CSE6730 Project: Jet Engine Flow Simulation Checkpoint 1

Sijian Tan, Kaiqun Peng, Cheng Zhang

Abstract

This project conducts a simulation of supersonic flow through a 2D scramjet engine inlet, utilizing an adaptive, first-order finite-volume method implemented in Python. Emphasizing total pressure recovery calculation at the engine inlet's end, the project also offers insightful visualizations of Mach and total pressure contours. The study aims to advance supersonic inlet aerodynamics and integrate computational science and engineering (CSE) to solve complex aerospace challenges. Our implementation is available on the course github repository.

1 Introduction

1.1 Background

1.1.1 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].

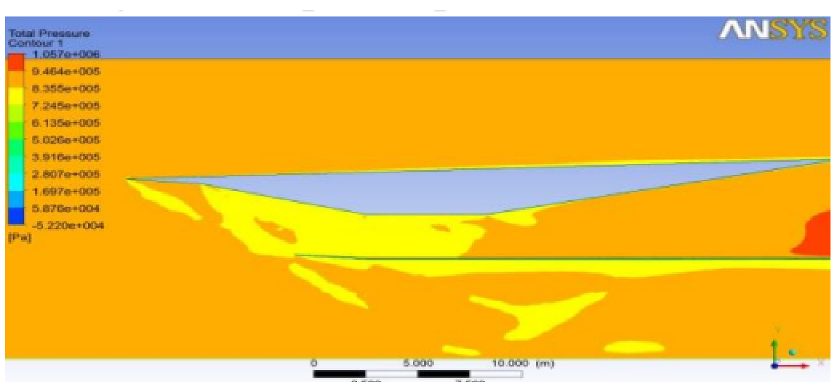


Figure 1: Ramp inlet total pressure contour [3]

1.1.2 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.

1.2 Motivations and Objectives

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.

2 System and Model Definition

The system under research consists of the scramjet engine inlet and the incoming supersonic airflow. Figure 2 presents a conceptual scheme of the system.

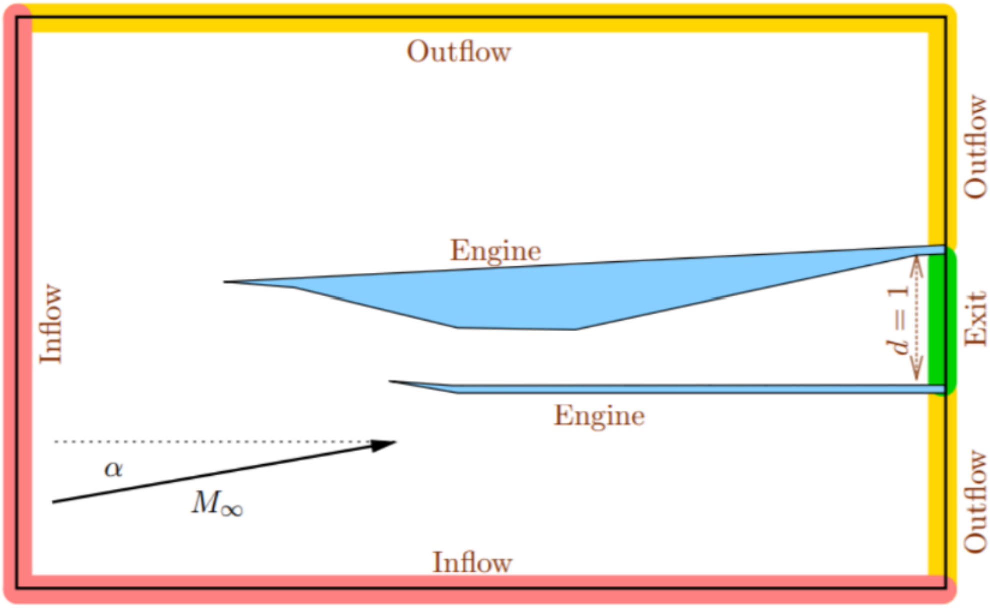


Figure 2: Problem setup

As the figure above illustrates, the system under investigation consists of a rectangular computational domain, wherein the geometry of the scramjet engine inlet is colored blue. The entire white region is filled with dry air and the focal point of this project is to figure out some key properties of the air. The boundary conditions are also crucial to the simulation and need to be cautiously defined. The left and bottom boundaries are defined as the inflow boundary, from where the air will enter the computational domain with prescribed velocity. The top and right sides are defined as the outflow boundaries, allowing the air to exit the computational domain. All the surfaces of the scramjet inlet (the blue region) are treated as adiabatic walls, making the blue region impermeable to the air. Besides, the adiabatic wall precludes the heat of air transfer into the wall.

