment around the sphere SE41,  $v=2m/s$ .

#### IV. CONCLUSION

This study investigated the adequacy of a proposed numerical model for calculating pressure and velocity fields around a sphere in real fluid flow. The calculated pressure field using the model (Fig. 7a-h) exhibited good

qualitative agreement with experimental data (Fig. 8a-d) from [1]. This agreement suggests the model effectively captures the essential features of the pressure field around a streamlined sphere. However, minor discrepancies were observed in static pressure (Fig. 7b, d, f, h). These discrepancies are attributed to discretization errors arising from the coarseness of the computational mesh used in the simulations.

To achieve higher accuracy in future studies, refining the computational mesh is recommended, particularly near the sphere boundary. This refinement should involve a denser cell distribution and the incorporation of a well-defined boundary layer with at least ten rows of cells within the computational domain. Implementing these refinements will likely minimize discretization errors and lead to more quantitatively accurate results.

#### V. ACKNOWLEDGEMENTS

The author expresses his gratitude to Assoc. Prof. Marin Marinov from Department of Technical Mechanics, Faculty of Engineering, Naval Academy for the assistance provided during this study.

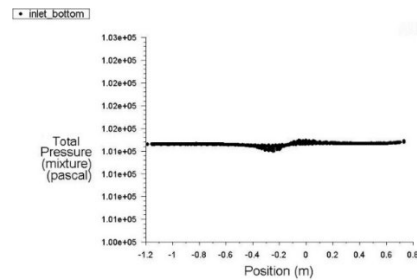


Fig. 7a. Total pressure -  $p_t$  for sphere RS 11,  $v=0,5$  m/s.

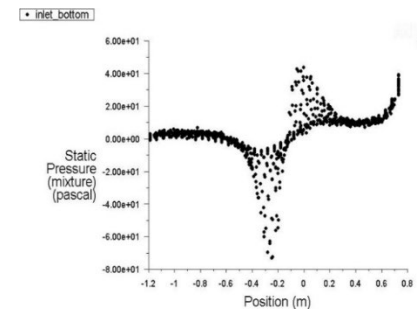


Fig. 7b. Static pressure -  $p$  for sphere RS 11,  $v=0,5m/s$ .

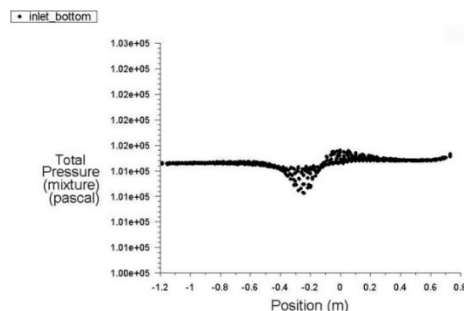


Fig. 7c. Total pressure -  $p_t$  for sphere RS 21,  $v=1$  m/s.



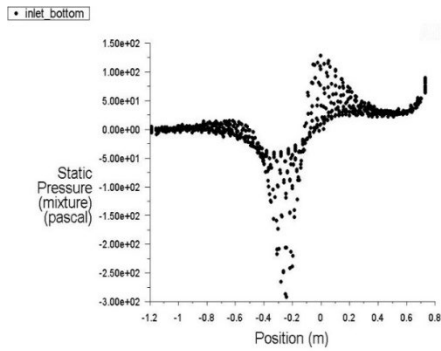


Fig. 7d. Static pressure -  $p$  for sphere RS 21,  $v=1$  m/s.

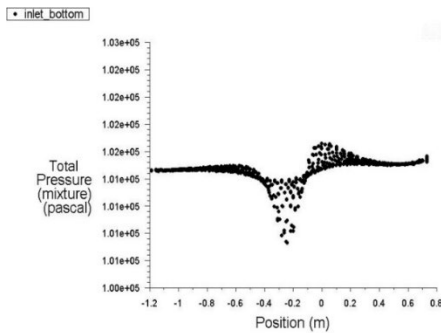


Fig. 7e. Total pressure -  $p_t$  for sphere RS 31,  $v=1,5$  m/s.

