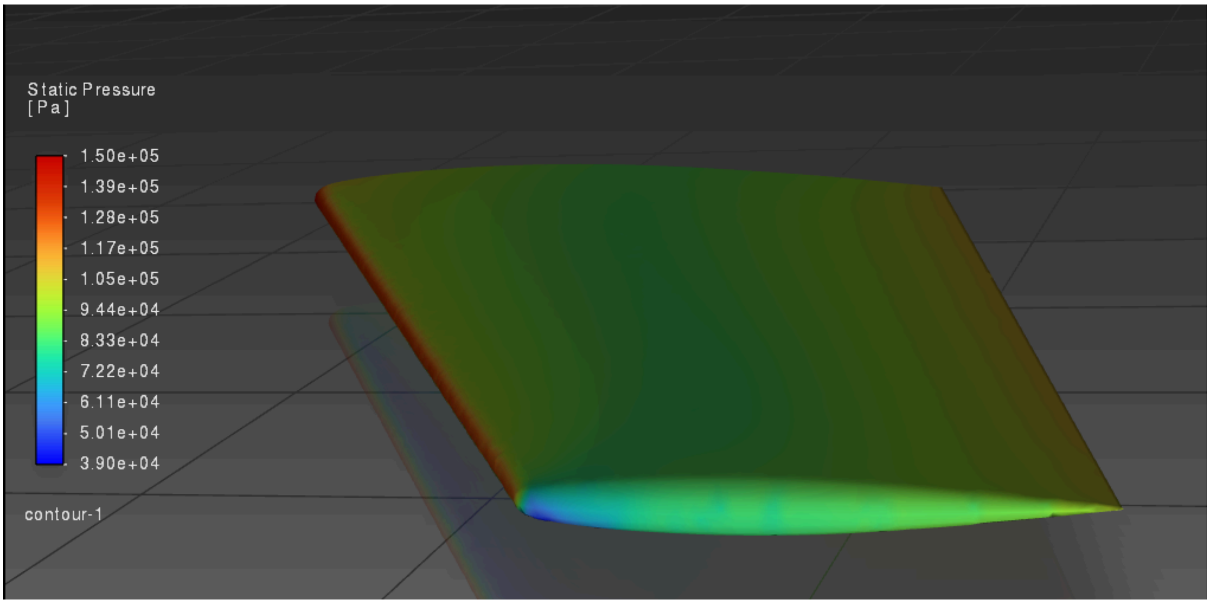
Aerodynamic Analysis of the NASA ONERA M6 Wing



1. Introduction/Problem Statement

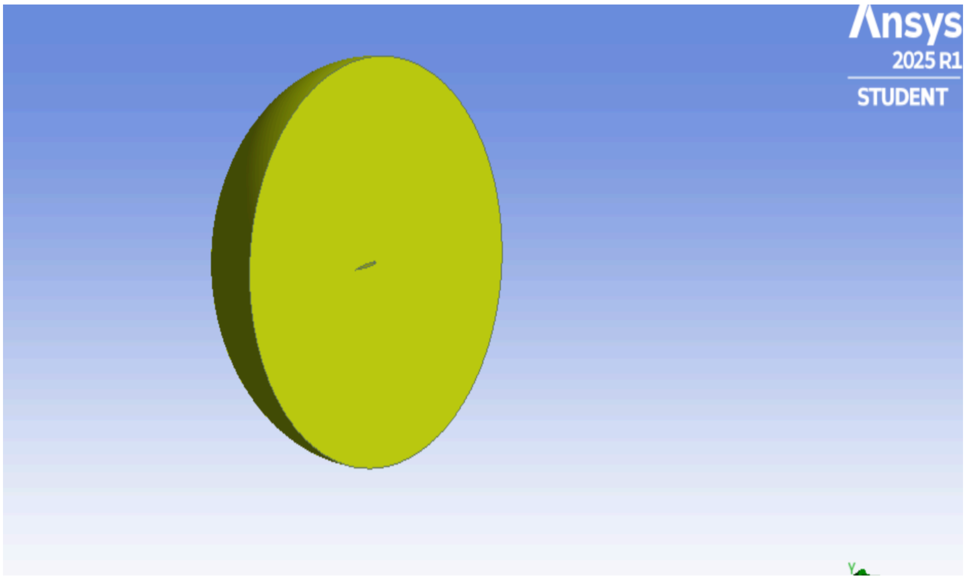
This project conducted a comprehensive aerodynamic analysis of the NASA ONERA M6 wing using Computational Fluid Dynamics (CFD). The primary goal was to determine the relationship between the wing's lift coefficient (CL), drag coefficient (CD), and its angle of attack (AoA). By understanding these relationships, the project aimed to pinpoint the wing's most aerodynamically efficient operating conditions and validate the simulation's findings against established reference data, which is a crucial step in modern aerospace engineering for optimizing aircraft design and performance.

