Numerical study of flow past a solid sphere at high Reynolds number

C H Yen, U J Hui, Y Y We, A Sadikin, NNordin, I Taib, K Abdullah, A N Mohammed, A Sapit, M A Razali

1 Department of Energy and Thermofluid Engineering
Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, Johor
E-mail: azmah@uthm.edu.my

Abstract. The present study gives a detail description of separation flow and its effect under high Reynolds number. The unsteady three dimensional flow simulation around sphere using numerical simulation computational fluid dynamics for high Reynolds number between 300 000 < Re < 600 000 is discussed. The separation angle and drag coefficient are also presented. The results show that the increasing Reynolds number affecting the formation of vortex shedding, separation point and drag coefficient. The agreement was good, confirming the reliability of the predicted data from computational fluid dynamic in flow analysis around sphere at high Reynolds number.

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 field. As a drag estimates to a large amount, the running costs of transport of fluids, drag reduction is a crucial design issue in engineering. In industry, these custom applications are used in internal combustion engine, chemical processing and aero industry. The nature of the flow around a sphere changes dramatically as the Reynolds number of the flow increases. Special attention is given to unsteady phenomena in aerodynamic and hydrodynamic. The flows around the sphere or the stability of the wake is the possible flow separation or boundary layer separation that occur behind the sphere will affect significantly the mass, momentum and energy transfer and to determine the values of the transport coefficient.

The visualization studies by [1] have shown that the fluid flow is attached to the sphere and there is no visible recirculation behind it at (0 ≤ Re ≤ 20). This is called attached flow region. Both theoretical arguments [2] and experimental velocity measurement demonstrate that there is a velocity defect region behind the sphere, which is the characteristic of a weak and stable wake. In this flow regime, the flow is attached on the surface of the sphere and separation does not occur. The presence and characteristics of the wake are noticeable in the asymmetry of the vortices field and to a lesser extent, the asymmetry of the streamline contours. In a steady-state wake region (20 ≤ Re ≤ 210), the wake become, wider and longer and its point of attachment on the sphere moves forward, as the value of Reynolds number increases. The separation angle for the wake, which starts at 180° at Re = 20 decreases monotonically with increasing Re to a value of approximately 120° at Re = 130. Above the
transition flow (210 ≤ Re ≤ 270), where at point near Re=210, [3] noted that the flow remained attached and stable but was no longer axisymmetric. The natural of the flow in this regime consists of two streamwises vortices tail of equal strength and opposite sign. Although the flow no longer possesses axial symmetry the flow still exhibits planar symmetry in the plane containing the two vortices tails. As the Reynolds number increases within this range a transition from the steady planar symmetric wake to a time-dependent planar symmetric wake occurs. The flow becomes unsteady flow (270 ≤ Re ≤ 1000) as the Reynolds number increases. The unsteadiness first appears as waviness in the double threaded wake [4]. The shedding of vortices started to become irregular near Re = 420 and was completely random at Re = 480 [4]. In aerodynamics, flow separation tends to increase the air drag on the front and rearsurfaces of a moving object with a pressure difference between the front and rear that will cause some drag results. Knowledge and understanding of the principles and concepts of fluid mechanics is important to analyze any fluid system is working. The reason is also much research on aerodynamic and hydrodynamic design that can slow this flow separation. Conversely, as the Reynolds Number increases, flow separation gradually develops, and this corresponds to a change from a regime of flow dominated by viscous effects, with viscous forces and pressure forces about equally important, to a regime of flow dominated by flow separation effects, with pressure forces far larger than viscous forces. Therefore, the work of [5] is extended to investigate the flow phenomena occurring around the sphere at high Reynolds Number which is 300 000 to 600 000 past through the sphere. The angle of flow separation and drag coefficient is also reported. These results can be used to capitalize on the trend in the design of all engineering applications of fluid mechanics.

2. Computational Fluid Dynamics (CFD)
The flow is simulated over a 38 mm rigid sphere in the flow direction in a 500 x 180 x 200 mm to ensure fully developed flow is achieved. The tube bundle was created in DesignModeler. Two dimensional model for the sphere were produced in CFX-PRE. The model is show in figure 1.

![Figure 1. The model](image)

The flow is simulated over a rigid sphere in the flow direction in a 500 x 180 x 200 mm to ensure fully developed flow is achieved. The tube bundle was created in DesignModeler. Two dimensional model for the sphere were produced in CFX-PRE. The computer model and the boundary conditions are shown in figure 2. Two mesh configurations of 72 000 and 158,000 cells were conducted for the grid independence test. The pressure coefficient around the sphere was analyzed. The results show there is no significant difference between the two mesh configurations as all lines of both configurations are almost overlapped. These indicate that using finer mesh does not improve the model prediction. Thus, meshing with lower number of mesh cells does not sacrifice the solution
accuracy. Since the Central Processing Unit (CPU) time increases exponentially with the number of grids, the lower mesh cells, 72 000 were chosen. Less mesh cells reduce CPU time during CFD simulation which permits a significant number of cases to be run.

