

Internship at CSIR National Aerospace Laboratories, Bangalore in Computational & Theoretical fluid dynamics division.

Report by,
Sahil Chaudhary
Ajeenkya DY Patil University, Pune
Duration: 18/08/2025-18/11/2025

□

Acknowledgment

I would like to express my sincere gratitude to **Mr. Shikhar Jaiswal A., Principal Scientist, CTFD Division at CSIR NAL**, for providing invaluable guidance, feedback and encouragement throughout the internship period. His expertise in CFD methodologies significantly enhanced my understanding of the subject in detail. I am also grateful to the CFD division team for assisting me with technical queries. Also, I would like to thank NAL for giving me the opportunity to learn by providing an internship.

□

Abstract

This report documents a three-month internship conducted in the Computational and Theoretical Fluid Dynamics Division at CSIR National Aerospace Laboratories (NAL), Bangalore, India, from August 18 to November 18, 2025. The internship provided a thorough immersion into the fundamentals and practical applications of Computational Fluid Dynamics (CFD), including in depth study of different conditions, mesh generation, numerical simulation, and post-processing techniques. Key software tools such as Pointwise, GMSH for preprocessing and mesh generation, SU2 for simulation, and Paraview for visualization and analysis were utilized to investigate various classical and advanced aerodynamic problems. During the period, extensive hands-on projects were completed, including mesh

generation and simulation of the NACA0012 airfoil, transonic simulation and experimental validation of the RAE2822 supercritical airfoil, comparison of ONERA M6 wing simulation data at different spanwise locations, and a dynamic separation study of the Eglin store using six degrees-of-freedom (6DOF) analysis. Simulation results were evaluated against benchmark experimental data, and comprehensive post-processing was performed to extract aerodynamic characteristics and interpret the underlying flow physics. The internship fostered both technical and analytical skills, offering valuable insights into advanced CFD workflows and strengthening the foundation for future research or industry roles in aerospace engineering.

□

INDEX

- Title
- Acknowledgement
- Abstract
- INDEX
- Chapter 1
 - 1.1 Introduction
 - 1.2 About CSIR NAL
 - 1.3 About CTFD Division
 - 1.4 Internship Objective
 - 1.5 CFD Softwares used and learned
- Chapter 2
 - 2.1 Initial phase of internship
 - 2.2 Second Phase of Internship
 - Figure 1:RAE2822 su2 cfg input
 - Figure 2:RAE2822 Sim vs Exp Pressure coefficient plot
 - Figure 3:RAE2822 experimental Cp by NPARC
 - 2.3 Third Phase of internship
 - Figure 4:ONERA Pointwise Mesh
 - Figure 5:ONERA Farfield Mesh
 - Figure 6:ONERA input su2 cfg
 - 2.4 Final Phase of internship
 - Figure 7:Eglin Wing-Store mesh at Position-0
 - Figure 8:Eglin input su2 cfg
- Chapter 3
 - 3.1 Problem statement
- Chapter 4
 - 4.1 Results
 - Figure 9:ONERA Cp Simulation plot at $y/b=0.20$
 - Figure 10:ONERA Cp experimental plot at $y/b=0.20$
 - Figure 11:ONERA Cp Simulation plot at $y/b=0.44$
 - Figure 12:ONERA Cp experimental plot at $y/b=0.44$
 - Figure 13:ONERA Cp Simulation plot at $y/b=0.65$
 - Figure 14:ONERA Cp Experimental plot at $y/b=0.65$
 - Figure 15:ONERA Cp Experimental plot at $y/b=0.80$
 - Figure 16:ONERA Cp Simulation plot at $y/b=0.80$
 - Figure 17:ONERA Cp Experimental plot at $y/b=0.90$
 - Figure 18:ONERA Cp Simulation plot at $y/b=0.90$
 - Figure 19:Eglin Store Position at 23
 - Figure 20:Eglin Store Position at 38
- Chapter 5
 - 5.1 Conclusion
- References

□

Chapter 1

1.1 Introduction

A subfield of fluid mechanics known as computational fluid dynamics (CFD) analyzes and solves fluid flow problems using numerical analysis and algorithms, especially in complex situations where analytical solutions are not feasible. CFD is essential to aerospace engineering because it allows for the simulation and comprehension of the behavior of gases and air surrounding aircraft, spacecraft, missiles, and propulsion systems. Engineers can forecast aerodynamic forces like lift and drag, optimize designs for performance and efficiency, and obtain a thorough understanding of crucial phenomena like turbulence, shock waves, aeroacoustics, and heat transfer by using CFD to solve the Navier-Stokes and Euler equations, which govern fluid motion.

