CSE6730 Project: Jet Engine Flow Simulation Literature Review

Sijian Tan, Kaiqun Peng, Cheng Zhang

1 Aerospace Engineering

Jet engines are widely used in today's aerospace industry. They generate immense thrust, ensuring sufficient support for aircraft to sustain flight in the sky. The simulation of airflow through jet engines perpetually captivates the aerospace industry, with the modeling of supersonic jet engine inlet flow presenting notable challenges. The supersonic inlet serves the crucial function of transitioning external supersonic flow to subsonic flow prior to its entry into the combustion section[1]. Due to the inherent properties of supersonic flow, shockwaves are generated during ingress into the engine, inducing substantial alterations in pressure and temperature. These characteristics notably elevate the complexity and challenges of the issue.

An essential metric for assessing the performance of a supersonic inlet is the inlet pressure recovery ratio. This is computed by determining the ratio of the total pressure at the compressor entrance to the free stream total pressure [2]. Numerous studies have been conducted in the realm of supersonic inlet flow simulation. For instance, B. Alekhya employed the ANSYS CFX tool to execute simulations of 2D cone and ramp inlets, conducting a performance analysis of the supersonic inlet that encompassed a discussion on inlet pressure recovery [3]. Similarly, N. Om Prakash Raj also conducted a numerical evaluation on the performance of scramjet inlet using the comerical software ANSYS Fluent [4].

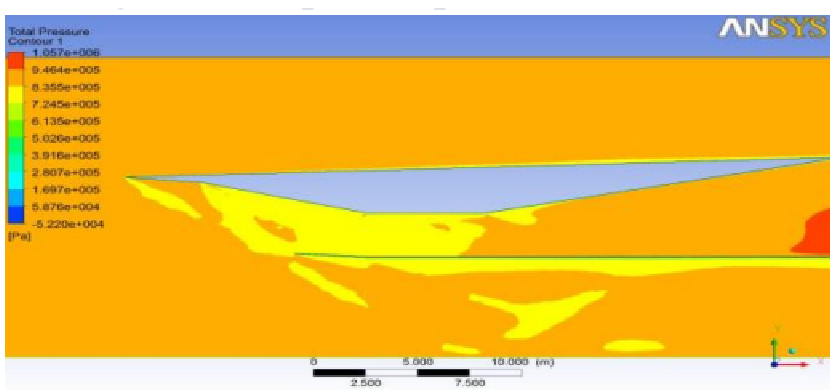

Figure 1: Ramp inlet total pressure contour [3]

2 Computational Science and Engineering

Supersonic flow simulation through two-dimensional scramjet engines has been a topic of interest within the computational fluid dynamics (CFD) community. Various methodologies, including the finite volume method (FVM), have been explored to address the challenges associated with accurately capturing shock and expansion wave structures in supersonic flows. Our approach employs the first-order, adaptive FVM, drawing inspiration from previous studies on the simulation of a supersonic laminar flow [5]. The FVM, a domain discretization method, divides the flow domain into numerous control volumes, applying the conservation laws of mass, momentum, and energy on each volume [6]. By solving the two-dimensional Euler equations, we aim to obtain the distribution of velocity, pressure, and temperature in the flow field.

Our implementation involves several steps. Initially, we will generate an appropriate grid to divide the computational domain, providing sufficient resolution to capture the flow characteristics, similar to the mesh adaptation techniques discussed in the aforementioned ANSYS study [6]. Then, we initialize the flow field and apply suitable boundary and initial conditions. In each time step, evaluate the residuals of the flow field, update the flow variables, and check the convergence criteria. To evaluate variations and convergence in the flow field, we will monitor the undivided L_1 norm of the residual vector while calculating and monitoring the output of average total pressure recovery (ATPR). Furthermore, we plan to conduct mesh adaptation iterations under different angle of attack conditions to improve solution quality, as explored in Choubey's work [7]. The post-processing steps to analyze the results include plotting and analyzing the convergence of the residual, the Mach number, total pressure fields, and the relationship between ATPR output and the number of cells in the mesh under various conditions. At the same time, our work seeks to provide a deeper understanding of the relationship between mesh resolution and ATPR output, potentially uncovering new insights into optimizing mesh configurations for improved simulation accuracy and efficiency.

3 Motivations

Most the simulations of supersonic inlet flow employed commercial CFD software, such as ANSYS Fluent, CFX. However, most of the software would not show details of the simulation, which makes the process like going through a black box. Our group is interested in applying the computational method learned from CSE 6730, including FVM, first order approximation, initial and boundary conditions, to replicate the CFD results in commercial software using Python. We will perform a CFD analysis on a simplified ramp supersonic inlet to evaluate the inlet performance, starting from mesh to final properties calculations, and validate our result with ANSYS Fluent simulation. A draft problem setup is shown below:

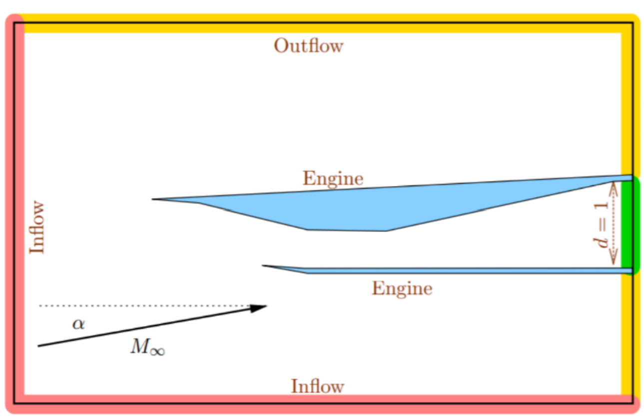

Figure 2: Problem setup

References

[1] Hongjun Ran and Dimitri Mavris. Preliminary design of a 2d supersonic inlet to maximize total pressure recovery. *AIAA 5th ATIO and16th Lighter-Than-Air Sys Tech. and Balloon Systems Conferences*, 2005. doi: 10.2514/6.2005-7357.

[2] URL https://www.grc.nasa.gov/www/k-12/airplane/inleth.html.

[3] Neela Manoj Kumar B.Alekhya. Design and performance analysis of supersonic inlets using computational fluid dynamics. *International Journal Magazine of Engineering, Technology, Management and Research*, 2016. ISSN 2348-4845.

[4] N. Om Prakash Raj and K. Venkatasubbaiah. A new approach for the design of hypersonic scramjet inlets. *Physics of Fluids*, 24(8):086103, 2012. doi: 10.1063/1.4748130.

[5] Bahia Dris and Najim Salhi. A finite volume simulation of a supersonic laminar flow: Application to a flat plate and compression corner model. *Journal of Materials and Environmental Science*, 8, 01 2017.

[6] Krishna Zore, Isik Ozcer, Luke Munholand, and John Stokes. Ansys cfd simulations of supersonic and hypersonic flows. 2020.

[7] Gautam Choubey and K.M. Pandey. Effect of variation of angle of attack on the performance of two-strut scramjet combustor. *International Journal of Hydrogen Energy*, 41(26):11455–11470, 2016. ISSN 0360-3199. doi: https://doi.org/10.1016/j.ijhydene.2016.04.048.