---

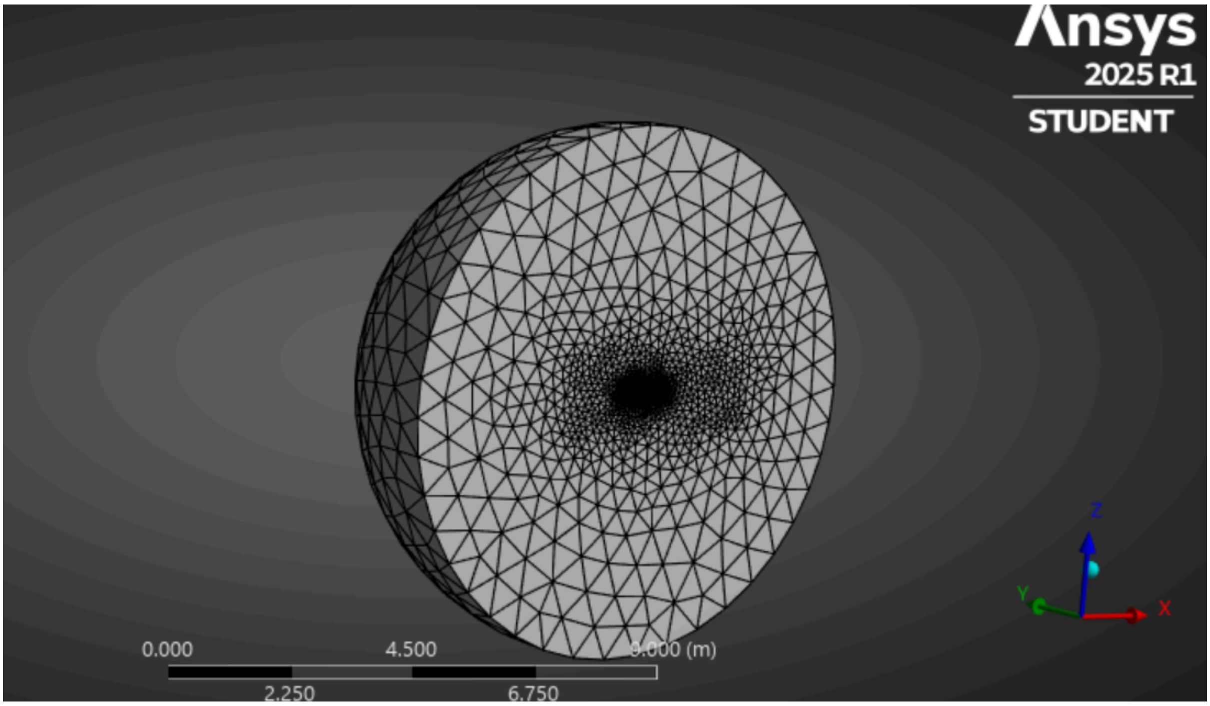
2. Methodology

The analysis was performed using a systematic, multi-phase approach to ensure the accuracy and reliability of the results. The following steps outline the methodology used to solve the problem:

