Project Midterm - Parachutes in Flight

Lovleen Kaur and Abinav Ramachandran Kamalakannan

I. INTRODUCTION

Parachutes are deployed and used in a varying applications ranging from aerospace, human descent and cargo delivery. Understanding the complex fluid-structure interactions during the deployment of a parachute is essential in developing robust and safe parachutes. By creating a numerical solver to study this behavior under various conditions, we hope to gain valuable insight on which factors affect the functioning of a parachute the most.

The numerical simulation framework used in this project is based on discrete elastic rods (DER) with hydrodynamic forces. Using numerical methods like Newton-Rhapson, we aim to simulate the forces and motion of the canopy, cords and the load. For the sake of simplicity, we avoid simulating the fluid around the parachute and have a simplified model to approximate the aerodynamic drag on the canopy.

The primary objectives of this study is to examine the steady state canopy profile under steady descent, and to determine the terminal velocities achieved across a range of operating parameters. These parameters include material properties, fluid properties and geometric factors. By studying the performance of the parachute for the various parameters, we hope to provide insights on parachute performances under different design choices and environmental conditions.

Extensive work have been previously done in simulating the behavior of deploying parachutes in supersonic regimes [2] and subsonic regimes [5]. Our paper aims to gain a broad understanding of which design parameters has the most significant impact on the performance of a parachute in various flow conditions and uses cases.

II. METHODOLOGY

The simulation work for this project is done with Hydrodynamics applied with Discrete Elastic Rods algorithm. We estimate the Reynolds number using a characteristic velocity v_{charac}, a characteristic length L_{charac}, fluid density ρ, and viscosity μ:

$$Re = \frac{\rho\, v_{\text{charac}}\, L_{\text{charac}}}{\mu}.$$

A. High Reynolds Number Hydrodynamics (Quadratic Drag)

For high Reynolds number flows ($Re \gg 1$), inertial forces in the fluid dominate and hydrodynamic drag scales quadratically with velocity. Where ρ denotes the fluid density and C_D is the drag coefficient. Each node represents a segment of undeformed Voronoi length l_k and circular radius r_0. The projected area associated with node k is

$$A_k = 2r_0\, l_k.$$

If the background fluid has uniform velocity $\mathbf{U}_\infty$, drag depends on the relative velocity

$$\mathbf{u}_k^{\text{rel}} = \mathbf{u}_k - \mathbf{U}_\infty,$$

where $\mathbf{u}_k$ is the backward–Euler velocity. If no flow is present, $\mathbf{U}_\infty = 0$.

The high–Re drag force at node k is

$$\mathbf{f}_k^{\text{highRe}} = -\tfrac{1}{2}\rho C_D A_k \|\mathbf{u}_k^{\text{rel}}\|\, \mathbf{u}_k^{\text{rel}}.$$

where k is defined as

$$k_k = \tfrac{1}{2}\rho C_D A_k,$$

so that

$$\mathbf{f}_k^{\text{highRe}} = -k_k\, \|\mathbf{u}_k^{\text{rel}}\|\, \mathbf{u}_k^{\text{rel}}.$$

Jacobian of the High-Reynolds Drag

Let $g(\mathbf{u}) = \|\mathbf{u}\|\mathbf{u}$. So its differential is

$$dg = \left(\|\mathbf{u}\|\mathbf{I} + \frac{\mathbf{u}\mathbf{u}^T}{\|\mathbf{u}\|}\right) d\mathbf{u}.$$

Thus,

$$\frac{\partial g}{\partial \mathbf{u}} = \|\mathbf{u}\|\mathbf{I} + \frac{\mathbf{u}\mathbf{u}^T}{\|\mathbf{u}\|}.$$

Applied to $\mathbf{u}_k^{\text{rel}}$,

$$\frac{\partial \mathbf{f}_k^{\text{highRe}}}{\partial \mathbf{u}_k^{\text{rel}}} = -k_k\left(\|\mathbf{u}_k^{\text{rel}}\|\mathbf{I} + \frac{\mathbf{u}_k^{\text{rel}}\left(\mathbf{u}_k^{\text{rel}}\right)^T}{\|\mathbf{u}_k^{\text{rel}}\|}\right).$$

Since

$$\frac{\partial \mathbf{u}_k^{\text{rel}}}{\partial \mathbf{x}_k(t_{j+1})} = \frac{1}{\Delta t}\mathbf{I},$$

the Jacobian with respect to DOFs is

$$\frac{\partial \mathbf{f}_k^{\text{highRe}}}{\partial \mathbf{x}_k(t_{j+1})} = -\frac{k_k}{\Delta t}\left(\|\mathbf{u}_k^{\text{rel}}\|\mathbf{I} + \frac{\mathbf{u}_k^{\text{rel}}\left(\mathbf{u}_k^{\text{rel}}\right)^T}{\|\mathbf{u}_k^{\text{rel}}\|} \right).$$

It is zero for all $j \neq k$. The Jacobian is therefore block-diagonal: a 2×2 block for planar rods or 3×3 for 3D rods.