From conceptual studies to thorough product development and final validation, CFD is essential to almost every step of the aerospace design process. It significantly lessens the need for expensive and time-consuming wind tunnel and flight experiments by enabling the study of issues like external aerodynamics, aeroelasticity, propulsion, and aeroacoustic noise. Engineers can analyze and optimize shapes, evaluate performance under a variety of conditions, and make sure that aircraft and spacecraft meet strict safety and operational requirements thanks to advanced CFD methodologies. Precision simulations of complete vehicles are now possible thanks to CFD's integration with high-performance computing, which helps the aerospace industry improve design innovation, lower fuel consumption, and boost overall efficiency.

1.2. About CSIR NAL

The Council of Scientific and Industrial Research (CSIR) founded the National Aerospace Laboratories (NAL), India's top government aerospace research organization, in 1959. NAL is based in Bengaluru. With the mission to create cutting-edge technologies, design and construct civilian aircraft, and support national aerospace programs with research and development, NAL is the only civilian aerospace laboratory in the nation. Throughout its history, NAL has maintained close partnerships with leading institutions like the Indian Space Research Organization (ISRO), the Defence Research and Development Organization (DRDO), and Hindustan Aeronautics Limited (HAL). Modern wind tunnels, fatigue testing laboratories, and a distinguished history in fields such as aerodynamics, materials, structures, and flight mechanics are among NAL's facilities.

1.3 About CTFD Division

Computational and Theoretical Fluid Dynamics (CTFD) Division of NAL is essential because it supports important national aerospace and strategic programs with simulation, analysis, and optimization. Using a combination of proprietary codes, open-source tools (like SU2), and commercial packages like NUMECA and Fluent, the division is skilled at simulating intricate fluid flows. It has gained specialized knowledge in structural load prediction, ground effects, propeller simulations, control power analysis, full aircraft aerodynamic studies, and dynamic scenarios, such as store separation and pilot bailout studies. The division's researchers conduct shape optimization studies, support design validation with numerical simulations and experimental data, and make substantial contributions to the development and validation of sophisticated CFD methodologies.

By actively participating in aircraft configuration studies, aerodynamic optimization, and assisting other NAL clusters with structural, material, and electronics analyses for aerospace projects, the division upholds a creative, cooperative atmosphere. It continues to be an acknowledged center of excellence in computational aerodynamics in India, and its efforts support national self-reliance in aviation technology.

1.4 Internship Objective

Building foundational and advanced competency in computational fluid dynamics within an aerospace research context was one of the internship's many goals. The first objective was to grasp the basic ideas of CFD by comprehending the Navier-Stokes equations, which are essential for examining viscous phenomena, and the Euler equations, which control the flow of compressible fluids. This involved understanding discretization strategies such as finite volume and finite element methods as well as turbulence modeling techniques that are

necessary for precise real-world simulations.

The internship then aimed to increase competence with important CFD tools. This involved using the SU2 solver to run simulations, learning mesh generation with Pointwise and GMSH, and using Paraview for post-processing and visualization tasks. Understanding the proper simulation setup, choosing the right boundary conditions, and keeping an eye on the convergence criteria were all given a lot of attention in order to guarantee accurate results.

From analyzing classical and supercritical airfoils in both subsonic and transonic regimes to examining intricate three-dimensional wing aerodynamics and simulating store separation with full six degrees of freedom (6DOF) motion, the program also sought to apply these skills to a number of real-world aerospace engineering problems.

Another major objective was validation, comparing pressure distributions and aerodynamic coefficients from simulations with experimental data, critically assessing the simulation accuracy, and identifying possible sources of discrepancies.

1.5 CFD Softwares used and learned

The Computational Fluid Dynamics software and tools used during the internship played a crucial role in achieving the project objectives. Pointwise, an industry-standard meshing tool, was used extensively for generating high-quality surface and volume meshes, including airfoil surface meshing, structured C-mesh creation, boundary layer extrusion, and exporting grids compatible with the SU2 solver. GMSH, an open-source mesh generator, was employed for creating simple geometries and unstructured meshes, and it was also used to export mesh files for SU2. The SU2 solver, developed by Stanford University, is a powerful open-source CFD code capable of handling both compressible and incompressible flows; it was used to simulate various test cases such as airfoil flows, the RAE2822 supercritical airfoil, the ONERA M6 wing, and store separation problems with six degrees of freedom (6DOF) motion modeling. SU2's capabilities, including inviscid Euler and RANS turbulent simulations, convergence monitoring, and extraction and plotting of pressure coefficient (C_p) data, were extensively utilized. ParaView, a scientific visualization tool, was used for post-processing the CFD results, enabling visualization of pressure and velocity contours, plotting C_p versus chordwise position, examining wing spanwise sectional views, and visualizing streamlines, shocks, and vortex dynamics. Together, these tools formed an integrated CFD workflow that enabled detailed mesh generation, simulation, and results analysis for the aerospace engineering problems addressed during the internship.



