5-2011

Evaluation of the Aerodynamic Differences of a Balloon Shape and a Sphere Using Computational Fluid Dynamic Modeling in Fluent

Daniel Burton Scholes
Utah State University

Follow this and additional works at: https://digitalcommons.usu.edu/etd

Part of the Mechanical Engineering Commons

Recommended Citation
https://digitalcommons.usu.edu/etd/872

This Thesis is brought to you for free and open access by the Graduate Studies at DigitalCommons@USU. It has been accepted for inclusion in All Graduate Theses and Dissertations by an authorized administrator of DigitalCommons@USU. For more information, please contact dylan.burns@usu.edu.
EVALUATION OF THE AERODYNAMIC DIFFERENCES OF A BALLOON SHAPE
AND A SPHERE USING COMPUTATIONAL FLUID DYNAMIC MODELING IN
FLUENT

by

Daniel B. Scholes

A thesis submitted in partial fulfillment
of the requirements for the degree

of

MASTER OF SCIENCE

in

Mechanical Engineering

Approved:

_______________________________  ________________________________
Byard Wood                        Robert Spall
Major Professor                   Committee Member

_______________________________  ________________________________
Alan Marchant                     Byron R. Burnham
Committee Member                  Dean of Graduate Studies

UTAH STATE UNIVERSITY
Logan, Utah

2011
ABSTRACT

Evaluation of the Aerodynamic Differences of a Balloon Shape and a Sphere Using Computational Fluid Dynamic Modeling in Fluent

by

Daniel B. Scholes, Master of Science
Utah State University, 2011

Major Professor: Dr. Byard Wood
Department: Mechanical and Aerospace Engineering

While tracking balloons for wind characterization, there was a question about the theoretical rise rate and corresponding coefficient of drag of a balloon shape as compared to a sphere. Since there are many studies published detailing the drag on spherical shapes, the question of whether or not a balloon can be treated as a sphere begged to be answered.

In this study we apply Computational Fluid Dynamic (CFD) modeling to compare the aerodynamic behavior and drag of a sphere to that of a balloon as it moves through fluid at Reynolds numbers from 10,000 to 100,000.

Fluent CFD models are created and used to estimate the coefficient of drag ($C_d$) vs. Reynolds number (Re) for a sphere and for a balloon shape. Details are given for the meshed model creation and the simulation methods. Sphere model results are compared to data provided in published literature. Sphere and balloon model results are compared to each other.
The results of this study show that the drag on a balloon is not statistically different from a sphere. While there are differences in the flow characteristics over the two shapes, a spherical shape is a good approximation for a balloon shape.
ACKNOWLEDGMENTS

I’d like to thank all who have helped me along the way to completing this thesis. I have been blessed more than I can ever repay, by many known and many unseen hands, who have helped to motivate me and make it possible to continue my education, even though it seemed like an impossible task.

I am very grateful to my graduate committee members, Dr. Byard Wood, Dr. Alan Marchant, and Dr. Robert Spall, who have helped to refine this document, and caused me to learn through their input and even more from their criticism.

I would especially like to thank my wonderful wife, Lorraine, for putting up with me and encouraging me for more than 20 years, but especially during the last two years as she has been left at home to manage the house and family.

My six boys have been neglected much during the past two years as I have focused on my education at the expense of family. Thank you for your patience, Jacob, Seth, Josh, Jerem, Eli, and Zack.

Daniel B. Scholes
CONTENTS

ABSTRACT .............................................................................................................. ii

ACKNOWLEDGMENTS ....................................................................................... iii

LIST OF TABLES .................................................................................................... v

LIST OF FIGURES .................................................................................................. vi

CHAPTER

1. INTRODUCTION AND PROBLEM STATEMENT ........................................ 1

2. CREATING THE CFD MODEL ................................................................. 5

   2.1. Sphere Model ............................................................................................. 5
   2.2. Balloon Model ........................................................................................... 6
   2.3. Mesh Model ................................................................................................ 6

3. RUNNING THE MODEL IN FLUENT .......................................................... 12

   3.1. Optimization of Mesh ............................................................................... 12
   3.2. Determination of Solution Method ......................................................... 13
   3.3. Run Simulation .......................................................................................... 19

4. RESULTS .......................................................................................................... 20

   4.1. Coefficient of Drag ................................................................................... 20
   4.2. Strouhal Number ...................................................................................... 23

5. DISCUSSION OF RESULTS ........................................................................... 27

   5.1. Coefficient of Drag ................................................................................... 27
   5.2. Vortex Shedding ....................................................................................... 28
   5.3. Balloon to Sphere Comparison ............................................................... 29
   5.4. Error Estimation ....................................................................................... 30

