

# Aerodynamic Study of a Ballute Using Computational Fluid Dynamics

Anthony R. Mastromarino III

Tagliatela College of Engineering - Mechanical Engineering

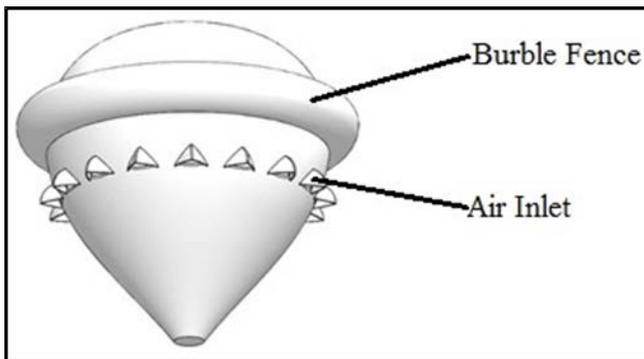
Dr. Maria-Isabel Carnasciali – Assistant Professor, Mechanical Engineering

## Abstract

Computational Fluid Dynamics (CFD) simulations were conducted to study the air flow around an aerospace grade deceleration device meant for atmospheric re-entry and landing on planet surfaces. The particular device in this study is called a ballute, which has a shape like that of a hot air balloon combined with a parachute. One variant is presently being tested by NASA's Jet Propulsion Laboratory for the purpose of opening a larger parachute which decelerates a heavy payload. This research focuses on the air flow around the air inlets of the ballute, and then varying its design to allow for localization of the air inlets. By using CFD software, the external airflow around a ballute model without air inlets was calculated using various approaches. The approaches were dependent on previous aerospace researchers' findings when simulating fluid dynamics using CFD. The research conducted is a computer simulated replication of experimental results published by the aerospace industry in regards to the relatively old technology of ballutes used as inflatable atmospheric deceleration devices.

## Introduction

A ballute is a combination of the two words balloon and parachute (ball – ute). The ballute takes the large body size of the balloon and adds the drag inducing qualities of a parachute. When the two are combined in the proper design, it is possible to decelerate payloads drastically (Hall). The ballute has a large ring (called a burble fence) and a set of air inlets to allow for inflation. The ballute being modeled has a diameter of 35 meters (approx. 115 feet). The diameter is comparable to two school buses parked bumper to bumper. See **Figure 1** for an image of the ballute model created in SolidWorks® for this research. The ballute is part of a larger system which is used for atmospheric re-entry. The total system is comprised of multiple deceleration and stabilization devices which are all unmanned. The appeal of using ballutes for atmospheric re-entry in aerospace practice consists of the weight of the system, the unmanned capability, and relative inexpensiveness.



**Figure 1** – The three dimensional SolidWorks® model that is used to simulate fluid flows in ANSYS® Fluent®. The air inlets and burble fence are variable facets of the design and will be experimentally determined to provide an optimal drag force

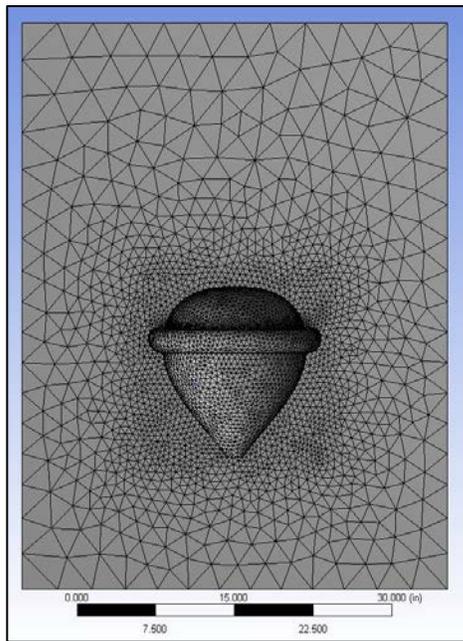
## Geometry and Modelling

Testing all of the different devices in a mock atmosphere re-entry happens rarely and is very expensive. Computational Fluid Dynamics (CFD) is a cost effective

tool enabling the study of various parameters and factors that impact the performance of the ballute. The ballute model was created using SolidWorks and imported into ANSYS® Design Modeler, and later into ANSYS® Meshing. An image of the three dimensional model is seen in **Figure 1**.

In this work, multiple scaled geometries were simulated. Meshes were created for each geometry. Meshing is done to closely replicate the triangulated geometry of a model. This triangulation was achieved, accounting for very small details in the model's geometry. The more details and triangles, the finer the approximation of the fluid dynamic transport equations. ANSYS® Fluent® calculates these transport equations at each node of the mesh triangulation. Mesh triangulation can be created using mathematical laws to approximate any three dimensional volume. The volume model used in this research can be seen in **Figure 2** as a triangulated cylinder with an exposed section view in the yz-plane. Mesh specifications were monitored to assure geometric precision.