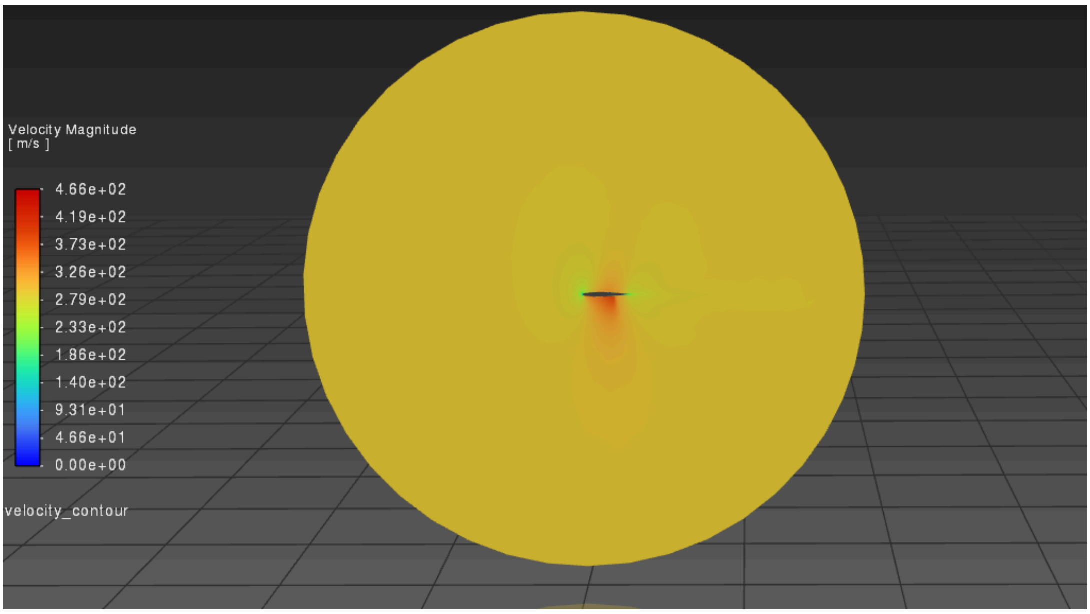
Phase 1: Geometry and Domain Setup The geometry of the NASA ONERA M6 wing was sourced from NASA's Turbulence Modeling Resource. To simulate the airflow, a computational domain—a virtual wind tunnel—was created around the wing geometry in ANSYS DesignModeler. This fluid volume was generated by using a Boolean subtract operation, where the wing's solid geometry was subtracted from the larger wind tunnel domain. To reduce computational cost and time, the model was halved along the wing's chord plane, leveraging the principle of symmetry. This approach significantly cut the number of cells needed for the simulation while still providing accurate results for the full wing.



Phase 2: Meshing A computational mesh was generated over the fluid domain. Meshing is a critical step in CFD as it divides the fluid volume into a finite number of smaller elements, or cells, where the governing equations of fluid flow are solved. The quality and density of the mesh directly influence the accuracy of the results. A finer mesh, particularly near the wing surface and in areas of high flow gradients, ensures that the complex flow phenomena, such as the boundary layer and shock waves, are captured with high precision. This balance between mesh density and computational cost was a key consideration in the project.



Phase 3: Simulation Setup The simulations were performed using ANSYS Fluent. The **Spalart-Allmaras** turbulence model was selected for this project. This particular model was chosen because NASA also used it in its own analyses of the ONERA M6 wing, which made a direct and reliable comparison of results possible. The angle of attack was varied from 0 to 17 degrees. This was achieved by adjusting the inlet boundary conditions' velocity components. For each angle of attack (α), the velocity components were calculated using coordinates to vary, i.e., for the x-axis it is cosine of the angle of attack and the z-axis it is sine of the angle of attack, with the y-axis remaining at 0.