Chapter 2

2.1 Initial phase of internship

During the first part of the internship, I practiced mesh generation and simulation on benchmark problems, beginning with the NACA 0012 airfoil meshing using both Pointwise and GMSH. This allowed me to concentrate on the fundamental CFD tools. I started by creating the airfoil's detailed surface geometry and then creating a structured C-mesh and O-mesh around it. This domain topology is well-known for its effective and seamless mesh distribution around streamlined bodies like airfoils and bluffs. In order to precisely capture viscous flow effects in the near-wall region, layers close to the airfoil surface were then extruded to add a fine boundary-layer mesh. For additional simulation, the mesh was exported in a format that worked with the SU2 solver.

In the simulation phase, I used the SU2 solver with the Euler equations to perform inviscid flow analysis at various angles of attack, simulating subsonic aerodynamic conditions around the airfoil. The right boundary conditions were used, such as no-penetration conditions on the airfoil surface to meet aerodynamic constraints and farfield boundary conditions modeled as Riemann invariants to simulate freestream flow. In order to guarantee numerical stability, the solver settings were carefully chosen, and residual tracking and aerodynamic coefficient stabilization were used to monitor convergence.

Following simulation completion, ParaView was used to post-process the results. Plotting pressure coefficient (C_p) distributions along the chord and visualizing pressure and velocity contours surrounding the airfoil were required for this. This first stage was essential for gaining confidence in mesh quality, solver configuration, and result interpretation as well as for becoming acquainted with the full CFD workflow. Later stages of the internship involved moving on to more complex cases and in-depth aerodynamic studies, which were made possible by the iterative learning process of meshing, simulation setup, solution analysis, and visualization.

2.2 Second Phase of Internship

The unique features of the RAE2822, a supercritical airfoil with a relatively flat upper surface and aft loading properties that contribute to its superior transonic aerodynamic behavior, were the main focus of the study. These features were intended to delay the onset of shock. Transonic flow conditions at roughly Mach 0.729 and AOA 2.31 were established using the mesh that was produced in the SU2 solver. In order to accurately represent turbulence effects, the simulations were carried out using the Reynolds-Averaged Navier-Stokes (RANS) equations with the Spalart–Allmaras turbulence model. To confirm solver stability and solution accuracy, steady-state convergence was continuously observed.

```
2
3 SOLVER= RANS
4
5 % Turbulence model
6 KIND_TURB_MODEL= SA
7
8 % Mathematical problem
9 MATH_PROBLEM= DIRECT
10
11 % Restart solution
12 RESTART_SOL= NO
13
14 % Free-stream conditions
15 MACH_NUMBER= 0.729
16 AOA= 2.31
17 FREESTREAM_TEMPERATURE= 288.15
18 REYNOLDS_NUMBER= 6.5E6
19 REYNOLDS_LENGTH= 1.0
20
21 % Reference values for coefficients
22 REF_AREA= 1.0
23 REF_LENGTH= 1.0
24 REF_ORIGIN_MOMENT_X= 0.25
25 REF_ORIGIN_MOMENT_Y= 0.0
26 REF_ORIGIN_MOMENT_Z= 0.0
27
28 % Boundary markers (must match mesh)
29 MARKER_FAR= ( FARFIELD )
30 MARKER_HEATFLUX= ( AIRFOIL, 0.0 )
31 MARKER_PLOTTING= ( AIRFOIL )
32 MARKER_MONITORING= ( AIRFOIL )
33
34 % Numerical method parameters
35 NUM_METHOD_GRAD= WEIGHTED_LEAST_SQUARES
36 CFL_NUMBER= 10.0
37 TTFD= 1000
```

```

38
39 OBJECTIVE_FUNCTION= DRAG
40