Meshing required many different interpretations of the triangulated space, which means that the same geometry was triangulated multiple times until the optimal triangulation was found. This process involved measuring the density of the mesh regions around the boundary layer. The boundary layer is the flow region adjacent to the wall in which the velocity gradients are significant (Cengel). Mesh was created to closely replicate the boundary layer.



**Figure 2** – The mesh triangulation created in ANSYS® Meshing. The ballute is incorporated into the mesh as a shell body. The mesh geometry is a model of the air encapsulating the ballute. The outside walls of this cylindrical column have been corrected in later generations of the mesh model to have a negligible effect on the air flow.

## Methods

This work was fully carried out using computer simulations. The physical properties of air on Earth at different altitudes were reviewed to determine the proper input conditions to simulate flight conditions of atmospheric re-entry. A very important non-dimensional parameter when interpreting air flow is the Reynolds number, which is essentially a measure of the strength of the flow of a fluid – it is the ratio of inertial forces to viscous forces (Cengel). Multiple non-dimensional parameters were considered including Reynolds and Strouhal number comparisons to different published research findings. The table of values considered for a select series of simulations are summarized in **Table 1**. Calculations were solved to find un-scalable terms for result comparison.

**Table 1** – Summary of properties of interest used in simulations. Note the separated section is the environmental conditions of air in the stratosphere on the planet Earth.

| Table 1 - Ballute Fluent Flows (SI) |          |              |              |                   |          |          |
|-------------------------------------|----------|--------------|--------------|-------------------|----------|----------|
| Scale                               | 1:1      | 1:10         | 1:10         | 1:10              | 1:10     | 1:1      |
| Diameter, m <sup>2</sup>            | 1.75     | 0.175        | 0.175        | 0.175             | 0.175    | 1.75     |
| Mach Number                         | 2.42     | 0.24         | 2.42         | 48.69             | 1        | 1        |
| Velocity, m/s                       | 821.37   | 82.14        | 821.37       | 1.66E+04          | 340      | 340      |
| Reynolds Number                     | 2.62E+05 | 2.62E+03     | 2.62E+04     | 2.62E+05          | 16       | 4.70E+03 |
| Temperature, K                      | 267.5    |              | Pressure, Pa |                   | 118      |          |
| Density, kg/m <sup>3</sup>          | 1.51E-03 | Altitude, km | 47.1         | Viscosity, kg/m*s | 1.67E-05 |          |

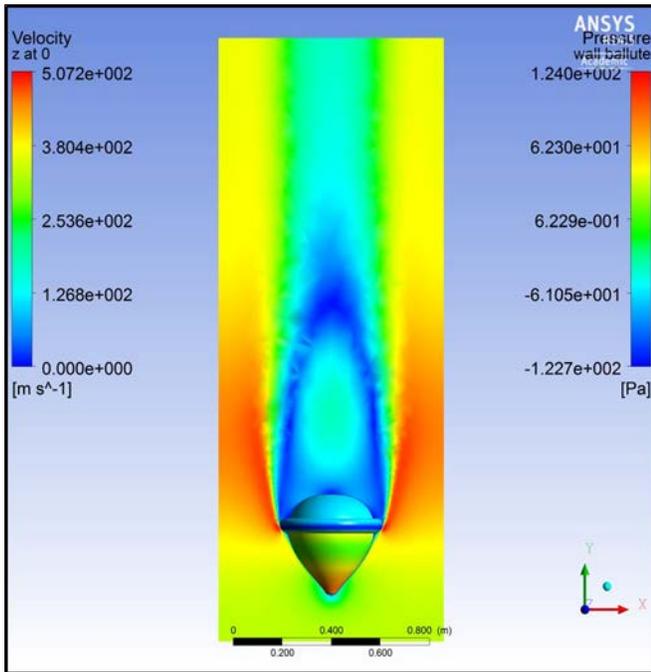
As a starting point, the ballute was modeled in its most simple form, neglecting the air scoops. The ballute was also scaled down by an order of ten and one hundred from an estimated actual design. The flow speed was reduced by

orders of Mach number. This was done to reduce the computational memory size, the computing power, and time needed to carry out each simulation. The actual value of the flow velocity is in the range of Mach 2 – 3, which converts to 1,500 - 2,280 miles per hour. The air flow velocity that was used for the ANSYS® model ranged from Mach 0.2 to 0.3. This velocity is very incomparable to the actual circumstances in which the ballute would be needed. This low Mach number was used to explore the feasibility of scaling aerodynamic effects.