6. CONCLUSIONS ................................................................................................. 32

REFERENCES ....................................................................................................... 33

APPENDIX .............................................................................................................. 34
## LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1:</td>
<td>Mesh Model and Boundary Layer Parameters</td>
<td>11</td>
</tr>
<tr>
<td>2:</td>
<td>Boundary Layer Separation Point and Drag Components</td>
<td>16</td>
</tr>
<tr>
<td>3:</td>
<td>Results Summary of Fluent Simulation of Spheres and Balloons</td>
<td>20</td>
</tr>
<tr>
<td>4:</td>
<td>Comparison of Coefficient of Drag (this study) vs. Previous Results</td>
<td>23</td>
</tr>
<tr>
<td>5:</td>
<td>Comparison of Balloon to Sphere $C_d$</td>
<td>29</td>
</tr>
<tr>
<td>6:</td>
<td>Grid Error Calculated for Re of 10,000 and 100,000</td>
<td>31</td>
</tr>
</tbody>
</table>
## LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Balloon.</td>
</tr>
<tr>
<td>2</td>
<td>Reynolds number versus Coefficient of Drag (Cd ) for a sphere [4].</td>
</tr>
<tr>
<td>3</td>
<td>Meshed sphere model in Gambit.</td>
</tr>
<tr>
<td>4</td>
<td>Balloon shape scaled to one meter diameter</td>
</tr>
<tr>
<td>5</td>
<td>Sample y⁺ plot of balloon shape at Re=20,000.</td>
</tr>
<tr>
<td>6</td>
<td>Sphere model cross-section and detail view with polyhedral mesh.</td>
</tr>
<tr>
<td>7</td>
<td>Wall Shear Stress in the flow direction over the sphere at Re=10,000.</td>
</tr>
<tr>
<td>8</td>
<td>Wall Shear Stress in the flow direction over the sphere at Re=100,000.</td>
</tr>
<tr>
<td>9</td>
<td>Pressure in the flow direction over the sphere at Re=10,000.</td>
</tr>
<tr>
<td>10</td>
<td>Pressure in the flow direction over the sphere at Re=100,000.</td>
</tr>
<tr>
<td>11</td>
<td>Coefficient of Drag versus time for a sphere at Reynolds Numbers of 10,000, 20,000, 50,000, and 100,000.</td>
</tr>
<tr>
<td>12</td>
<td>Coefficient of Drag versus time for a balloon at Reynolds Numbers of 10,000, 20,000, 50,000, and 100,000.</td>
</tr>
<tr>
<td>13</td>
<td>Graphical comparison of calculated versus published data for a sphere.</td>
</tr>
<tr>
<td>14</td>
<td>Fast Fourier Transform of transverse velocity (in Y direction) data recorded 3.5 diameters downstream from sphere center.</td>
</tr>
<tr>
<td>15</td>
<td>Fast Fourier Transform of transverse velocity (in Y direction) data recorded 3.5 diameters downstream and .5m up from sphere center.</td>
</tr>
<tr>
<td>16</td>
<td>Fast Fourier Transform of transverse velocity (in Z direction) data recorded 3.5 diameters downstream and offset .5m offset from sphere center.</td>
</tr>
<tr>
<td>17</td>
<td>Error bar chart for balloon and sphere Cd with error bars equal to +/- 1 GCI.</td>
</tr>
<tr>
<td>18</td>
<td>Fast Fourier Transform plot of Cd for Re=20,000 sphere.</td>
</tr>
<tr>
<td>19</td>
<td>Fast Fourier Transform plot of Cd for Re=50,000 sphere.</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>21:</td>
<td>Fast Fourier Transform plot of $C_d$ for Re=100,000 sphere.</td>
</tr>
<tr>
<td>22:</td>
<td>Fast Fourier Transform plot of $C_d$ for Re=10,000 balloon.</td>
</tr>
<tr>
<td>23:</td>
<td>Fast Fourier Transform plot of $C_d$ for Re=20,000 balloon.</td>
</tr>
<tr>
<td>24:</td>
<td>Fast Fourier Transform plot of $C_d$ for Re=50,000 balloon.</td>
</tr>
<tr>
<td>25:</td>
<td>Fast Fourier Transform plot of $C_d$ for Re=100,000 balloon.</td>
</tr>
<tr>
<td>26:</td>
<td>Plot of sphere velocity magnitude at Step 1 (0.sec)</td>
</tr>
<tr>
<td>27:</td>
<td>Plot of sphere velocity magnitude at Step 2 (1.0sec)</td>
</tr>
<tr>
<td>28:</td>
<td>Plot of sphere velocity magnitude at Step 3 (2.0sec)</td>
</tr>
<tr>
<td>29:</td>
<td>Plot of sphere velocity magnitude at Step 4 (3.0sec)</td>
</tr>
<tr>
<td>30:</td>
<td>Plot of balloon velocity magnitude at Step 1 (0.sec)</td>
</tr>
<tr>
<td>31:</td>
<td>Plot of balloon velocity magnitude at Step 2 (1.0sec)</td>
</tr>
<tr>
<td>32:</td>
<td>Plot of balloon velocity magnitude at Step 3 (2.0Sec)</td>
</tr>
<tr>
<td>33:</td>
<td>Plot of balloon velocity magnitude at Step 4 (3.0sec)</td>
</tr>
</tbody>
</table>
CHAPTER 1
INTRODUCTION AND PROBLEM STATEMENT

Small helium filled balloons, as illustrated in Figure 1, are useful for measuring and tracking wind currents. As a helium balloon is released, it momentarily accelerates vertically until the drag force equals the loft force. At this point, the balloon rises with a near constant upward velocity in calm air. The loft force is affected by the mass of the balloon, its size and volume, and the purity of the helium used.

Pilot balloons (pibals) are spherical balloons, inflated with helium or hydrogen, which are manually tracked and used to characterize local wind conditions. Pibals have been used for many years by hot air balloon pilots, aviators, and meteorologists to characterize wind profiles. An optical theodolite is the traditional instrument used for tracking the ascent of a pibal. A theodolite is a manual mechanism, similar to a surveyor’s transit, used to measure azimuth and elevation angles at intervals along the pibal trajectory. ValidWind [1] is a recently developed method for tracking balloons to measure wind currents. It uses a vision tracking system with an integrated LIDAR rangefinder, inclinometer, and compass to track the trajectories of balloons.