```

Figure 1: RAE2822 su2 cfg input

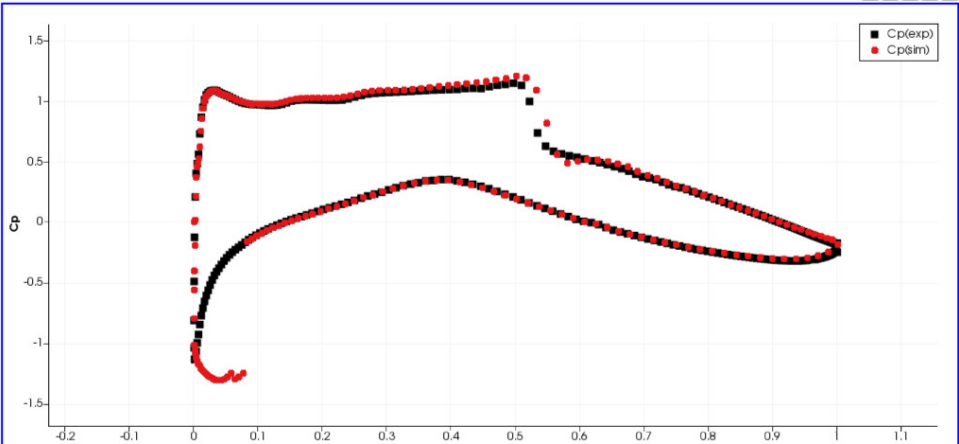


Figure 2: RAE2822 Sim vs Exp Pressure coefficient plot

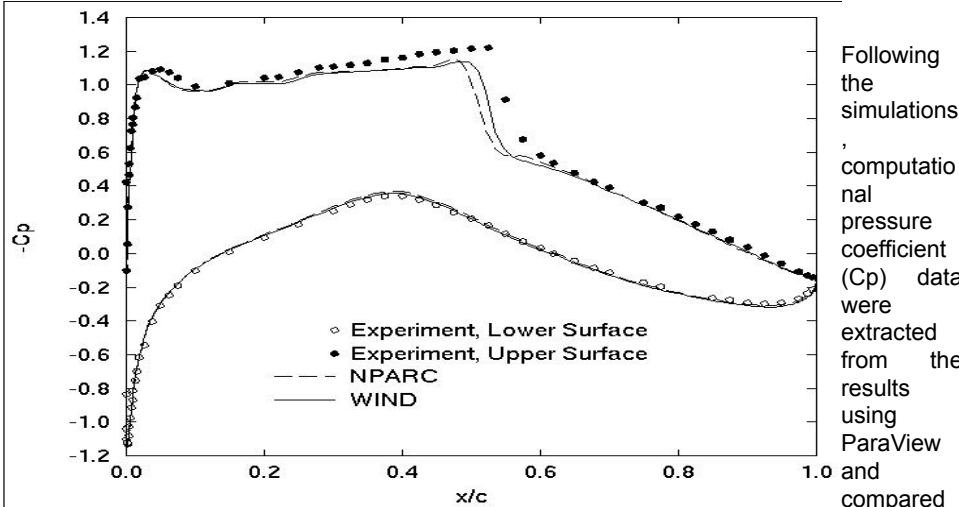
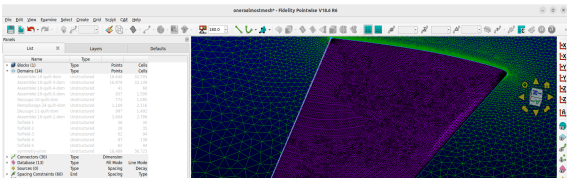


Figure 3: RAE2822 experimental Cp by NPARC

Following the simulations, computational pressure coefficient (C_p) data were extracted from the results using ParaView and compared with experimental C_p data imported from reference tests. The comparison revealed an excellent match of the shock location between simulation and experiment, generally within a small margin of difference. There were minor deviations observed in the pressure recovery near the trailing edge, but the boundary layer behavior was captured well, signifying that the turbulence model and mesh resolution were sufficient to resolve key flow physics. This comprehensive process illuminated the effectiveness of the CFD methodology in reproducing complex transonic phenomena associated with supercritical airfoils.

2.3 Third Phase of internship

A popular classical benchmark case for verifying three-dimensional wing aerodynamics and shock prediction in transonic flow regimes is the ONERA M6 wing. A thorough 3D unstructured mesh was created for the wing geometry during the internship, with particular focus on honing the mesh in the mid-span area where shock and flow characteristics are most noticeable. Complex shock-boundary layer interactions and spanwise flow variations were sufficiently captured by the mesh quality and resolution.



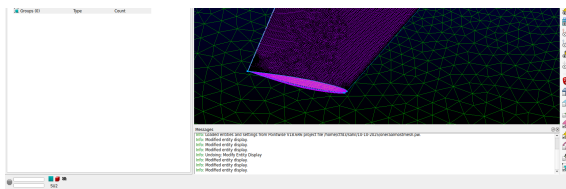


Figure 4: ONERA Pointwise Mesh

A Mach number of roughly 0.8395 and an angle of attack of 3.06° were chosen as the simulations' typical transonic flow parameters; these values are in line with the ONERA M6's traditional experimental wind tunnel tests. The Spalart-Allmaras (SA) model, which is appropriate for capturing the intricate turbulent flow characteristics seen in transonic wing aerodynamics, was used for the turbulence modeling. To guarantee numerical accuracy and stability, the solver was set up to track steady-state convergence.