The model was constructed with a grid 0.5 mm in length. The meshing gave a total of 15,000 nodes and had 72,000 elements that consisted of prisms. The sphere was set to solid surfaces with no slip. The working fluid is air. The opening boundary condition at the outlet was set to atmospheric pressure and the inlet boundary was set to a normal velocity minimum of 115 m/s – 230 m/sto give Reynolds number more in a range of 300 000 to 600 000 which is turbulence flow. An inflation layer of 1.0 mm thickness and containing 20 layers with an expansion factor of 1.2 was inserted between the tube walls and the bulk fluid to capture the effects near the wall. The transient simulation was set and SIMPLE was chosen for the pressure correction method. The turbulence model is using realizable k-epsilon because it provides superior performance for flows involving separation and the flow was assumed to be fully turbulent. The simulation was run until the residual of the pressure and velocities were less than 0.00001.

![Figure 2](image1.png)

*Figure 2. The boundary conditions (a) velocity inlet (b) pressure outlet (c) wall (d) sphere wall*

### 3. Results and discussion

#### 3.1. Flow regimes around solid sphere

Figure 3 and figure 4 shows the velocity streamline and velocity contour around the sphere for Reynolds number ranged from 300 000 to 600 000 respectively. The velocity streamline for Reynolds number 300000 until 600000 only had slightly different. The increase in Reynolds number caused circulation flow formed in the wake region.
Figure 3. The velocity streamline around the spheres (a) $Re = 300\,000$ (b) $Re = 320\,000$ (c) $Re = 340\,000$ (d) $Re = 350\,000$ (e) $Re = 400\,000$ (f) $Re = 450\,000$ (g) $Re = 500\,000$ and (h) $Re = 600\,000$
Figure 4. The velocity contour around the spheres (a) Re = 300 000 (b) Re = 320 000 (c) Re = 340 000 (d) Re = 350 000 (e) Re = 400 000 (f) Re = 450 000 (g) Re = 500 000 and (h) Re = 600 000
The near-wake re-circulating region shrinks and the wavy motion of the wake vanishes as the Reynolds number increase. The vanishing of progressive wavy motion of the wake behind the sphere is obvious at Reynolds number 400000. There were reverse flow which form circulation flow at the wake region.

3.2. Pressure around solid sphere and separation angle
The fluid particles on the midplane strike the cylinder at the stagnation point, bringing the fluid to a complete stop and thus raising the pressure at that point. It is observed that it is maximum at this point for all the Reynolds number (300 000 – 600 000). The flow separation occurs when a point near the surface of the sphere began to diverge or fall to the minimum flow. The separation point occurs when the shear stress at the wall is zero. It showed that as the Reynolds number is increases, the flow separation occurs earlier. It is due to the fact that the flow has difficulty to attach longer to the sphere as the velocity increases. This is because the inertia effects more dominant than the viscous effect as the Reynolds number increases, so that the boundary layer separates from its wall more quickly. This finding is found to be in close agreement with the past researchers [6,7,8].

3.3. Drag coefficient
The component of the resultant pressure and shear forces that acts in the flow direction is called the drag force. The drag coefficient was deduced from the drag force from the CFD simulation. The drag coefficient is shown in figure 5. From the graph it was concluded the drag coefficient decrease as the Reynolds number increase due to the flow in the boundary layer becoming turbulent, which moves the separation point further on the rear of the body, reducing the size of the wake and thus the magnitude of the pressure drag.

Figure 5. Drag coefficient versus Reynolds number
4. Conclusion
The flow behaviour around sphere is changing as the Reynolds number is increases. The flow is found to be in vortex shedding at all cases. The flow separates early as the Reynolds number increases. This finding is found to be in close agreement with the past researchers [6,7,8]. These predictions are ought to help engineers to improve the aerodynamic and hydrodynamic design application.

References
[1] Taneda S 1956 Experimental investigation of the wake behind a sphere at low reynolds number Journal of the Physics Society of Japan 11(10) 1104-1108
[2] Hinch E J 1993 The approach to steady state in oseen flow Journal Fluid Mechanics 256 601-605
[3] Margavey R H, Bishop R I 1961 Transition ranges for three-dimention wakes Can.J.Phys., 39 1418-1422
[4] Jones D A, Clarke D B 2008 Simulation of flow past a sphere using the fluent code Maritime Platform Division DSTO
[5] Sadikin A, Mohd Yunus N A, Abdullah M K, Mohammed A N 2014 Numerical study of flow past a solid sphere at moderate reynolds number Applied Mechanics and Materials 660 674-678
[6] Michaelides E E 2006 Particles, bubbles and drop, their motion, heat and mass transfer World Scientific Publishing Co. Pte. Ltd 107-116
[7] Lee S 2000 A numerical study of the unsteady wake behind a sphere in a uniform flow at moderate reynolds number Journal Computers and Fluid 29 639-667
[8] Seeley L E, Hummel R L, Smith J W 1975 Experimental velocity profiles in laminar flow around sphere at intermediate reynolds numbers Journal Fluid Mech. 68 591-608