Computational Fluid Dynamics Validation of Supersonic Parachute Analysis

UROP Research Report

By Tanner Bowell

Introduction

For many years now humans have been exploring outer space for a number of reasons. Whether the goal is to send a satellite into orbit or to explore other planets, we have always been interested in discovering more about our universe. One of the most studied planets over the years has been Mars. There is interest in discovering if there is, or ever has been, any life there as well as determining if it is an adequate planet for inhabitance. Regardless of the desire, the best way to study the planet is to of course go there. Numerous Mars Rovers have been sent to land on Mars in the hopes of learning more about the terrain, atmosphere and biology. One issue that arises is the limitation as to where the rover can land due to the extremely high entry velocities. So far, the Rovers have only landed in areas equivalent to 2 kilometers below our sea level. NASA has interest in the ability to land in areas of higher elevation to explore unseen land. In addition to landing on higher ground, the Mars Science Laboratory (MSL) of NASA recently launched a Rover that is much heavier than in the past.

A possible solution to land a heavier Rover on higher ground is to deploy the parachute much earlier, at velocities up to Mach 2.5 (two and a half times the speed of sound). As a result, extreme pressure fluctuations occur due to a highly unsteady wake that is shed off of the Rover capsule and interacts with the shockwaves that form at the parachute canopy[3] (see figure 1). The unsteady interaction forces fluid inside the canopy which then escapes, causing the canopy to partially collapse on itself. This cycle repeats itself over and over during descent[4] (see figure 2).

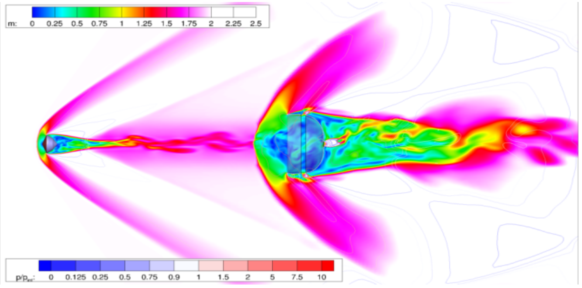


Figure 1: Unsteady wake from capsule interacts with parachute canopy shockwave

These findings have come from previous research and experimental results. Professor Graham Candler from the University of Minnesota and his research team developed a numerical method to model a supersonic parachute entry scenario using US3D Computational Fluid Dynamics (CFD) software. After the previous study they refined the method with the intent to reexamine the cases. The goal of this research project is to run these cases and analyze the results.

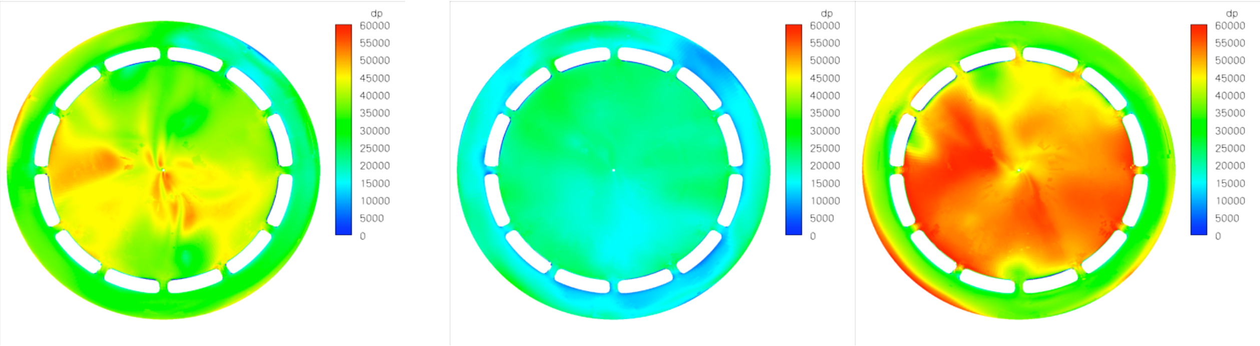


Figure 2: Pressure cycle on the parachute canopy. Orange and red represent very high pressure while blue and cyan represent very low pressure.

Numerical Method

The computations for this project were done using US3D software that was developed at the University of Minnesota. Because the flow field is highly turbulent and unsteady, the numerical method uses Detached Eddy Simulation (DES). Ordinary CFD software uses the Reynolds-averaged Navier-Stokes (RANS) method to model the flow. The drawback of the RANS method is that it is not able to correctly analyze large-scale unsteadiness. The large-eddy simulation (LES) is another method that, unlike DES, is capable of resolving the highly turbulent flow in the wake region. The issue with LES is that it is extremely expensive to model the flow in turbulent boundary layers. The DES method is ideal in this research because it converts between the two methods to maximize efficiency and reduce cost. The RANS model is used near surfaces while the LES method is used farther away from the surface[3].