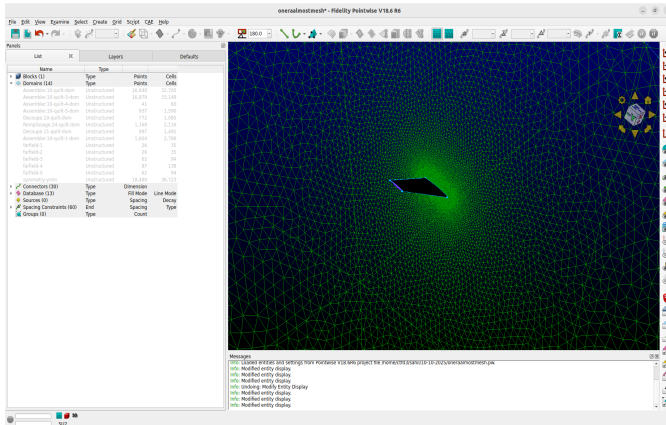


Figure 5: ONERA Farfield Mesh

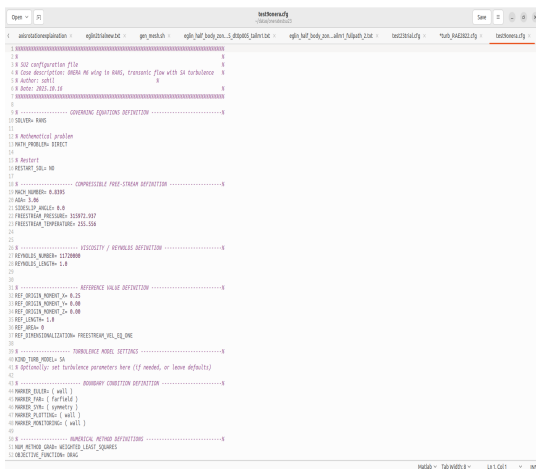


Figure 6: ONERA input su2 cfg

Pressure coefficient (C_p) distributions were extracted during post-processing along a number of spanwise stations on the wing surface, namely at normalized spanwise locations $y/b = 0.20, 0.44, 0.65, 0.80, 0.90$ and 0.95 . To confirm the accuracy of the CFD simulations, these computed C_p values were contrasted with matching experimental data from wind tunnels. The comparison revealed that the tip flow has a higher suction peak and the shock strength increases along the wing's span, both of which are in good agreement with the experimental findings. Overall, there was a reasonable match between the CFD results and the wind tunnel data, indicating that the simulation setup and turbulence models used for this classical aerodynamic validation case were reliable.

2.4 Final Phase of internship

I created the mesh in Pointwise at the beginning of the store separation simulation to guarantee excellent mesh quality and geometric representation fidelity. Applying rotation transformations based on mathematical formulations using rotation matrices was a crucial component of store motion simulation.