Phase 4: Post-Processing and Validation Following each simulation, the lift and drag forces were extracted to calculate the CL and CD values. These coefficients were then plotted against the angle of attack to visualize the wing's aerodynamic performance. The **lift-to-drag ratio (CL/CD)** was also calculated and plotted to determine the wing's aerodynamic efficiency. Calculating this ratio is essential as it is a direct measure of how well a wing converts lift into drag; a higher ratio indicates greater efficiency. Finally, the simulation results were compared against known data from NASA to validate their accuracy.

---

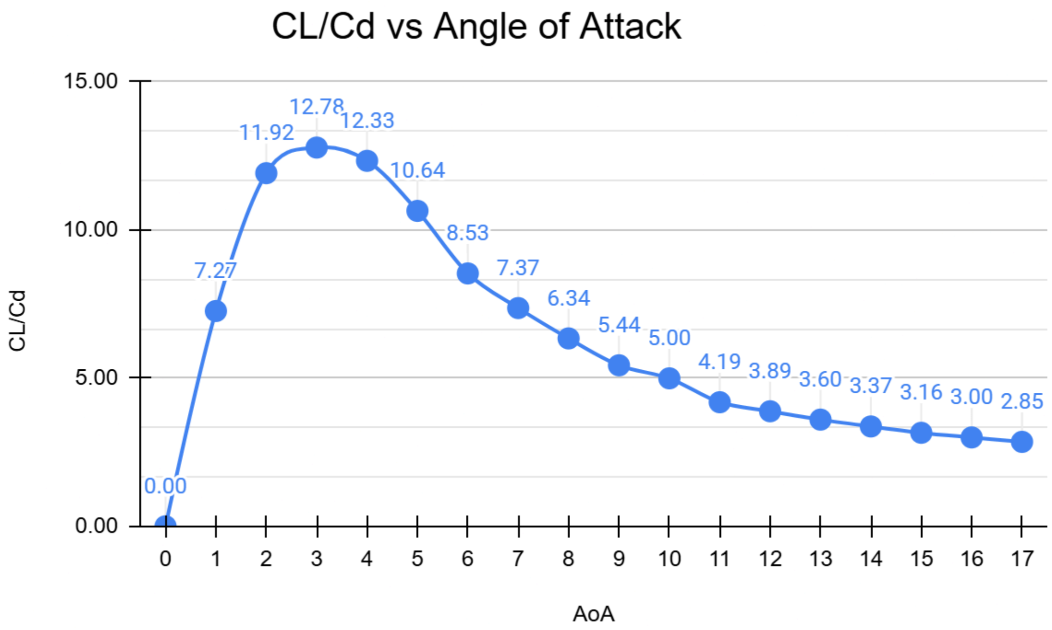
3. Results

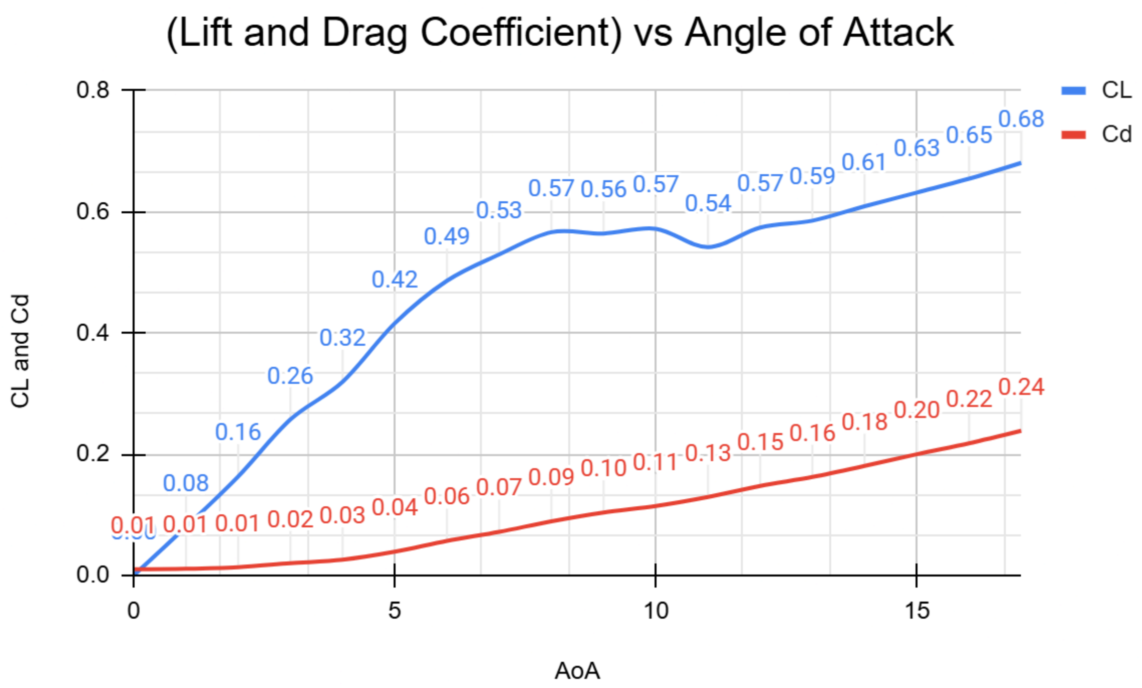
The simulation yielded a comprehensive set of data, which was analyzed to understand the aerodynamic behavior of the NASA ONERA M6 wing. The findings were consistent with established aerodynamic principles and provided valuable insights into the wing's performance.