Experiments

The article on supersonic testing of the disk gap band parachute by Sengupta outlines six different cases that were experimentally tested. These experiments were done as part of separate research and are used to compare with the computational results. The experiments included flow velocities of Mach 2, 2.2 and 2.5. For each of these velocities, cases were run in which the parachute was constrained and unconstrained. The focus of this report is on the constrained cases for each of the Mach numbers. The main parameter that was examined was the drag force. This force acts in the direction of the oncoming flow and is an important indicator of the pressure flucuations. Rather than examining the drag force directly, we use the drag coefficient:

$$C_D = \frac{F_{drag}}{\frac{1}{2}\rho V_\infty^2 A} = \frac{F_{drag}}{\frac{\pi}{8}\rho V_\infty^2 d^2}, \text{ where d = 0.8 m}$$

Table 1 outlines the different parameter conditions that are associated with each case. Notice that the velocities are not extremely high, but since the temperatures are low they can produce high Mach numbers (the speed of sound is proportional to $T^{1/2}$). The results of the experiment showed a pattern of decreasing drag coefficients as the Mach number increased. This is an expected pattern and the hope was that the simulations resulted in the same pattern and values. Figure 3 shows the experimental results compared to the simulation results.

Mach #		Velocity (m/s)	Density (kg/m^3)	Temperature (K)
2	Constrained	520.15	0.12435	168.4
2	Unconstrained	520.16	0.12372	167.46
2.2	Constrained	548.52	0.12192	157.22
2.2	Unconstrained	547.16	0.12003	155.26
2.5	Constrained	583.24	0.11648	139.15
2.5	Unconstrained	581.10	0.11461	136.62

Table 1: Experimental and Simulation Run Conditions

Results

The computations were performed on the Titan Supercomputer cluster at the University of Minnesota. The cluster was accessed using the terminal program on a Unix based computer. Through the terminal I was able to change the case parameters, submit jobs for calculating, and analyze the results. Because the purpose of these simulations was to validate the experimental results, we used the same conditions as in table 1.

In addition to the velocity, density and temperature, there are a number of other parameters that can be changed in the input file for the CFD solver. Because I have not had any previous experience with CFD, there was a lot of trial and error with many of these parameters. One of the challenges was to ensure that the solver did not take too big of a timestep. If this happens the data can be smeared and the results will either be incorrect or represent something non-physical. The low dissipation fluxes also needed to be considered. This parameter was tricky because there are several options, but regardless of the choice it was important to inform the CFD solver as to how to handle the dissipation. The parameter that controls this was set to a value that works well with strong shocks (these can arise in blunt body flows). This is a good choice because the capsule has a very blunt shape that leads to strong shock waves because it causes the flow to deflect by large angles. The formation of these shocks can be seen at the leading edge of the capsule in Figure 1.

Once the parameters were set to the desired values, useful data could be obtained. Each constrained case was run until the mean values of the drag force converged. In order to determine when this occurs, I wrote a simple program using C++ that took the drag force values and found the mean over the dataset. The program allowed input for a start time (when it was desired to start looking at the data) and a stop time. With this capability it was simple to see what the mean value over the first half and second half of the data is. If these values are reasonably similar it can be determined that enough data has been obtained for that case. Figure 3 shows the results from the simulation and experiments.