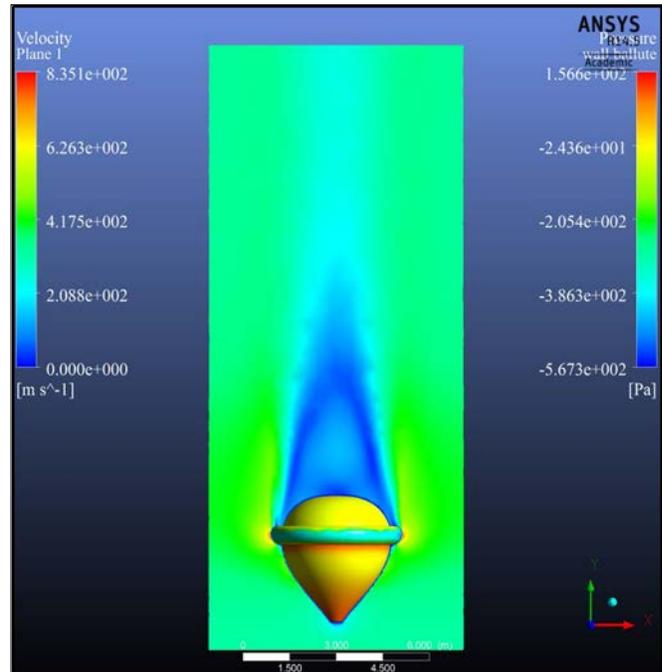
The method that ANSYS® uses to calculate the different fluid phenomena, for time-independent (steady state) situations, is to quantify the momentum vector field dispersion around the geometry, specific to the mesh. The purpose of quantifying a momentum vector field is to effectively approximate the intertwining velocity gradients of the motion of air particles, or packets of air particles (SAS IP). The practice of quantifying momentum vector fields is known as transport equation calculation. This practice, when calculating time-dependent (transient) situations, involves more aspects of fluid dynamics than will be expressed in this paper. Transient situations were simulated in ANSYS® for this research, and the technique for doing so will be improved upon in the future.

## Results

A study of the ballute with no air scoops was conducted on a small scale model and a larger scale model to discern major discrepancies in a comparison between the two models. The study concluded that there is a discernable difference between scaling a model. The results are not scalable between different sized models. This one hundredth scale model was calculated to compare with a larger scale model. The larger, 1:1 scale model was calculated two times as evidenced by **Table 1**. The **Table 1** Fluent® flows listed for the 1:10 scale were simulated because the size of the mesh necessary to compute this size ballute was far smaller in the element and node count, which saves computation time. The larger scale (1:1 in **Table 1**) Fluent® flows took longer to calculate before the transport equation residuals would converge to one millionth of a significant figure.



**Figure 3** – 1:100 scale ballute external air flow patterns at thirty seconds, with a Reynolds number of  $1.13 \times 10^4$  (11,300), characteristic air flow velocity of 82.14 meters per second, and a burble diameter of 0.175 meters squared.



**Figure 4** – 1:10 scale ballute external air flow patterns at thirty seconds, with a Reynolds number of  $1.13 \times 10^5$  (113,000), a characteristic air flow velocity of 821.4 meters per second, and a burble diameter of 1.75 meters squared.

The goal of the aerodynamic scaling effect exploration was to compare the one hundredth scale results with the tenth scale results. The conclusion of this test was that aerodynamics cannot be accurately scaled. Refer to **Figure 3**.

Few Fluent flows did ever reach convergence, which is a very necessary condition to be met because when the residual values are not changing from one iteration of the same calculation to the next iteration, it shows that the air flow has reached a steady state. This means that, since the transient period has passed when air flow patterns are unpredictable, measurements can be taken with the valid assumption that the present conditions at steady state will continue until the air flow changes upstream velocity or angle of attack. Once the residuals converge, comparative data can be shown between two solutions that have converged calculations.

Due to the complexity of the transport equation calculations necessary to compute an internal air flow of this supersonic inflatable aerodynamic deceleration (SIAD) device, the computational model with air inlets, as seen in **Figure 1**, was not used for any Fluent® simulations. The time necessary to compute this complex model must only be used when the result is sure to be accurate and comparable to experimental results.

The larger scale ballute model was calculated numerous times in order to create a proper comparison to the scaled down ballute model that had a converged calculation. It was later concluded that, the model is not scalable, which therefore make it impossible to expect the results of a scaled down object's air flow to be nearly approximate to the scaled up object's air flow patterns.