There are several advantages to using standard balloons for wind tracking as compared to a sphere. Two obvious advantages are cost and availability. While small party balloons are available at nearly any convenience store for pennies a piece, spherical balloons are an expensive specialty item.
Stability is another advantage of the elongated balloon shape; the elongated balloon tail swings back and forth as it rises, it tends to point downward. A spherical balloon swings and rolls much more. Orientational instability creates a practical problem when a tracking target is attached to the balloon (as in ValidWind). Orientational instability creates uncertainty in the applicability of idealized dynamic results.

For interpretation of balloon tracking results it is important to understand any differences between the aerodynamics of a normal balloon shape vs. a sphere as they rise through the air. There is much published data about fluid flow around a fixed sphere, but little or no data available for the drag on a balloon with an elongated and somewhat streamlined shape. Drag measurements of actual balloons of various sizes and lift ratios have been measured and recorded at the Energy Dynamics Lab [2]. This field data can be used to validate the simulation results obtained from this study.
In Fluent [3], a computational fluid dynamics modeling program, this situation is modeled as a stationary balloon (sphere or balloon shape) with air moving past it at a constant velocity. The oscillatory motion of an actual balloon would be excessively complex to model, so a static fixed balloon is a necessary simplification for the purposes of this project.

This project evaluates the aerodynamic differences between the balloon shape and a sphere. A range of Reynolds numbers from 10,000 to 100,000 is evaluated, since this range spans the conditions for small and moderate size balloons. A model of a sphere is run in Fluent to verify the model relationship using the published relationship between $C_d$ and $Re$ for a sphere [4], as shown in Figure 2 as a baseline. Then a similar model of the balloon shape is run to compare the aerodynamics of the balloon shape vs. the sphere. The comparison will show whether the $C_d$ for the balloon shape is comparable to a sphere. Strouhal number, a measure of the vortex shedding frequency, will also be evaluated as another point of comparison between the aerodynamics of the balloon shape and sphere.
Figure 2: Reynolds number versus Coefficient of Drag ($C_d$) for a sphere [4].
CHAPTER 2
CREATING THE MODELS

2.1 Sphere Model

The basic geometry of the spherical object and surrounding fluid is created inside Gambit [5], a meshing software program commonly used for Computational Fluid Dynamics (CFD) analysis.

The 3D model geometry consists of a 1m diameter spherical object placed inside a 20m diameter flow field, as shown in Figure 3. The centers of the two spheres are offset by 3m in the X direction to allow for more downstream effects to fit within the meshed flow area. An intermediate capsule shaped volume encloses a finer mesh for the region immediate surrounding and downstream of the sphere. The capsule consists of a 4m sphere centered on the object, another sphere offset downstream by 4m, and a connecting cylinder.

![Figure 3: Meshed sphere model in Gambit.](image-url)
2.2 Balloon Model

To create a balloon model the shape was measured from an 11 inch latex balloon and recreated in a 3D CAD model. The model was then scaled from the 11 inch diameter to a 1 meter diameter. See Figures 1 and 4 for balloon details. The geometry of the 11 inch balloon is used because it is a standard party balloon size and is very common and readily available.

2.3 Mesh Model

Applying the mesh to the model geometry is probably the most critical part of the model creation. A mesh effectively breaks the model into many tiny elements all connected together. Fluent then solves the fluid equations for each element and combines them together to form a solution for the model. This requires many iterations to converge on an accurate solution.
This stage of the project was very time consuming. Various mesh densities were applied to the geometry, results were analyzed, modifications were made, and the process was repeated many times. If the model mesh is too coarse, the resulting solution is not very accurate. And when a mesh is too fine, the resulting model can be too large to run on any computer. So there is a balancing point trying to find a mesh that will give reasonably accurate results and will still run on the available computer hardware. As is true for most tasks, experience breeds intuition, and that intuition helps to narrow the choices needed to complete the task. Before coming to the mesh combination described below there were a large number of other combinations tried. Some of these meshes appeared to work well with higher Reynolds number models and others worked well with lower Re number models. The final mesh is a compromise between the many different meshes that were tried.

2.3.1 Mesh Description

The 1m sphere surface is meshed with a triangular mesh of 0.02m. A boundary layer mesh is applied to this sphere surface. The boundary layer mesh is a fine mesh applied to the surface. It is defined by a first layer thickness, a growth factor, and the number of layers.

Details of the boundary layer were optimized at each Reynolds number in order to keep y+ values close to unity. Very large or very small y+ values can lead to inaccurate results [6].

Fluent calculates the value of y+ which is a dimensionless number relating wall shear stress to the boundary mesh thickness. The formal definition of y+ is
where the friction velocity \( u^* \equiv (\tau/\rho)^{1/2} \) is wall shear stress divided by fluid density, \( y \) is the distance to the wall (or first layer thickness), and \( \nu \) is the kinematic viscosity of the fluid. Figure 5 is a sample plot of \( y^+ \) values over a balloon surface.