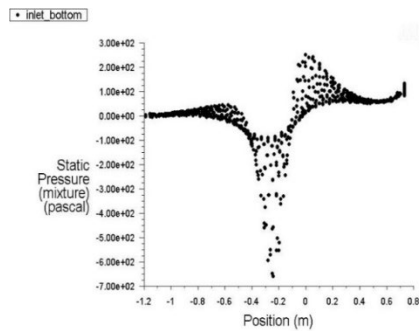


Fig. 7f. Static pressure -  $p$  for sphere RS 31,  $v=1.5$  m/s.

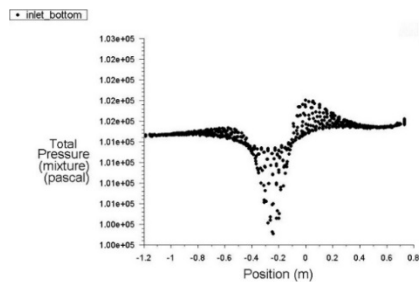


Fig. 7g. Total pressure -  $p_t$  for sphere RS 41,  $v=2$  m/s.

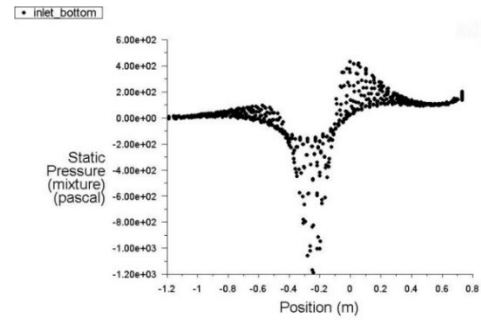


Fig. 7h. Static pressure -  $p$  for sphere RS 41,  $v=2$ m/s.

## REFERENCES

- [1] M. Marinov, Investigation and Control of the Hydrodynamic Field (HDF) of Ships – Dissertation, Nikola Vaptsarov Naval Academy, Varna, 1999.
- [2] P. Mehta, Flow over a sphere - tutorial using Ansys Fluent, 2014. [Online]. Available: <https://www.youtube.com/watch?v=8QgmO92AL70&t=23s> [Accessed: Jan. 12, 2024].
- [3] Cornell University. FLUENT - Turbulent Flow Past a Sphere - Problem Specification. B Simulation Wiki. 2009. [Online]. Available: <https://confluence.cornell.edu/display/SIMULATION/FLUENT+-+Turbulent+Flow+Past+a+Sphere+-+Problem+Specification> [Accessed: Jan. 12, 2024].
- [4] J. Blazek, Computational Fluid Dynamics: Principles and Applications, 1st Edition. May 11, 2001. eBook ISBN: 9780080545547, Chapter 5: "Simulation of Viscous Flows." 2001.
- [5] A. W. Date, Introduction to Computational Fluid Dynamics. Cambridge University Press, 8.08.2005., Chapter 9: "External Flow Simulations." 2005.
- [6] J. S. Olafsen and P. M. Chaikin, Publisher: Cambridge University Press. Experimental and Computational Techniques in Soft Condensed Matter Physics, Chapter 10: "Simulating Fluid Flow.", 2010.
- [7] K. Shyy, A. Udaykumar, T. K. Sengupta, Computational Fluid Dynamics Study on the Flow around a Sphere at Low Reynolds Numbers., Physics of Fluids, Vol. 9, Issue 11, 1997, pp. 3634-3641.
- [8] B. Kandasamy, S. Mittal, Numerical Study of the Flow around a Sphere at Low Reynolds Numbers. International Journal for Numerical Methods in Fluids, Vol. 46, Issue 10, 2004, pp. 1073-1093.
- [9] C. Tropea, M. Porfiri, H. H. Fernholz, A Comparative Study of the Flow around a Sphere at Low Reynolds Numbers, Experiments in Fluids, Vol. 45, 2008, pp. 99-107.