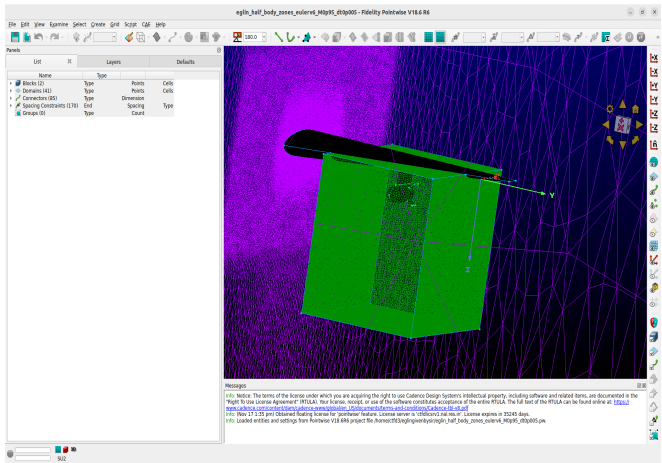


Figure 7: Eglin Wing-Store mesh at Position-0

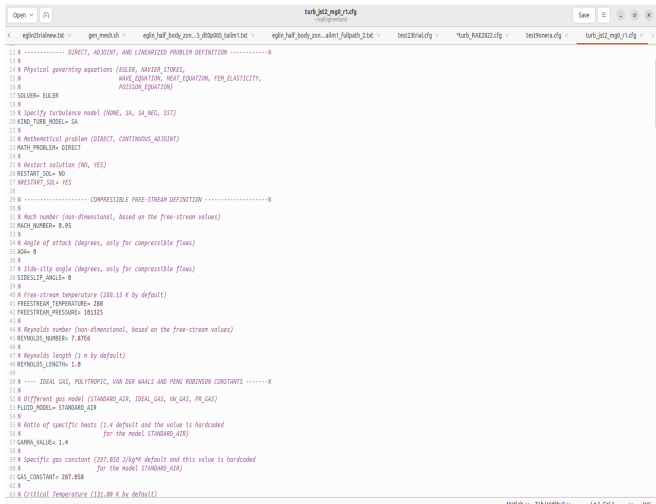


Figure 8: Eglin input su2 cfg

Through matrix multiplication, where the rotation matrix encodes the axis and angle of rotation, the new mesh point positions are determined. The entire mesh geometry could be precisely transformed in three dimensions. In order to maintain mesh integrity and simulation accuracy, this method maintains angles and distances during rotation. The axis-angle parameters were used to construct the rotation matrix, and the rotated coordinate vectors were expressed mathematically as the product of the rotation matrix and the original coordinates. Inspired by experimental data, I implemented controlled rotation and translation of the store mesh by scripting these exact coordinate transformations in Pointwise, beginning from a zero position. The store moved dynamically with both translation and rotation, implementing six degrees of freedom (6-DOF) motion, and the aerodynamic effects were then simulated using this transformed mesh in the SU2 CFD solver. Aerodynamic force and trajectory predictions were more reliable throughout the separation simulation thanks to this thorough, exacting methodology, which also made sure that the dynamic mesh faithfully represented store motions in the real world. This approach highlights the integration of advanced geometry manipulation with high-fidelity CFD simulation techniques to study complex aerodynamic scenarios.

□

Chapter 3

3.1 Problem statement

Understanding, applying, and validating sophisticated CFD techniques to simulate and analyze crucial aerodynamic phenomena pertinent to aerospace design and safety is the main challenge statement for this internship at National Aerospace Laboratories' (NAL) Computational and Fluid Dynamics Division. The internship is centered on using computational tools to solve complicated fluid flow problems, such as compressibility, viscous flows, and turbulence effects, that are mostly governed by the Euler and Navier-Stokes equations.

Accurately capturing the aerodynamic behavior of various aerospace configurations is the main challenge. These include complex three-dimensional wing aerodynamics, classical two-dimensional airfoil flows, both subsonic and transonic, including supercritical airfoils, and transient event simulations like external store separation involving six degrees of freedom (6DOF) rigid body motion. High-quality meshes that adequately resolve important flow features like wake regions, boundary layers, and shock waves must be carefully generated for each scenario.

Numerical solvers must be configured in detail for simulations, including turbulence modeling, boundary condition specification, and convergence criteria enforcement. Furthermore, validation against experimental benchmark data is essential for gaining confidence in the CFD results and comprehending model limitations due to the intricacy and nonlinearity of the flows.

Capturing dynamic mesh motion and rotating body behavior in a coupled fluid-structure interaction environment is a special challenge that comes with the project. This is especially true when modeling the translational and rotational trajectories of stores during separation. In order to ensure geometric fidelity and numerical stability of moving meshes, this calls for integrating geometric transformations through rotation matrices with solver workflows.

The ultimate goal is to create a solid, validated framework that can precisely forecast intricate aerodynamic phenomena by combining theoretical CFD knowledge with practical tool proficiency using programs like Pointwise, GMSH, SU2, and ParaView. This framework promotes independence in domestic aerospace technology development by supporting effective aerospace design optimizations and safety assessments in both research and industrial contexts..

□

Chapter 4

4.1 Results

Overall, my internship presents comprehensive findings from the CFD simulations conducted on various aerospace configurations. For the NACA 0012 airfoil, I analyzed the pressure coefficient distributions at multiple angles of attack, observing consistent trends with theoretical expectations and validating the mesh quality and solver setup. The study of the supercritical RAE2822 airfoil revealed an accurate prediction of shock location and pressure recovery characteristics under transonic conditions, aligning well with experimental C_p data.

In the ONERA M6 wing simulations, the 3D unstructured mesh facilitated capturing significant spanwise variations in shock strength and suction peaks, which matched reasonably with wind tunnel measurements.

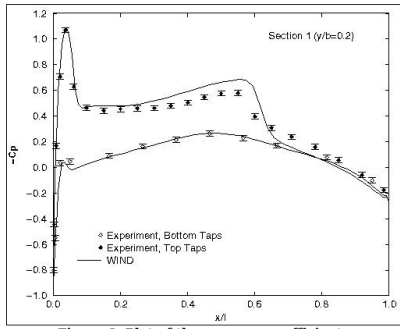


Figure 8. Plot of the pressure coefficients on the wing surface at section 1 ($y/b=0.2$).