B. High-Reynolds Forces Solver Algorithm

The computation starts with,

$$\mathbf{F}_h = 0_{(n_{\text{dof}})}, \qquad \mathbf{J}_h = 0_{(n_{\text{dof}} \times n_{\text{dof}})},$$

and loops over all nodes k to:

- Compute $\mathbf{u}_k$ and relative velocity $\mathbf{u}_k^{\text{rel}}$.
- Compute projected area $A_k = 2r_0 l_k$.
- Evaluate the high-Re drag force and assemble it into $\mathbf{F}_h$.
- Evaluate the corresponding Jacobian (with regularization if applicable) and assemble it into the (k, k) block of $\mathbf{J}_h$.

C. Discretization of Domain and External Forces:

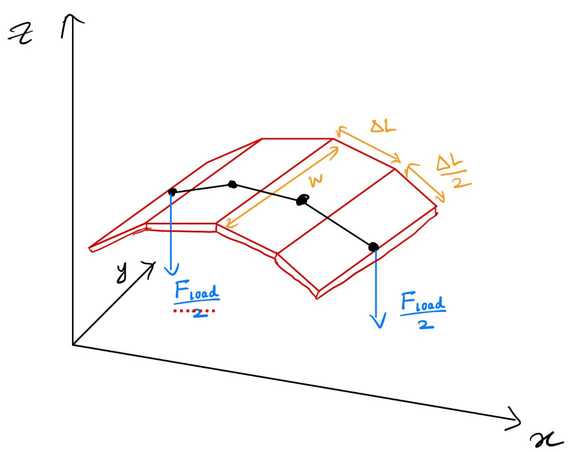

Fig. 1: Discrete Representation of a Parachute and Load

Figure 9 shows the discrete representation of a parachute, each node of the parachute (in black ink) has two main forces acting on it. There is the force of gravity pulling the parachute down and the drag force acting in the opposite direction. The magnitude of the drag force will depend on the shape of the parachute and that is computed by the hydrodynamics force part of the computational scheme. The main assumption is the hydrodynamics logic is the coefficient of drag and that is because the value used is largely an estimation for parachutes after they are deployed fully. This number is highly dependent on the shape of the parachute but for the purposes of this study, the value is held a reasonable constant for the final shape. In reality, the C_D (drag coefficient) varies heavily with the flow conditions, gas composition, and parachute shape. The aim of the project is to study the behaviors of this flexible structure under varied applied loads and material properties. The high fidelity aero-structures coupling analysis or drag profile in transient state of deployment is not the main target of this project. The parachute here is assumed to start flat and expand into its shape. This number is taken from a NASA Glenn Research Center quick information and learning page for basics of parachute dynamics [6].

$$C_D = 1.75$$

The C_D values can be estimated using flight data or from prior higher fidelity simulations. NASA has variety of papers that describe parachute analysis and C_D for different flight envelopes. One such paper provides C_D properties for Mars landing [1]. Another similar resource cover the numerical analyses that details the CFD work necessary to tackle this problem [3]. These sources examine this process much more in depth rather than a simplified constant coefficient like this in case.

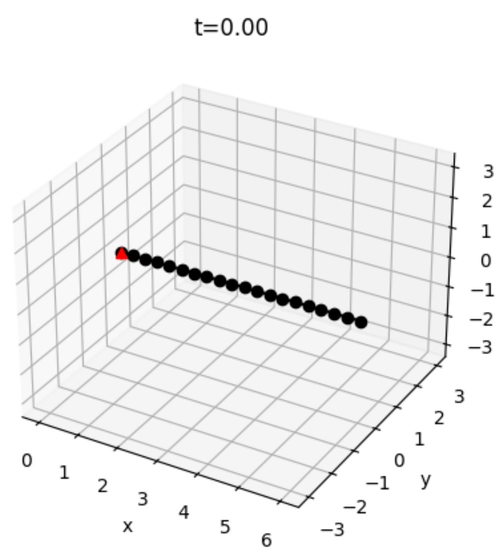

Fig. 2: Initial State of the Discrete Parachute

D. Choice of dt:

Time step size depends on the method, the system being analyzed, computational cost, and the speed needed. For schemes that are stable, we can select larger time steps and scheme that are less stable or unstable, the time step must be small. In this case, the time step for the implicit scheme does not have a stability concern but rather its a matter of damping in the solution. The selected time step of 0.01 resulted in a solution that converged well and reached a steady state that is a expected of a parachute.

