CFD Simulations of Drag and Separation Flow Around Ellipsoids

Yazan Taamneh*

Department of Mechanical Engineering, Tafila Technical University P. O. Box 179, 66110 Tafila, Jordan

Abstract

Computational fluid dynamics (CFD) simulations are carried out for incompressible fluid flow around ellipsoid in laminar steady axisymmetric regime (20 ≤ Re ≤ 200). The ratio of the major to the minor axis of the ellipsoid are ranged over a/b = 0.5 to 2. A commercial finite volume package FLUENT was used to analyze and visualize the nature of the flow around ellipsoids of different axis ratio. The simulation results are presented in terms of skin friction coefficient, separation angles and drag coefficient. It was found that the total drag coefficient around the ellipsoid is strongly governed by the axis ratio as well as the Reynolds number. It was observed that the Reynolds number at which the separation first occur increase with axis ratio. Separation angels and drag coefficient for special case of a sphere (AR = 1) was found to be in good agreement with previous experimental results and with the standard drag curve. The present study has established that commercially-available software like FLUENT can provide a reasonable good solution of complicated flow structures including flow with separation.

Keywords: CFD Simulation; Laminar Flow; Drag Coefficient ; Separation Angle.

© 2011 Jordan Journal of Mechanical and Industrial Engineering. All rights reserved

Nomenclature

a [m] ellipsoid major diameter in the flow direction
b [m] ellipsoid minor diameter in the direction normal to the flow
AR [-] axis ratio a/b
Cd [-] total drag coefficient
Cf [-] friction drag coefficient
Cp [-] pressure drag coefficient
P [N/m²] pressure
Re [-] Reynolds number
U_∞ [m/s] free stream velocity
Vx [m/s] x-component velocity
Vr [m/s] r-component velocity

Greek Symbols

μ [Pa.s] fluid dynamic viscosity
ρ [kg/m³] fluid density
Θs [degree] separation angle

1. Introduction

The flow separation around simple and complex bluff body is one of the most important and challenging problems in fluid mechanics. The separated flow around a body is difficult to predict and results in many undesirable phenomena such as drag increase, lift loss and fluctuations in the pressure filed, etc. The accuracy of the predicted flow field depends on model equations, numerical methods and grid spacing among other factors. Experimental investigations of the steady wake behind a sphere at low Reynolds numbers have been performed by [1,2]. They found that for Reynolds numbers less than 24 the flow around the sphere is perfectly laminar, no flow separation occurs, and the flow on the downstream side of the sphere is identical to that on the upstream side. The flow past a sphere over a larger range of Reynolds numbers have been investigated experimentally by [3,4]. They found that the flow was axisymmetric and stable up to Re = 200, while in [5] found the same behavior occurring up to Re = 210. These observations are in good agreement with the calculations of [6], who investigated the linear stability of the steady axisymmetric flow past a sphere and found that the flow undergoes a regular bifurcation at a Reynolds number of about 210 and results in the development of a non-axisymmetric wake.

The use of computational fluid dynamics codes to simulate the flow around geometrically complicated shapes such as airplanes, cars and ships has become standard engineering practice in the last few years. Therefore, several authors have developed numerical techniques for calculating viscous flow, applied them to a spheroid, and compared their predictions to the experimental results previously mentioned. The numerical work has developed from solutions of the boundary layer equations with a predetermined pressure distribution [7-
12]. Numerical studies of the fluid flow past different shape of spheroid particles over the Reynolds number range, $1 \leq Re \geq 500$ at different aspect ratio are presented by [12]. They found that the effect of shape of particles on individual and total drag coefficient was small at low Reynolds number and magnifies with increasing Reynolds number. Separation points where the boundary layer leaves the surface were not clearly considered in their study.

Direct numerical simulation based on spectral-type methods to simulate the flow between $Re = 25$ and $Re = 1000$ were carried out by [11]. Their simulations showed that the flow past a sphere is axisymmetric up to a Reynolds number of approximately 212, and that beyond this Reynolds number the flow undergoes a transition to three-dimensionality through a regular bifurcation.