To control \( y^+ \), the boundary layer thickness is increased or decreased. This is accomplished by changing the first layer thickness; the growth rate and total number of layers are also modified to maintain a similar total boundary layer thickness. If a model optimized for \( \text{Re}=10,000 \) is run at \( \text{Re}=100,000 \), the \( y^+ \) of the model becomes very large. And if a model optimized for higher Reynolds number model is run at a smaller Reynolds number, the \( y^+ \) values get very small. The \( \text{Re}=10,000 \) and 20,000 simulations were run from the same model, and the \( \text{Re}=50,000 \) and 100,000 simulations were run from another model, thus requiring only two models per shape.

The capsule shaped surface (Figure 3) has a surface mesh of 0.07m applied to it. And the outer sphere has a surface mesh of 0.5m across it. These surface mesh parameters determine the size of the 3D mesh elements in the area immediately around each surface.

After the geometry is complete and meshed; walls, inlet, outlet, and fluid zones are specified inside the Gambit software environment. The model meshes can then be exported and then read into Fluent.
Volume meshes are applied to the fluid regions around the sphere. Initially, tetrahedral hybrid elements form the volume mesh. This makes approximately 2.5 million cells in the mesh; cell quantity varies depending on the boundary layer. These tetrahedral cells are later converted to polyhedral cells inside Fluent (Figure 6). The polyhedral conversion reduces the total cell count to about 700k. The benefit of the polyhedral cells as compared to the tetrahedral cells is that the CFD model converges faster. The information contained within the 2.5 million (tetrahedral) cell volume mesh model is condensed into a smaller number of cells with the same information.

Table 1 lists the mesh parameters for each the Fluent models.
Figure 6: Sphere model cross-section and detail view with polyhedral mesh.
Table 1: Mesh Model and Boundary Layer Parameters

<table>
<thead>
<tr>
<th>Shape</th>
<th>Mesh Type</th>
<th>Solver</th>
<th>Re</th>
<th>Time Step (s)</th>
<th>Max $\gamma^+$</th>
<th>Object Surface Mesh (m)</th>
<th>Capsule Surface Mesh (m)</th>
<th>Outer Surface Mesh (m)</th>
<th>Total Cell Count</th>
<th>Boundary Layer</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1st BL Thickness (m)</td>
</tr>
<tr>
<td>Sphere</td>
<td>PolyHedra</td>
<td>LES</td>
<td>10k</td>
<td>0.01</td>
<td>0.769</td>
<td>1.376</td>
<td>0.737</td>
<td>1.363</td>
<td>6.13E+05</td>
<td>0.241</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>20k</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>6.96E+05</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>50k</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>100k</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Balloon</td>
<td>PolyHedra</td>
<td>LES</td>
<td>10k</td>
<td>0.01</td>
<td>0.785</td>
<td>1.360</td>
<td>0.734</td>
<td>1.350</td>
<td>6.45E+05</td>
<td>0.241</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>20k</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>6.50E+05</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>50k</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>100k</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
CHAPTER 3
RUNNING THE MODEL IN FLUENT

3.1 Optimize Mesh

During early trials, meshes as coarse as 100k cells were tried, but the results did not match the published data for a sphere [7,8]. These coarse meshes ran well on a single or quad core personal computer, but as the mesh was refined, simulation became extremely slow. It became necessary to perform the simulation on a multi-node cluster of computers. Fluent has the ability to be run with multiple parallel processors; this increases the speed of the calculations considerably. Most of the computers available in the computer lab have two dual core processors, which allow Fluent to run on 4 parallel processors on a computer simultaneously. While the 4 parallel processor mode is faster than the single processor mode, a larger multi-core processing system is needed to be able to solve meshes larger than 250,000 cells.

Fortunately we have access to the High Performance Computing Center (HPC) here at USU where there are multiple multi-node computing clusters set up for this type of computing.

The HPC worked well for this project. Originally all models were run on 16 parallel nodes, but were later run on 32 nodes. One drawback to the HPC is that all runs must be completed in a 24-hour period. To work around this, data was reviewed every day, and the run was resubmitted for multiple days to accumulate the necessary simulation time. With the larger 2.5 million cell models, about 5000 iterations, equivalent to 50 seconds of simulation within the model, could be run in a day. A quantity of 4000 to 5000 iterations were required to get the model to initially converge to a fully
developed flow pattern, and then the model needed to be run 3 or 4 more days to get 150 or 200 seconds of dynamic simulation data for each model.

Optimizing the boundary layer thickness to get acceptable \( y^+ \) values with every change in Reynolds number was challenging and time consuming. The boundary layer thickness is increased or decreased to adjust the \( y^+ \) value, but the \( y^+ \) value also changes as the model converges. Often what appeared to be a good value at the beginning of a simulation run, turned out to be a very low value after the solution had converged to a fully developed flow pattern.

3.2 Solution Method Selection

3.2.1 Steady vs. Transient

Over our full range of Reynolds number (10,000 - 100,000) the aerodynamic behavior of both the sphere and the balloon shape is transitional with unsteady flow and vortex shedding. Flow across the upstream surface of the sphere is laminar, but the boundary layer separates and turbulence forms on the downstream side and vortices are shed around the aft end. The flow is not axisymmetric, so 2D simulation is not realistic.

Simulations of the sphere model were tried with the Fluent model set as “Steady.” Results of this “Steady” model were not close to published results, especially at the higher end of the Reynolds number models. Running the model in “Transient” mode provided improved results.