Compare **Figure 3** and **Figure 4** to see these un-scalable aerodynamic effects of air flow around the differently sized ballute bluff bodies.

### Future Research

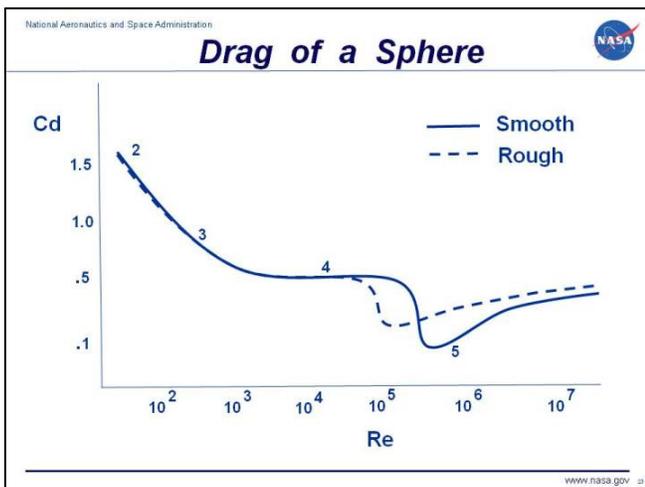
The research conducted has made substantial progress in the direction of creating an ANSYS® simulation model that would solve for the proper conditions of air flow around a ballute. This task has still not been completed, however it has been orientated in a new direction. The new methods that have been investigated for creating proper results of a highly turbulent air flow involve selecting the correct turbulence model set for transport equation approximations. There is a library of conditional transport equation models that can be selected to simulate air flow within Fluent®. The challenge-is to find the proper transport equation model for the highly turbulent conditions and factoring in the large body size of the ballute.

There have been various attempts made at simulating the air flow using the realizable k-epsilon and k-omega turbulence models, as these models are a general form for most turbulent fluid conditions. They appropriately approximate the air flow simulation, without complexity in understanding exactly what must be input for the equation to work correctly. Each turbulence model requires an understanding of the transport equations used in order to properly manipulate the coefficient terms to tailor the transport equations to the situation that must be simulated. More complex models have been used, including the Reynolds Stress Model, which will require more modifications until the proper simulation has been created. This model seems to contain the proper transport equation

set that will properly simulate what must be artificially created in order to replicate an experimental result for an aircraft of this size. Simpler models have also been used, including simulating the ballute with laminar air flow; this did not prove to be a valid approximation and was used more for the purpose of understanding how the laminar transport equation model works.

In an effort to properly set up the Fluent® model to give accurate drag and lift force coefficients, the ballute has been substituted for a sphere of the same characteristic diameter as the ballute. This means that the ballute burble fence diameter has the same diameter as a smooth sphere. By substituting the ballute model for a sphere, it makes the mesh geometry much simpler for the computer to generate and for the transport equations to simulate fluid flow around the substituted sphere body. There are various reasons for substituting the ballute body for a sphere, another of which being that there is a large amount of research conducted on smooth spheres, as it is a very elementary bluff body to comprehend when studying fluid dynamics.

Data for flow over a sphere has been collected and published by NASA and is depicted in **Figure 5**. The data available plots coefficient of drag versus Reynolds number based on the sphere's diameter and flow velocity. The Reynolds number is a non-dimensional quantity that allows for ease of comparison between the two experiments. If our model is correctly formulated, the results that have been published by NASA should be reproducible by the ANSYS® Fluent® simulation software. The only challenge in that is how to properly configure the simulation so that it accurately depicts the fluid dynamics of the environment in the real world.



**Figure 5** – This figure shows the generalized drag coefficient curve for both a smooth and rough sphere for increasing Reynolds number. (Benson)

The drag force coefficient,  $C_d$ , is a non-dimensional parameter that can also be used for comparison between experiments. Another important non-dimensional parameter to monitor when simulating flows is the coefficient of lift,  $C_l$ . There are specific values that have been determined for the lift coefficient for a sphere for a given Reynolds number. Due to the symmetry of flow past a

sphere, values for  $C_l$  are all nearly zero regardless of the change in Reynolds number. This result has been consistent throughout the sphere testing and should continue to be in the future of this research.

The current value that is to be expected in comparison to the data available from NASA is a coefficient of drag of approximately 1.1 for a corresponding Reynolds number of 100. The model simulation will then be changed to use a Reynolds number of 1,000 to find that the simulation reports the correct drag force coefficient of approximately 0.5. The next simulation run will incorporate a Reynolds number of 10,000 and so on until NASA's plotted line can be approximated by plotting six data points.