There seems to be lack of computational works on flow separation around ellipsoid in axisymmetric flow regime. Therefore, this paper aims to provide a CFD simulation study of axisymmetric viscous laminar flow around ellipsoids by using commercial finite volume package FLUENT. Another sub goal of the present study is to test whether FLUENT, a commercial Computational Fluid Dynamics (CFD) software package, is capable of providing the solutions for the problem under consideration.

2. Theoretical Formulation

2.1. Governing equations

The governing equation for laminar 2D steady-state incompressible in axisymmetric geometry are the continuity equation and the two equations of motion:

$$\rho \left( \frac{\partial v_r}{\partial r} + \frac{v_r}{r} \right) = 0 \quad (1)$$

$$\rho \left[ \frac{1}{r} \frac{\partial (rv_r)}{\partial r} + \frac{1}{r^2} \frac{\partial (rv_r)}{\partial \theta} \right] = \frac{\partial p}{\partial r} + \frac{1}{r} \frac{\partial}{\partial r} \left\{ \mu \left( \frac{1}{r} \frac{\partial v_r}{\partial r} + \frac{1}{r^2} \frac{\partial v_r}{\partial \theta} \right) \right\} + \frac{1}{r^2} \frac{\partial}{\partial \theta} \left\{ \mu \left( \frac{1}{r} \frac{\partial v_r}{\partial r} + \frac{1}{r^2} \frac{\partial v_r}{\partial \theta} \right) \right\} \quad (2)$$

$$\rho \left[ \frac{1}{r} \frac{\partial (rv_r)}{\partial r} + \frac{1}{r^2} \frac{\partial (rv_r)}{\partial \theta} \right] = \frac{\partial p}{\partial r} + \frac{1}{r} \frac{\partial}{\partial r} \left\{ \mu \left( \frac{1}{r} \frac{\partial v_r}{\partial r} + \frac{1}{r^2} \frac{\partial v_r}{\partial \theta} \right) \right\} + \frac{1}{r^2} \frac{\partial}{\partial \theta} \left\{ \mu \left( \frac{1}{r} \frac{\partial v_r}{\partial r} + \frac{1}{r^2} \frac{\partial v_r}{\partial \theta} \right) \right\} \quad (3)$$

where $x$ is the axial coordinate, $r$ is the radial coordinate, $v_r$ is the axial velocity and $v_\theta$ is the radial velocity, $p$ is the static pressure, $\mu$ is the molecular viscosity, $\rho$ is the density and $\nabla \cdot \mathbf{u} = \frac{\partial v_r}{\partial x} + \frac{\partial v_r}{\partial r} + \frac{v_r}{r}$ no external body force is considered in this study.

2.2. Boundary conditions

The x-coordinate denote the direction of the bulk flow and along the major axis of ellipsoid. The r-coordinate is along the minor axis of the ellipsoid. Figure 1 shows the coordinate system for the 2-D ellipsoid model.

Since a half body section rotated about an axis parallel to the free stream velocity (axisymmetric body) is considered. The bottom boundary of the domain is modeled as an axis boundary.

The top and left boundaries of the domain are modeled as velocity inlet, the right boundary is modeled as an axisymmetric boundary condition is assumed to hold at all fluid-solid interface, i.e. at the top surface of the ellipsoid. The boundary conditions which describing the current simulated computational domain as well as the surface boundary layer is depicted in Figure 2.

3. Numerical Methods

A finite volume method is employed using a commercial software FLUENT 6.2 to solve the governing equations subject to specified boundary conditions. Since the boundary layer separation is intimately connected with the pressure and velocity distribution in the boundary layer, accurate separation point prediction are dependent on accurate resolution of the boundary layer near the surface of the body. Therefore, for the purpose of grid construction, the computational domain for ellipsoid model is divided into two regions: the boundary layer region and the free stream region (see Figure 2). The boundary layers are attached to the ellipsoid and the direction of the boundary layer grid is defined such that the grids extended into the interior of the domains. More cells are constructed near the surface of the ellipsoid to compensate the high velocity gradient in the boundary layer region of the viscous flow. A commercial software GAMBIT is used for grid generation. The coupling between the pressure and velocity fields is achieved using PISO. A second order upwind schemes is used for the convection. Here in this study, following [13], we define the total drag coefficient, $C_d$, the pressure drag coefficient, $C_p$, the skin friction coefficient, $C_f$ and a Reynolds number, $Re$ as follows:

![Figure 1. Schematic of the physical problem](image1)

![Figure 2. Solution domain and computational grid with boundary conditions and close up view of the boundary layer at $AR = 2$](image2)
\[ C_d = \frac{2D}{\rho U^2 \pi A} \cdot C_f = \frac{2 \varepsilon}{\rho U^2 \pi A} \cdot \frac{\alpha - U \cdot \tau}{\mu} \]

where \( D \) is the sum of the local skin friction and pressure drag, \( \varepsilon \) is the pressure of the stream, \( A \) is appropriate reference area and \( U \) is free stream velocity. The grid independence is achieved by comparing the results of the different grid cell size. It was found that 75000 cells is satisfactory, and any increase beyond this size would lead to an insignificant change in the resulting solution.

4. Results and Discussion

Simulation results for axisymmetric laminar flow around sphere (AR =1) are compared to experimental data to verify the validity of the CFD simulation solution. Figure 3 shows the total drag coefficient as a function of Reynolds number for special case of a sphere (AR = 1). As can be seen from Figure 3, there is an excellent agreement in the Reynolds number dependence of \( C_d \) between CFD simulations in this study and the experimental measured dependence by [7].

The effects of Reynolds number on the total drag coefficient for ellipsoids of different axis ratio are shown in Figure 4.

\[ \text{Drag coefficient, } C_d \]

Figure 3. Comparison of computed drag coefficient with the experimental correlation of Clift et al. [5] for sphere (AR = 1).

The numerical prediction of separation angle values for special case of sphere AR = 1 matched very close Rimon and Cheng [8]. Figure 6 (a-c) shows the velocity vectors around rear half of ellipsoid for different axis ratio at Re = 200. The separation region and vortex shedding are clearly visible near the rear half of ellipsoid. It can be seen that as the axis ratio increase the separation region tends to disappear. Figure 7 (a-c) shows the velocity vectors around the rear half of ellipsoid of axis ratio AR = 0.5 at various Reynolds number. It can be observed that as the Reynolds number increase the separation ring moves forward so that the attached recirculating wake widens and lengths.

\[ \text{Drag coefficient, } C_d \]

Figure 4. Variation of the total drag as a function of Reynolds number for various axis ratio.

It is clear that \( C_d \) values gradually decrease with increase in Reynolds number for all axis ratio. It can be seen that the ellipsoid of axis ratio AR = 2 exhibit the lowest drag coefficient due to the ellipsoid geometry. The simulated values of skin friction coefficient over the ellipsoid of different axis ratio at various Reynolds number is shown in Figure 5 (a-c).
Table 1. Angle of separation for viscous axisymmetric laminar flow around ellipsoids.

<table>
<thead>
<tr>
<th>Reynolds number, Re</th>
<th>AR = 0.5</th>
<th>AR = 1</th>
<th>AR = 2</th>
<th>AR = 1, AR ≥ 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>113.4454</td>
<td>No separation</td>
<td>No separation</td>
<td>No separation</td>
</tr>
<tr>
<td>40</td>
<td>105.8824</td>
<td>146.7227</td>
<td>No separation</td>
<td>145.02</td>
</tr>
<tr>
<td>100</td>
<td>98.31932</td>
<td>130.084</td>
<td>No separation</td>
<td>129.37</td>
</tr>
<tr>
<td>200</td>
<td>95.29411</td>
<td>117.9832</td>
<td>161.8487</td>
<td>116.2</td>
</tr>
</tbody>
</table>

5. Conclusions

Drag and separation flow around ellipsoid in laminar steady axisymmetric region using Computational fluid dynamics (CFD) simulations are carried out. The nature of the flow around ellipsoids of different axis ratio was visualized. The dependency of the total drag coefficient on the Reynolds number and axis ratio of ellipsoids was shown. It was found that the Reynolds number at which the separation first occur increase with axis ratio i.e. for AR ≥ 2 there may be no separation region regardless of the Reynolds number. Comparison the simulation results with the experimental data validate the commercially-available software FLUENT in providing a reasonable good solution of complicated flow structures, including flow with separation.

References