As can be seen from the variation of the coefficient of drag over time, (see section 4.1) the model does not reach a steady-state condition where drag forces remain constant. Due to the shedding of vortices the fluid forces are continuously changing. Because of the continually changing nature of this problem, Fluent gives the option of running the
model as “Transient.” A time step is used and a fixed number of iterations per time step is specified. The software steps through time incrementally as the model is solved.

3.2.2 LES vs. other Turbulent Solvers

Fluent provides alternative algorithms to solve for turbulent flow. Comparing results for the sphere models using different turbulence models inside Fluent, Large Eddy Simulation (LES) gives the most consistent results across the flow range as compared to published results for coefficient of drag. While Detached Eddy Simulation (DES) provided reasonably good results at Re=10,000, it was low by a factor of 2 at RE=100,000. Other solvers were also tried but none of those used provided consistently good results when compared to the published results. Two Reynolds Averaged Navier-Stokes (RANS) methods used are: K-epsilon (K-E) and Reynolds Stress Model (RSM). Both K-E and RSM were used to compare with LES and DES models at Re=10,000. Only RSM, DES and LES were used at Re=100,000. The following paragraphs describe how we compared and validated the results.

One way to validate our results is to compare the quantity of interest “C_d” with other published results. Additional insight is gained by considering the physics behind the drag on a sphere.

Considering flow over a sphere, a boundary layer forms along the surface as the fluid moves across it. Over our range of Re, this boundary layer is laminar. The boundary layer remains attached to the surface as long as the wall shear stress remains positive, or the normal velocity gradient remains positive. At the point where the shear stress becomes zero, the boundary layer separates from the surface. This separation point defines the downstream wake size and affects the pressure gradient in the wake region of
the object. So we can detect the separation point by plotting wall shear stress in the
direction of the flow along the object.

Considering drag forces, there are two components acting on the sphere surface;
viscous forces and pressure forces. These added together equal the drag force.

\[ F_d = F_p + F_v \]  \hspace{1cm} (2)

The definitional relationship between \( F_d \) and \( C_d \) is;

\[ F_d = C_d \left( \frac{1}{2} \rho v^2 A \right) \]  \hspace{1cm} (3)

where \( \rho \) is the density of the fluid, \( v \) is the fluid velocity, and \( A \) is the cross-sectional area
of the object. Similarly we may define coefficients of drag for the contributions of
pressure (\( C_p \)) and viscous forces (\( C_v \)).

\[ C_d = C_p + C_v \]  \hspace{1cm} (4)

Figures 7 and 8 plot the absolute value of wall shear stress for sphere simulations
at Re=10,000 and 100,000. Boundary layer separation is indicated where this stress
approaches zero. Note that at Re=10,000 (Figure 7) both DES and LES have separation
points close to 90 degrees. But KE and RSM are considerably farther along the sphere
surface. At Re=100,000 (Figure 8) RSM and DES have separation points farther
downstream than LES, possibly indicating that the boundary layer has turned turbulent in
those models.
Figure 7: Wall Shear Stress in the flow direction over the sphere at Re=10,000.

Figure 8: Wall Shear Stress in the flow direction over the sphere at Re=100,000.
Table 2: Boundary Layer Separation Point and Drag Components

<table>
<thead>
<tr>
<th></th>
<th>Separation Pt (deg)</th>
<th>( C_p )</th>
<th>( C_v )</th>
<th>( C_d )</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Re=10,000</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Published (7)</td>
<td>88</td>
<td></td>
<td></td>
<td>0.387</td>
</tr>
<tr>
<td>LES</td>
<td>90</td>
<td>0.401</td>
<td>0.045</td>
<td>0.447</td>
</tr>
<tr>
<td>DES</td>
<td>88</td>
<td>0.432</td>
<td>0.039</td>
<td>0.471</td>
</tr>
<tr>
<td>K-E</td>
<td>114</td>
<td>0.962</td>
<td>0.080</td>
<td>1.042</td>
</tr>
<tr>
<td>RSM</td>
<td>142</td>
<td>0.699</td>
<td>0.132</td>
<td>0.831</td>
</tr>
<tr>
<td><strong>Re=100,000</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>LES</td>
<td>89</td>
<td>0.388</td>
<td>0.025</td>
<td>0.413</td>
</tr>
<tr>
<td>DES</td>
<td>102</td>
<td>0.180</td>
<td>0.020</td>
<td>0.200</td>
</tr>
<tr>
<td>RSM</td>
<td>101</td>
<td>0.658</td>
<td>0.073</td>
<td>0.731</td>
</tr>
</tbody>
</table>

Table 2 shows values for the location of separation as compared to published values [9]. By comparing the boundary layer separation point between different solvers we see that accurate simulation of the separation point corresponds with improved accuracy in the \( C_d \) measurement. Since no published separation points were found for the Reynolds number 100,000 case we have only the \( C_d \) values to compare. From the Table we can see that LES is much more consistent for both ends of our Reynolds number range.

We also see from Table 2, that \( C_p \) is the primary component of the drag. \( C_v \) accounts for only about 10% of the total drag. Pressure plots are quantitatively different for each method. In Figure 9 at Re= 10,000, LES and DES match very closely, but the K-epsilon (K-E) and Reynolds Stress Model (RSM) show a much higher pressure profile, resulting in increased values of drag.
For Re= 100,000 (Figure 10), the RSM model again has a higher pressure profile resulting in high drag values. DES on the other hand has a low average pressure across the sphere and results in a very low drag value. LES has a pressure profile very similar to its lower Re pressure profile which results in a drag value that is nearly identical to the lower flow drag.