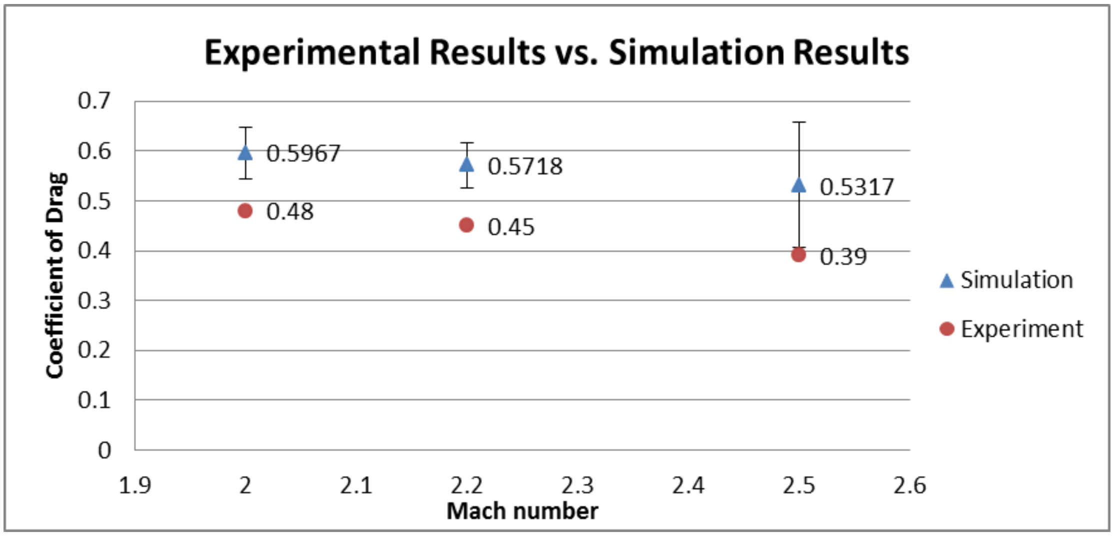


Figure 3: Experimental vs. Simulation results for the drag coefficient

As you can see from the plot, the drag coefficient results from the simulation follow the same trend as the experimental values. The issue is that the values seems to be shifted up slightly higher. When considering the error bars (with a value equal to the standard deviation), the drag coefficients line up with the data to a fairly reasonable degree. Trying to determine the reasons for the disparity in the results can prove to be challenging. The most likely reason is the CFD solver parameters. Other choices for low dissipation fluxes or time integration could have been used to yield more accurate results. These adjustments will be considered as the research is continued.

In addition to the drag coefficients, other data has been collected during the simulations as well. The x,y, and z forces and moments have been calculated at each time step. These forces and moments have been plotted as a function of time and show a periodistic nature. Figures 4,5 and 6 shows the x-forces for the Mach 2, 2.2 and 2.5 plotted versus time. These patterns show that there is in fact some sort of periodicity that can be analyzed.

The most efficient way to determine the frequency that these forces oscillate at would be to perform a Fast Fourier Transform (FFT). This is not possible for this data because the time step between the data points is not equal. Therefore, we can perform a Discrete Fourier Transform. This is not as efficient but still serves an equivalent purpose. At this point in the research data is still being collected, but when it is completed it will be possible to determine the frequencies present in the data.

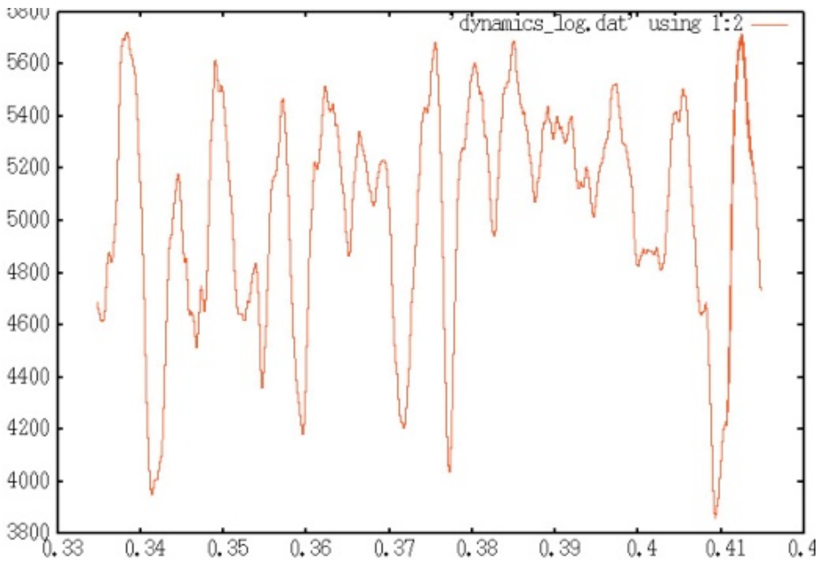
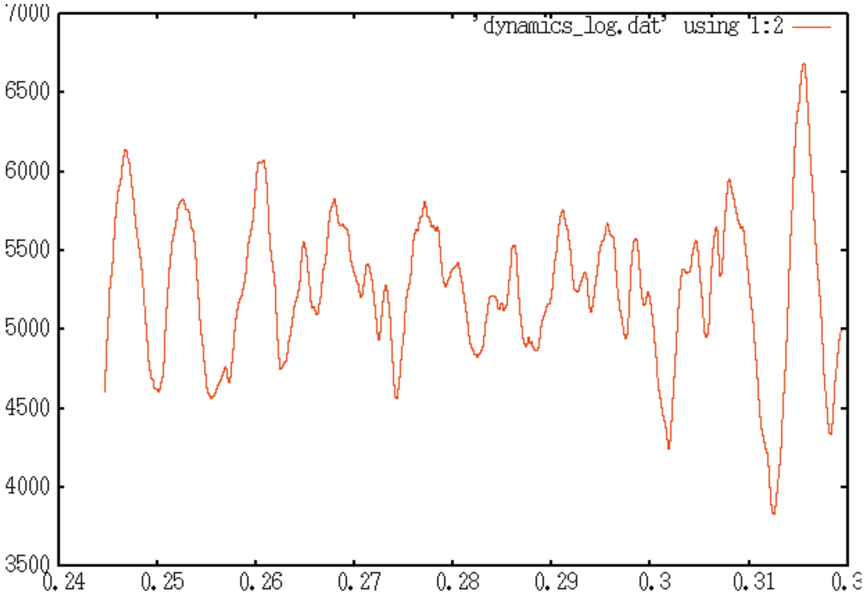