III. RESULTS

Figures 3 to 5 results are obtained by using an even load distribution of 100 N on each side of the parachute. This represents the gravitational force by the hanging mass under the parachute.

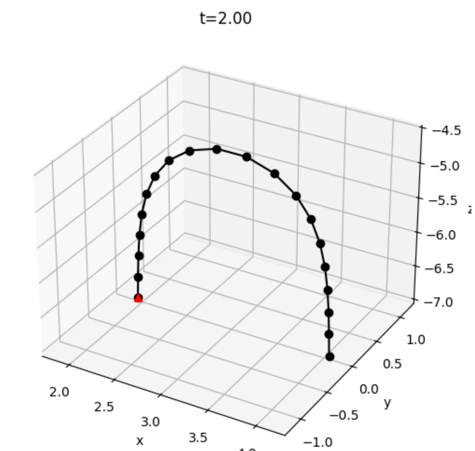

Fig. 3: Even Load: Final State of the Discrete Parachute

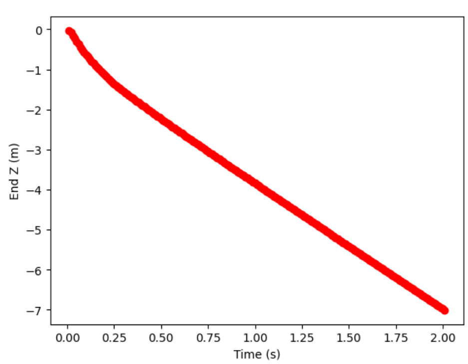

Fig. 4: Even Load: Z Direction Velocity Profile

Figures 6 to 9 results are obtained by using an uneven load distribution of 125 N and 75 N on each side of the parachute. This uneven load is created to to test the implementation of a turn motion for a parachute, a motion that causes it to drift in the x direction.