Figure 9: Pressure in the flow direction over the sphere at Re=10,000.

Figure 10: Pressure in the flow direction over the sphere at Re=100,000.
Based on agreement with the baseline $C_d$ dependence and known characteristics of wake separation, the LES solver was used for all the comparative simulations of the sphere and the balloon shape.

3.3 Run Simulation

The models were run in Fluent as Transient (time dependent) with a time step of 0.01s. All models were run with an object diameter of 1m, density of the fluid was set to 1kg/m$^3$, and velocity was set to 1m/s. To adjust the Reynolds number,

$$Re = \frac{\rho v D}{\mu}$$  \hspace{1cm} (5)

only the viscosity was changed to make corresponding Reynolds numbers of 10,000, 20,000, 50,000, and 100,000.

Following are the other parameters set in Fluent:

- Boundary Conditions: Sphere is a stationary, wall/no-slip; Outer Sphere is a Velocity Inlet with magnitude and direction set at 1m/s and in the “X” direction.
- Models: Large Eddy Simulation, Smagorinsky-Lilly SGS.
- Solution Methods: Pressure-Velocity coupling scheme: SIMPLEC, Momentum bounded central differencing.
- Solution Controls: under-relaxation factors: Pressure 0.9, Momentum 0.5.
- Reference Values: Area, density, length, velocity, and viscosity (adjusted to control Re) values are input for calculation of $C_d$ and other calculated values.
- Run Calculation: Max iterations per time step: 12, time step size: 0.01 seconds.
CHAPTER 4
RESULTS

4.1 Coefficient of Drag

The data obtained from the simulations include the $C_d$ plotted over time. This dynamic data is plotted in Figures 11 and 12 for each Reynolds number and for both the balloon and the sphere models. From the $C_d$ data we are able to look at the dynamical results statistically. Table 3 shows the mean value of the drag coefficient with its maximum, minimum peak values and standard deviation for each case.

Table 4 compares the sphere data from this study to curve fits from two analyses of previously published data [7, 8]. Figure 13 plots the comparison.

Table 3: Results Summary of Fluent Simulation of Spheres and Balloons

<table>
<thead>
<tr>
<th></th>
<th>RE</th>
<th>$C_d$ (mean)</th>
<th>Max</th>
<th>Min</th>
<th>Std Dev $\sigma$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sphere</td>
<td>10k</td>
<td>0.4440</td>
<td>0.5097</td>
<td>0.3992</td>
<td>0.0180</td>
</tr>
<tr>
<td></td>
<td>20k</td>
<td>0.4526</td>
<td>0.5587</td>
<td>0.4061</td>
<td>0.0230</td>
</tr>
<tr>
<td></td>
<td>50k</td>
<td>0.4489</td>
<td>0.5198</td>
<td>0.4047</td>
<td>0.0205</td>
</tr>
<tr>
<td></td>
<td>100k</td>
<td>0.4288</td>
<td>0.5387</td>
<td>0.3794</td>
<td>0.0255</td>
</tr>
<tr>
<td>Balloon</td>
<td>10k</td>
<td>0.4359</td>
<td>0.5972</td>
<td>0.3822</td>
<td>0.0288</td>
</tr>
<tr>
<td></td>
<td>20k</td>
<td>0.4329</td>
<td>0.5722</td>
<td>0.3864</td>
<td>0.0260</td>
</tr>
<tr>
<td></td>
<td>50k</td>
<td>0.4318</td>
<td>0.5864</td>
<td>0.3710</td>
<td>0.0341</td>
</tr>
<tr>
<td></td>
<td>100k</td>
<td>0.4072</td>
<td>0.4945</td>
<td>0.3466</td>
<td>0.0241</td>
</tr>
</tbody>
</table>
Figure 11: Coefficient of Drag versus time for a sphere at Reynolds Numbers of 10,000, 20,000, 50,000, and 100,000.
Figure 12: Coefficient of Drag versus time for a balloon at Reynolds Numbers of 10,000, 20,000, 50,000, and 100,000.
4.2 Strouhal Number

The Strouhal number is a dimensionless number used to describe oscillations in the flow around an object.

\[ St \equiv \frac{fL}{V} \quad (6) \]
where $f$ is the vortex shedding frequency, $L$ is the characteristic length (object diameter), and $V$ is the velocity relative to the fluid. The vortex shedding frequency can generally be found from a Fourier Transform of the $C_d$ data recorded against time.

To evaluate the Strouhal number from the recorded coefficient of drag data, a Fast Fourier Transform (FFT) was applied to the data. From this the Power Spectrum Density (PSD) was plotted against frequency. The frequency enclosing 50% of the spectral power was assigned as the characteristic Strouhal frequency.

The FFT plots of the drag data did not show any dominant frequencies, but appear to be more chaotic. But since plots of the velocity profile (see figures in the appendix) show a more periodic shedding of vortices. Therefore, data from down-stream transverse velocities were evaluated. Several sampling points were added to the model and data was recorded at the downstream points for velocity in the Y and Z directions, which are transverse to flow in the X direction. Figures 14, 15, and 16 show examples of the FFT from points three diameters downstream from the sphere at Re=100,000. The first point is directly downstream on the X axis, the second point is offset by 0.5m in the Y direction, and the 3\textsuperscript{rd} point is offset 0.5m in the Z direction. In Figure 14, a clear peak in the PSD is coincident with the 50% cumulative power point and at a Strouhal number of 0.2. In Figure 15, there is also a peak at St=0.2 that coincides with 50% cumulative power point. But in Figure 16, where we are evaluating the transverse velocity in the Z direction, the peak falls at approximately St=0.4, and the 50% cumulative power point is at 0.35.