- **Lift and Drag Coefficients:** The simulation data for the **lift coefficient (CL)** and **drag coefficient (CD)** showed expected trends. The CD was found to gradually increase with the angle of attack, starting from a minimum value and rising steadily. This is a typical behavior as induced drag becomes more significant at higher angles.

The CL initially showed a sharp, linear increase with the angle of attack. However, after approximately 8 degrees, the rate of increase began to slow down, and the curve entered a distinct plateau, where the increase was minimal. This behavior is indicative of flow separation beginning to occur over the wing's upper surface.

- **Aerodynamic Efficiency:** The lift-to-drag ratio (CL/CD) was calculated and plotted against the angle of attack to determine the wing's aerodynamic efficiency. The ratio was found to increase from a low value at 0 degrees, reaching its maximum at an angle of attack of **3 degrees**. After this point, the ratio began to decrease as the rate of increase in drag surpassed the rate of increase in lift. This validated the critical flight condition for maximum efficiency, which is a key parameter for pilots and designers.





---

4. Challenges and Limitations

This project, while successful, faced a few challenges and limitations that could be addressed in future work to improve the accuracy and completeness of the results.

- **Mesh Density Limitation:** The student version of ANSYS Fluent imposed a limitation on the total number of mesh cells. While the meshing was optimized to capture key flow features, a higher cell count could have provided more granular detail and potentially improved the overall accuracy of the simulation results.
- **Angle of Attack Step Size:** The simulations were conducted at 1-degree intervals. To get a more precise understanding of the wing's behavior, particularly around the peak efficiency point and the stall region, a smaller step size (e.g., 0.5 degrees) would be necessary to capture the exact maximum values more accurately.

---

5. Conclusion

This project successfully achieved its goal of analyzing the aerodynamic characteristics of the NASA ONERA M6 wing. The simulations accurately captured the fundamental relationships between the angle of attack and the lift and drag coefficients. The findings, including the gradual increase in drag and the plateauing of lift, are consistent with established aerodynamic principles. The project also successfully identified the optimal angle of attack for maximum aerodynamic efficiency at **3 degrees** and validated these findings against a NASA analysis, demonstrating a proficient application of CFD techniques for aerodynamic studies.

---

6. References

1. https://www.researchgate.net/publication/327792764_CFD_Analysis_on_the_Effect_of_Winglet_Cant_Angle_on_Aerodynamics_of_ONERA_M6_Wing#:~:text=The%20geometry%20of%20the%20winglets,validated%20with%20the%20NASA%20results.
2. https://turbmodels.larc.nasa.gov/onerawingnumerics_val.html
3. https://arc.aiaa.org/doi/10.2514/6.2016-1357