Figure 4: Mach 2 drag force as a function of time

Figure 5: Mach 2.2 drag force as a function of time



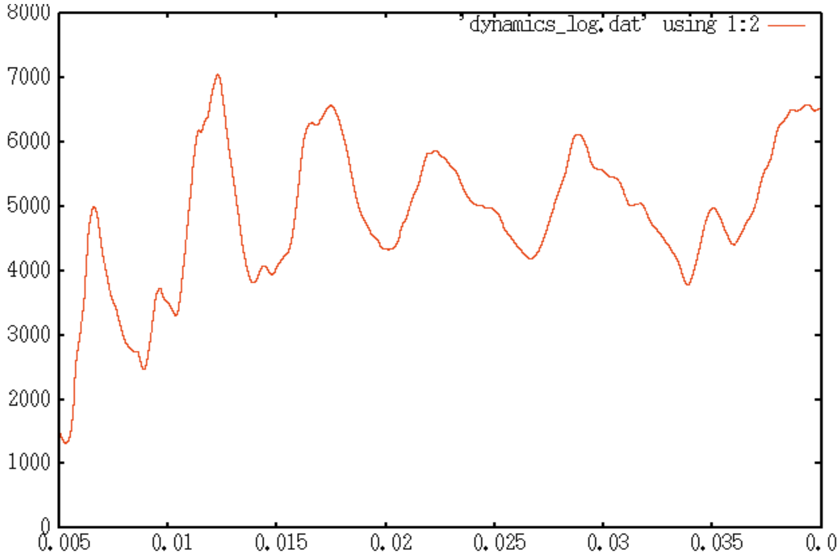


Figure 6: Mach 2.5 drag force as a function of time

Conclusion

The research described in this report is the first step of many on this topic. So far we have only analyzed the constrained cases for the three different Mach numbers. The results did not perfectly match up to the experimental values, but when considering the standard deviation of the data they match reasonably well. In a real-world application we know that the parachute will not be constrained. This research will be continued to consider the unconstrained cases in which the parachute is allowed to rotate freely. The goal, as it was with the constrained cases, is that the CFD simulations show that the drag coefficients line up with the experimental results. In addition to aligning the mean drag coefficients, it will also be important to determine the periodicity of the forces and moments. By using frequency analysis, such as a Discrete Fourier Transform, we can determine the dominant frequencies of the forces and moments over the entry and descent of the Rover. Knowing these frequencies we will have the ability to predict the dynamics. This information could lead to a better design of the parachute or other components to avoid collapse and ensure a safe landing.

References

[1]Michael Barnhardt, Travis Drayna, Ioannis Nompelis, Graham V. Candler, and William Garrard. “Detached Eddy Simulations of the MSL Parachute at Supersonic Conditions.” *American Institute of Aeronautics and Astronautics* (2007). Web. 9-24-2011

[2]Sengupta, M. Wernet, J. Roeder, R. Kelsch, A. Witkowski, and T. Jones, "Supersonic Testing of 0.8 m Disk Gap Band Parachutes in the Wake of a 70 deg Sphere Cone Entry Vehicle," Presented at the 20th AIAA ADS Conference, May 2009, Seattle, WA.

[3]Vladimry Gidzak, Michael Barnhardt, Travis Drayna, Ioannis Nompelis, Graham V. Candler, and William Garrard. “Comparison of Fluid-Structure Interaction Simulations of the MSL Parachute with Wind Tunnel Tests.” *American Institute of Aeronautics and Astronautics* (2009). Web. 9-23-2011.

[4]Vladimyr Gidzak, Michael Bernhardt, Travis Drayna, Ioannis Nompelis, Graham V. Candler, and William Garrard. “Simulation of Fluid-Structure Interaction of the Mars Science Laboratory Parachute.” *American Institute of Aeronautics and Astronautics* (2008). Web. 9-24-2011.