Looking at all three plots the first looks like a textbook example for the Strouhal number, the second still appears to give a similar result, where the third plot indicates less surety in the method.
Figure 14: Fast Fourier Transform of transverse velocity (in Y direction) data recorded 3.5 diameters downstream from sphere center.

Figure 15: Fast Fourier Transform of transverse velocity (in Y direction) data recorded 3.5 diameters downstream and .5m up from sphere center.
Figure 16: Fast Fourier Transform of transverse velocity (in Z direction) data recorded 3.5 diameters downstream and offset .5m offset from sphere center.
CHAPTER 5
DISCUSSION OF RESULTS

5.1 Coefficient of Drag

Most published results show the coefficient of drag for a sphere at Re=10,000 to be at or near 0.4. From there $C_d$ increases to a broad maximum between 0.45 and 0.50 around Re=100,000 (Figure 2). As the curve continues, it drops off again as critical turbulence is reached at Re ~ 200,000. The simulation results in this study show $C_d$ decreasing slightly between Re=10,000 and Re=100,000. The curves from both Cheng [7] and Clift [8] show a slight positive slope over this range. The calculated value of $C_d$ at 100,000 may increase with more mesh refinement: in this study we observed that $C_d$ increased slightly with every refinement of the mesh at Re=100,000. But the opposite was true at Re=10,000. The coarser mesh tended to give higher $C_d$ values, but with refinement, the $C_d$ decreased slightly. Therefore it is possible that additional mesh refinement would change the $C_d$ slope from negative to positive, and more closely match the published results. Table 6, in section 5.3, shows a comparison of the coarse and fine mesh results, which support this conclusion.

We also observed some differences between simulation results using polyhedral vs. tetrahedral meshes. Polyhedral meshes created from the tetrahedral meshes ran faster and converged quicker (less time per iteration). But drag values were not always consistent with the two different meshes. There may have been some loss of accuracy between the tetrahedral to polyhedral mesh conversion. This is an impression based on running multiple versions of similar mesh models but more studies are needed. For the
purpose of this study, the model accuracy was sufficient to compare the balloon and the sphere.

5.2 Vortex Shedding

Strouhal numbers associated with the coefficient of drag simulation data are considerably lower than those described in much of the literature. A value $St=0.2$ is considered a nominal value for flow over a sphere. The data from $C_d$ in this study suggest that it is in the range of 0.029 to 0.07. This difference is primarily due to inconsistency in characterizing $St$ for chaotic systems. In some papers [10] $St$ appears to be selected at a local peak of the FFT. No dominant, isolated peaks exist in the plots of the $C_d$ data for the current simulations; the variability in $C_d$ is not consistent and periodic, but random or chaotic. This condition led us to propose an objective definition of $St$ as the frequency enclosing 50% of the signal power.

Strouhal number is commonly used to describe the rate of vortex shedding. To investigate this phenomenon, we evaluated the time dependence of transverse velocities at points in the downstream wake of the sphere. Various points in the downstream wake were monitored, and velocities in the Y or Z (transverse) directions were recorded in a new simulation run at $Re=100,000$. For a point 3m directly downstream from the sphere, there was a strong peak at $St=0.2$. The peak also coincided with the 50% power point on the FFT plot (Figure 14). This result demonstrates that the dominant frequency for vortex shedding is at $St\sim0.2$ and thus provides independent validation of our CFD modeling of the sphere.
5.3 Balloon to Sphere Comparison

The primary purpose of this study is to compare the drag on a balloon shape to the drag on a sphere of the same diameter.

In comparing the differences in flow between the sphere and balloon models, we find a consistent offset in the mean values of the coefficient of drag: mean drag of the balloon is less for all values of Re as compared to the corresponding sphere models. The difference is not large, but it is less for all cases. As shown in Table 5, the largest difference is only 2 percent.

Table 5: Comparison of Balloon to Sphere $C_d$

<table>
<thead>
<tr>
<th>RE</th>
<th>Balloon $C_d$</th>
<th>Sphere $C_d$</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>10,000</td>
<td>0.4359</td>
<td>0.4440</td>
<td>0.81%</td>
</tr>
<tr>
<td>20,000</td>
<td>0.4416</td>
<td>0.4526</td>
<td>1.10%</td>
</tr>
<tr>
<td>50,000</td>
<td>0.4318</td>
<td>0.4489</td>
<td>1.70%</td>
</tr>
<tr>
<td>100,000</td>
<td>0.4072</td>
<td>0.4288</td>
<td>2.16%</td>
</tr>
</tbody>
</table>

In comparing the boundary layer separation point between the different models, the separation point was nearly 90 degrees for all of the sphere simulations. The same separation point was observed for the balloon models. If the separation points had been different, the drag values probably would not have agreed so closely.

In the Appendix (A2) (Figures 26-33) are contour plots of the velocity magnitude across a section of the models over time (1.0s interval, 3s total). This helps to visualize the turbulent nature of the flow in the wake of the balloon and sphere. The flow patterns are different between the sphere and balloon. The wake behind the sphere shows more pronounced side to side movement. The balloon wake does not display significant side to side motion and is not as wide. This may be related to the more streamlined shape of the
balloon on the downstream side. The sphere has a much more blunt tail profile, creating a more turbulent wake behind it.