The remaining research includes using the model developed, which yields the correct drag force coefficient for a sphere, for the purpose of finding the simulated drag force coefficient for the shape of the ballute. The coefficient of drag for the ballute will also be calculated for an array of Reynolds numbers, to create a plot of the relationship between the coefficient of drag and the Reynolds number of a ballute. The relationship should, hypothetically be somewhat comparable to that of a cone, but this is a hypothesis at this point.

In addition to developing the correct model for the ballute air flow simulations, the research will focus on changing the position of the air inlets or burble to find the optimal drag force production for means of the highest possible deceleration forces. The ballute's design could also be varied to allow the air inlets or burble fence to create a specific amount of drag force, with the guide of a plot of the relationship of the drag force coefficient and the position of the air inlets, or the position of the burble fence. Another avenue that this research could pursue in the future is to change the size of the ballute and plot how the change in diameter of the ballute creates a change in the drag force coefficient. Many different routes could be taken with this research, most of which could be very useful in the design of alternative deceleration devices for future space missions.

## Conclusions

This research was conducted with the intention of finding out how varying the air inlets of the ballute would change the drag force coefficient value. After spending a time researching, it was found that this was a bigger problem to be solved than it seemed at first understanding. This goal can be achieved in the future, though it may take multiple efforts to finally solve this problem. The ballute could successfully be replicated in SolidWorks® as a three dimensional model, and this model's air inlets can be changed very easily along with the diameter of the burble fence. The ballute could successfully be meshed with adequate quality, the research will continue in the future in the different directions described in the previous section.

## References

- Benson, T. (Ed.). (2014, June 12). Drag of a Sphere. Retrieved October 23, 2014, from <http://www.grc.nasa.gov/WWW/k-12/airplane/dragosphere.html>
- Cengel, Y., & Cimbala, J. (2012). Properties of Fluids, Internal Flow. In *Fundamentals of Thermal-Fluid Sciences* (4.th ed., p. 424,540). New York: McGraw-Hill.
- Greg, Alexander. "Aerospaceweb.org | Atmospheric Properties Calculator."Aerospaceweb.org | Atmospheric Properties Calculator. 1 Jan. 2012. Web. 7 Aug. 2014.
- Hall, Jeffery L. "A Review of Ballute Technology for Planetary Aerocapture." *IAA Conference on Low Cost Planetary Missions* 1.1 (2000): 1-10. *Jpl.nasa*. Web. 14 Aug. 2014.
- Hanafizadeh, Pedram, Sina Karbalaee M., Behdad Sharbaf E., and S. Ghanbarzadeh. "Drag Coefficient and Strouhal Number Analysis of Cylindrical Tube in Two Phase Flow. " *Energy Equipment and Systems* 1.1 (2013): 35-38. *EnergyEquipSys*. Web. 14 Aug. 2014.
- Imaoka, S. (2008). Using New Meshing Features in ANSYS Workbench Simulation. *ANSYS Advantage, II* (2), 46-48. Retrieved July 9, 2014
- SAS IP. (2013, March 15). Transport Equations for the Standard k-  $\epsilon$  Model. Retrieved October 24, 2014, from // Theory Guide :: 0 // 4. Turbulence // 4.3. Standard, RNG, and Realizable k-  $\epsilon$  Models // 4.3.1. Standard k-  $\epsilon$  Model

## Acknowledgements

Many thanks are in order to my research mentor and partner in this project, Dr. Maria-Isabel Carnasciali. None of this could have been possible without her help and knowledge of fluid sciences. It was her passion for this science that was inspiring to myself to continue working towards solutions for arising problems in this research. She has helped me realize that studying as a graduate in mechanical engineering is something to aspire for and one day achieve.

I would also like to thank the Connecticut Space Grant Consortium for providing the funding necessary to conduct this research. None of this could have been done without the incentives and great opportunities provided by the state of Connecticut's Space Grant Consortium.

I would like to finally thank the SURF program at the University of New Haven for the lessons learned throughout the research about the ethics and correct practice of conducting research. I would also like to thank the SURF program for designating housing for me during the summer.

## Biography

I, the co-author of this document, Anthony R. Mastromarino III, have written this after spending the better half of the second semester and summer of my freshman year studying computational fluid dynamics with my mentor, Dr. Maria-Isabel Carnasciali. I am now a sophomore at the University of New Haven with a major in Mechanical Engineering and a minor in Physics. I plan to continue my education in graduate school where I will be able to research topics at the forefront of science and engineering. I plan to graduate from the University of New Haven in the spring of 2017.