Figure 10: ONERA C_p experimental plot at $y/b=0.20$

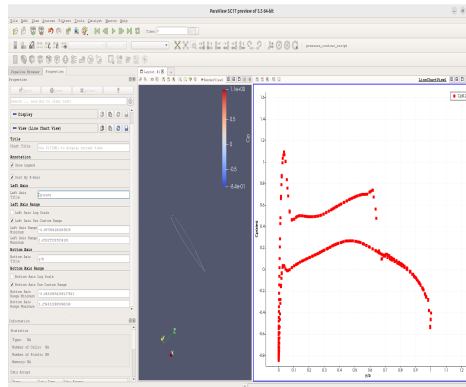


Figure 9: ONERA C_p Simulation plot at $y/b=0.2$

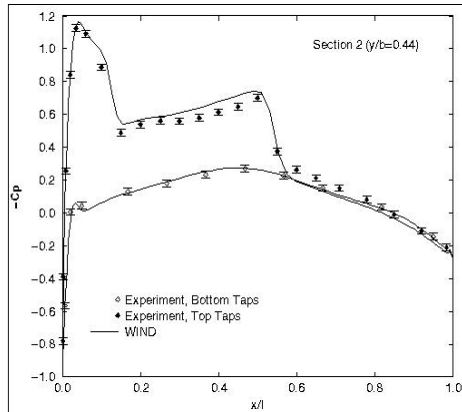


Figure 12: ONERA C_p experimental plot at $y/b=0.44$

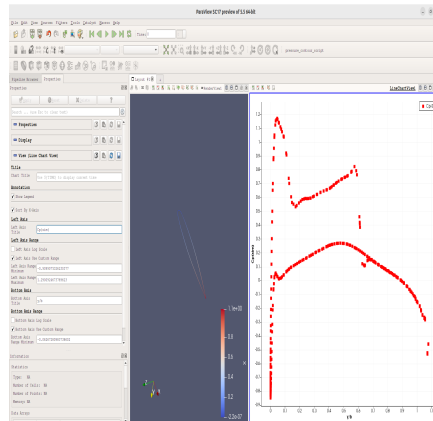


Figure 11: ONERA C_p Simulation plot at $y/b=0.44$

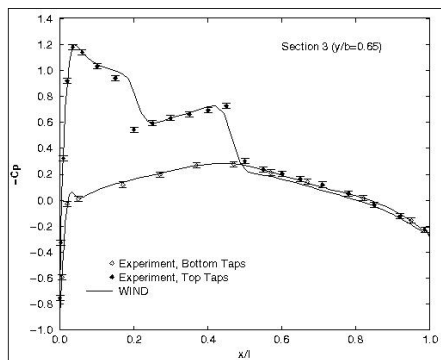


Figure 14: ONERA C_p Experimental plot at $y/b=0.65$

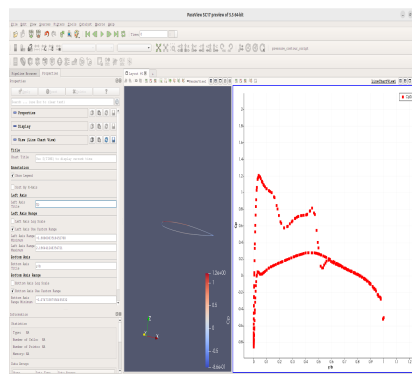
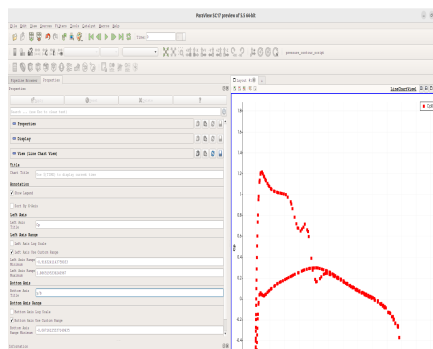
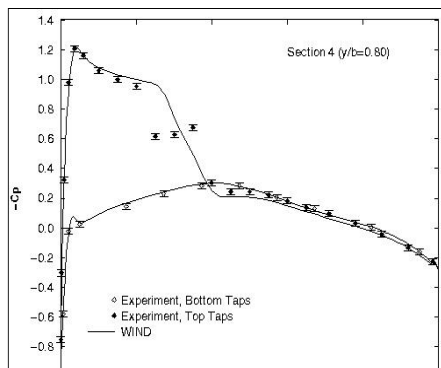