5.4 Error Estimation

For CFD problems there is a standard method for determination of error associated the model and method used to solve the problem. We follow the formulation outlined by Roache [11].

To implement the error estimation method, we must run two separate meshed models. The first model is the refined mesh used to solve our problems. The second model is identical except that it has a coarser mesh with reduced grid spacing. We chose to increase the spacing by a factor of two. The first boundary layer thickness was also increased by two times.

The computation error estimate associated with the refined grid is:

\[ E_1 = \frac{f_2 - f_1}{1 - r^p} \]  \hspace{1cm} (7)

where \( f_2 \) and \( f_1 \) are model results from coarse and fine grid models, respectively; \( r \) is the grid spacing ratio (\( r=2 \) if the refined grid spacing is \( \frac{1}{2} \) of the unrefined mesh); and \( p \) is the formal order of accuracy of the algorithm used (for LES, \( p=2 \)).

A Grid-Convergence Index (GCI) is defined as:

\[ GCI = F_s \ E_1 \]  \hspace{1cm} (8)
where $F_s$ is a safety factor and $E_1$ is the error estimation for the fine grid from equation (7). Roach [11] recommends a safety factor of 3. $E_1$ and GCI for two of our spherical cases are shown in Table 6.

GCI is interpreted as a conservative estimate of the absolute modeling error. Figure 17 graphically summarizes our $C_d$ estimates with error bars equal to ±GCI (appropriate values from Table 6). Note that the current simulation results are consistent with baseline results (i.e. from Cheng [7]) across the full range of Re.

Table 6: Grid Error Calculated for Re of 10,000 and 100,000

<table>
<thead>
<tr>
<th>RE</th>
<th>Sphere $C_d$ Coarse</th>
<th>Sphere $C_d$ Fine</th>
<th>$E_1$</th>
<th>GCI</th>
</tr>
</thead>
<tbody>
<tr>
<td>10,000</td>
<td>0.4732</td>
<td>0.4440</td>
<td>-0.0097</td>
<td>0.0292</td>
</tr>
<tr>
<td>100,000</td>
<td>0.3555</td>
<td>0.4288</td>
<td>0.0244</td>
<td>0.0733</td>
</tr>
</tbody>
</table>

Figure 17: Error bar chart for balloon and sphere $C_d$ with error bars equal to +/- 1 GCI.
CHAPTER 6
CONCLUSIONS

- $C_d$ of a balloon shape is nearly the same as that of a sphere. Considering the error in the model and the simulation method, the balloon and sphere are identical statistically.

- $C_d$ as a function of time varies chaotically, without strong periodicity.. Semi-periodic vortex shedding is observable in the transverse down-stream fluid velocities, with $St \sim 0.2$.

- Although flow characteristics around the balloon shape are not identical to a sphere. Pressure forces and net drag are similar, and boundary layer separation occurs in a similar location between the two different geometries. But wake shape is visibly different.

- Based on the results of this study, the balloon shape acts very much like a sphere. For the purpose of tracking balloons through the air, treating the balloon as a sphere is a valid approximation.
REFERENCES


A1. Frequency Content of $C_d$ Variations

Figures 21 through 25 show the Fast Fourier Transform plot used to calculate the Strouhal number. 50% power is the point where half of the total power spectrum summation is located. This point indicates the calculated Strouhal number from $C_d$.

![Figure 18: Fast Fourier Transform plot of $C_d$ for Re=10,000 sphere.](image18)

![Figure 19: Fast Fourier Transform plot of $C_d$ for Re=20,000 sphere.](image19)
Figure 20: Fast Fourier Transform plot of $C_d$ for Re=50,000 sphere.

Figure 21: Fast Fourier Transform plot of $C_d$ for Re=100,000 sphere.
Figure 22: Fast Fourier Transform plot of $C_d$ for $Re=10,000$ balloon.

Figure 23: Fast Fourier Transform plot of $C_d$ for $Re=20,000$ balloon.
Figure 24: Fast Fourier Transform plot of $C_d$ for Re=50,000 balloon.

Figure 25: Fast Fourier Transform plot of $C_d$ for Re=100,000 balloon.
A2. Visualization of Flow around Objects

Figures 26 through 29 show cross sectional plots of velocity magnitude of a sphere at time intervals of 1.0 seconds. This is useful to see vortex shedding and general flow patterns around the object.

Figure 26: Plot of sphere velocity magnitude at Step 1 (0.sec).

Figure 27: Plot of sphere velocity magnitude at Step 2 (1.0sec).
Figure 28: Plot of sphere velocity magnitude at Step 3 (2.0sec).

Figure 29: Plot of sphere velocity magnitude at Step 4 (3.0sec).
Figures 30 through 33 show cross sectional plots of velocity magnitude of a balloon at time intervals of 1.0 seconds. This is useful to see vortex shedding and general flow patterns around the object.

**Figure 30:** Plot of balloon velocity magnitude at Step 1 (0.0 sec).

**Figure 31:** Plot of balloon velocity magnitude at Step 2 (1.0 sec).
Figure 32: Plot of balloon velocity magnitude at Step 3 (2.0Sec).

Figure 33: Plot of balloon velocity magnitude at Step 4 (3.0sec).