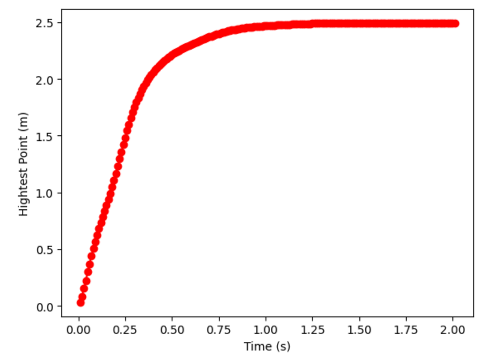

Fig. 5: Even Load: Parachute Relative Height of Mid Point

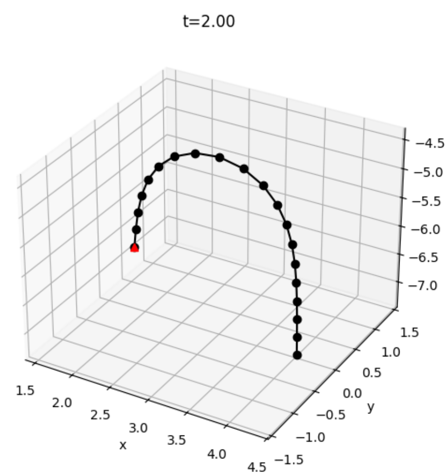

Fig. 6: Uneven Load: Final State of the Discrete Parachute

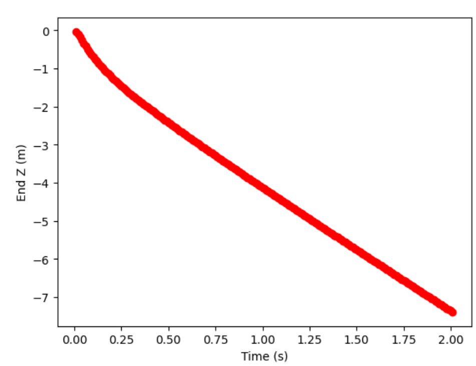

Fig. 7: Uneven Load: Z Direction Velocity Profile

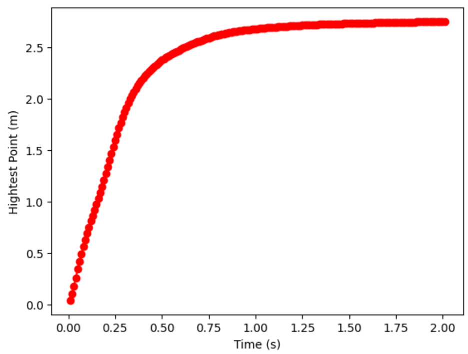

Fig. 8: Uneven Load: Parachute Relative Height of Mid Point

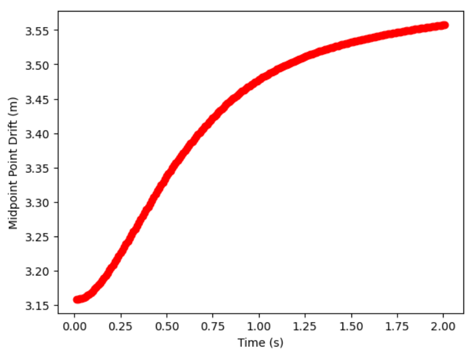

Fig. 9: Uneven Load: X-Direction Drift

IV. DISCUSSION

An ideal parachute will provide enough drag for a low terminal velocity and have a compact geometry after deployment. The numerical solver provided in the previous section enables us to simulate the deployment of the parachute and find its steady state solution. This steady state solution is reached when the Z velocity becomes linear, meaning the parachute has reached terminal velocity of the load and the shape of the canopy profile.

The results shown in figures 3 to 9 are in agreement with the predicted behavior. The final shape for the parachute under even load is symmetric and the arch displays a shape common to parachutes. It also reaches terminal velocity relatively fast. The results for the uneven load show non symmetric end profile and a drift for x position which is to be expected for a real parachute as well.

In reality, we expect the performance of the parachute to heavily rely on the drag force and the final canopy profile. The canopy profile will in-turn strongly rely on the material properties of the canopy itself, the chord and the actual sizes of the canopy to the chord. By using an approximation for the drag force coefficient, we restrict the focus of our initial study to the effects of the material properties and general shape of the parachute.

V. NEXT STEPS

Parachutes have a wide range of uses in various environments. To reflect these varying use cases, the geometry and load weight can be scaled. For each case, different dimensions of canopy and chords, as well as material properties will be tested through simulations. Through repeated tests we hope to gather insights on how the materials and geometry of a parachute will affect its performance for diverse use cases.

Next, we aim to incorporate more load points and strings for more accurate loading and flow physics. This problem can also be solved with a classic CFD approach of Immersed Boundary Method but that is largely outside the scope of the methods learned in this class and more suited for a CFD course[4].

This problem is very dynamic and has fluids and structures coupling that rapidly changes the drag profile of the parachute. Advanced modeling analysis of parachutes like those covered in published works from NASA Ames are highly complex and require computational resources well beyond the scope of a graduate class but they do provide insight into expected behaviors possible within numerical analysis [2].

ACKNOWLEDGMENT

The core of the python code and the numerical logic used in this project is directly taken from the course materials provided by Professor M. Khalid Jawed for MAE 263F.

REFERENCES

[1] J. R. Cruz , "Estimates for the Aerodynamic Coefficients of Ringsail and Disk-Gap-Band Parachutes Operating on Mars ," Oct. 2017.
[2] J. Boustani, G. Anugrah, M. F. Barad, C. C. Kiris, and C. Brehm, "A Numerical Investigation of Parachute Deployment in Supersonic Flow," AIAA Science and Technology Forum and Exposition.
[3] L. Jiang et al., "Numerical Study on Aerodynamic Performance of Mars Parachute Models with Geometric Porosities," Space: science and technology, vol. 2022, Jan. 2022.
[4] Kim, Y., Peskin, C. S. (2006a). 2–D parachute simulation by the immersed boundary method. SIAM Journal on Scientific Computing, 28(6), 2294–2312. https://doi.org/10.1137/s1064827501389060
[5] G. Xing-long, Z. Qing-bin, T. Qian-gang, and Y. Tao, "Fluid-Structure Interaction Simulation of Parachute in Low Speed Airdrop," Proceedings of the World Congress on Engineering , vol. 3, no. WCE 2013, Jul. 2013.
[6] NASA Glenn Research Center. (n.d.). Velocity during recovery. https://www.grc.nasa.gov/www/k-12/VirtualAero/BottleRocket/airplane/rktvrecv.html