Figure 13: ONERA C_p Simulation plot at $y/b=0.65$



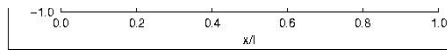


Figure 15: ONERA C_p experimental plot at $y/b=0.80$



Figure 16: ONERA C_p Simulation plot at $y/b=0.80$

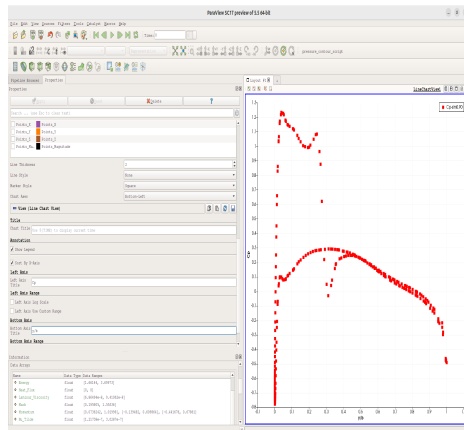


Figure 18: ONERA C_p Simulation plot at $y/b=0.90$

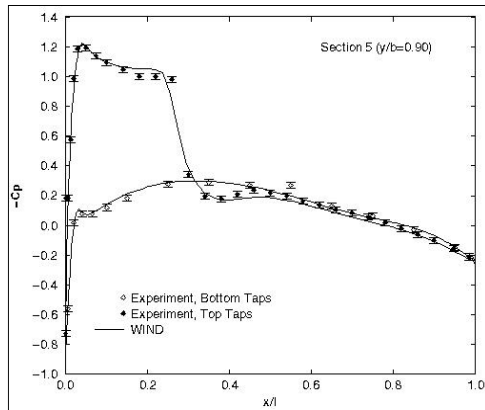


Figure 17: ONERA C_p experimental plot at $y/b=0.90$

During the store separation analysis, I applied translation and rotation transformations using rotation matrix formulations to simulate complex motion paths. The static and dynamic SU2 simulations enabled extraction of aerodynamic forces and moments, allowing assessment of the store's behavior during release. Throughout these studies, extensive post-processing in ParaView enabled visualization of flow features such as shocks, vortices, and wake structures. Additionally, mesh independence and convergence checks confirmed the numerical robustness of the simulations. These results demonstrate the efficacy of the CFD tools and methodologies used, effectively bridging computational predictions with experimental benchmarks, and highlighting areas for further refinement in future studies.

I did the store separation at different experimental positions:

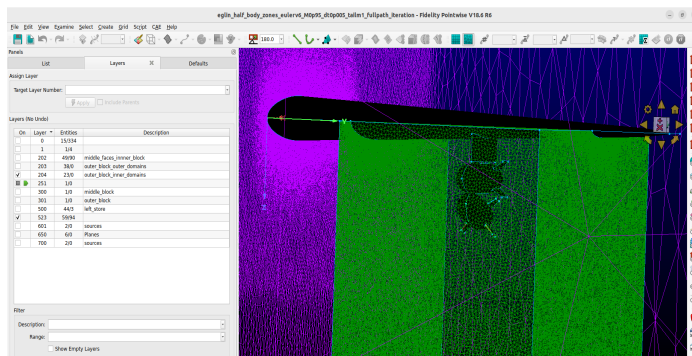




Figure 19: Eglin Store Position at 23

Values are: Translation XYZ=(0.0084, -0.007, 0.0711) & Rotation XYZ=(2.82, -5.20, -4.95) in degrees.

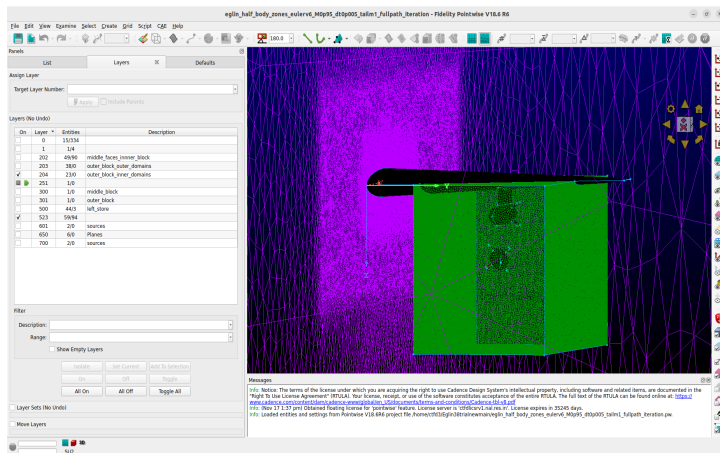


Figure 20: Eglin Store Position at 38

Values are: Translation XYZ=(0.0311, -0.0123, 0.1593) & Rotation XYZ=(6.31, -2.27, -12) in degrees.

Chapter 5

5.1 Conclusion

The internship provided an invaluable opportunity to apply and deepen my understanding of computational fluid dynamics in the context of aerospace engineering. Through hands-on experience with advanced meshing tools like Pointwise and GMSH, and rigorous simulations using the SU2 solver, I successfully analyzed a variety of aerodynamic problems from classical airfoil flows and transonic wing aerodynamics to complex dynamic scenarios such as store separation with six degrees of freedom. The integration of rotation matrix-based coordinate transformations with dynamic mesh simulations enabled accurate modeling of realistic aerospace phenomena. Validation against experimental data demonstrated the reliability of the computational approaches and highlighted the critical role of mesh quality and turbulence modeling in achieving precise results. Overall, the internship strengthened both my technical CFD skills and my professional capabilities in problem-solving, data analysis, and technical communication. This experience lays a strong foundation for future contributions to aerospace research and development, particularly in advancing indigenous computational tools and methodologies for aircraft design and safety evaluation.

References

1. <https://www.grc.nasa.gov/www/wind/valid/raetaf/raetaf01/raetaf01.html>
2. <https://www.grc.nasa.gov/WWW/wind/valid/m6wing/m6wing01/m6wing01.html>
3. <https://apps.dtic.mil/sti/tr/pdf/ADP010735.pdf>
4. <https://su2code.github.io/>

□

-

-