The goal of this project in a technical way is to use CFD simulation, specifically Finite Volume Method (FVM) to solve the Euler Equations in fluid dynamics, which is a simplification of Naiver-Stokes Equations under the assumptions of **inviscid, adiabatic and incompressible** flow.

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0, \tag{1}$$

$$\frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} = -\frac{\partial P}{\partial x}, \tag{2}$$

$$\frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} = -\frac{\partial P}{\partial y} \tag{3}$$

Here:

- ρ is the fluid density.
- $\mathbf{V} = [u, v]^T$ is the velocity vector.
- P is the pressure.

There are 3 unknowns in total: u, v, P. The density ρ is assumed to be constant for the entire computational domain. After solving this, we can use the stagnation relations to calculate the total pressure before and after the engine inlet, and then get the total pressure recovery ratio.

3 Platforms

Initially, SolidWorks (Dassault Systèmes, 2023 [8]) is a computer-aid-design (CAD) software and will be employed in this project to create the CAD file of the scramjet engine geometry. Following this, the grid-generating software Gridpro (Program Development Company, 2023 [9]) will be chosen to convert the CAD file into the computational grid file. The grid file is a fundamental basis for the CFD simulation of the scramjet engine inlet. The core of this project is the CFD simulation of the flow around the scramjet engine inlet, and the code for the simulation will be written and executed in Python. Besides, Python also serves the purpose of the post-processing and visualization of the results from CFD simulation.

4 Current Progress

4.1 CAD to Mesh

A 2D CAD file is generated using SolidWorks, then exported to Gridpro to generate a .grd file. Triangular grids are used for mesh.

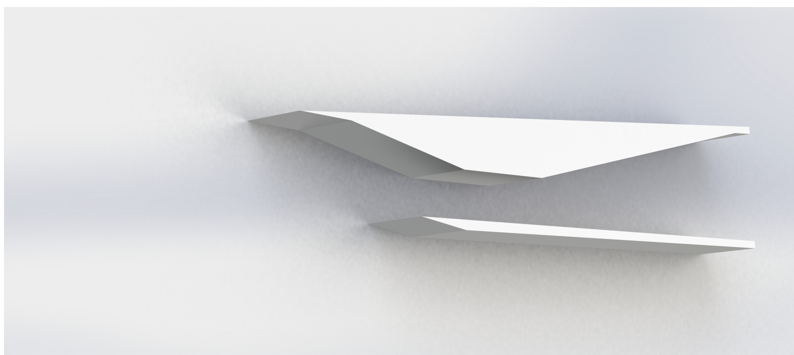
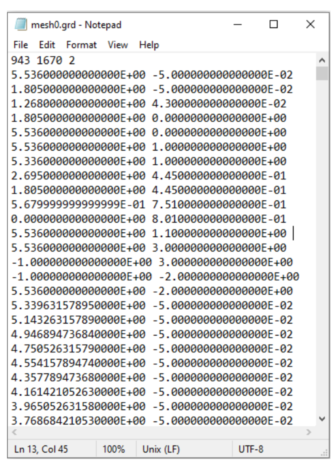


Figure 3: CAD File Generated from Solidworks



```
943 1670 2
5.536000000000000E+00 -5.000000000000000E-02
1.805000000000000E+00 -5.000000000000000E-02
1.268000000000000E+00 4.300000000000000E-02
1.805000000000000E+00 0.000000000000000E+00
5.536000000000000E+00 0.000000000000000E+00
5.536000000000000E+00 1.000000000000000E+00
5.336000000000000E+00 1.000000000000000E+00
2.695000000000000E+00 4.450000000000000E-01
1.805000000000000E+00 4.450000000000000E-01
5.679999999999999E-01 7.510000000000000E-01
0.000000000000000E+00 8.010000000000000E-01
5.536000000000000E+00 1.100000000000000E+00
5.536000000000000E+00 3.000000000000000E+00
-1.000000000000000E+00 3.000000000000000E+00
-1.000000000000000E+00 -2.000000000000000E+00
5.536000000000000E+00 -2.000000000000000E+00
5.339631578950000E+00 -5.000000000000000E-02
5.143263157890000E+00 -5.000000000000000E-02
4.946894736840000E+00 -5.000000000000000E-02
4.750526315790000E+00 -5.000000000000000E-02
4.554157894740000E+00 -5.000000000000000E-02
4.357789473680000E+00 -5.000000000000000E-02
4.161421052630000E+00 -5.000000000000000E-02
3.965052631580000E+00 -5.000000000000000E-02
3.768684210530000E+00 -5.000000000000000E-02
```


Figure 4: Grid File

The grid file contains all the information for mesh generation, include:

1. Number of nodes
2. Number of edges

3. XY coordinates of each node
4. The nodes combination of each edge

4.2 Plot Mesh

Then we use *plotmesh.py* file to plot the initial coarse mesh, as shown below:

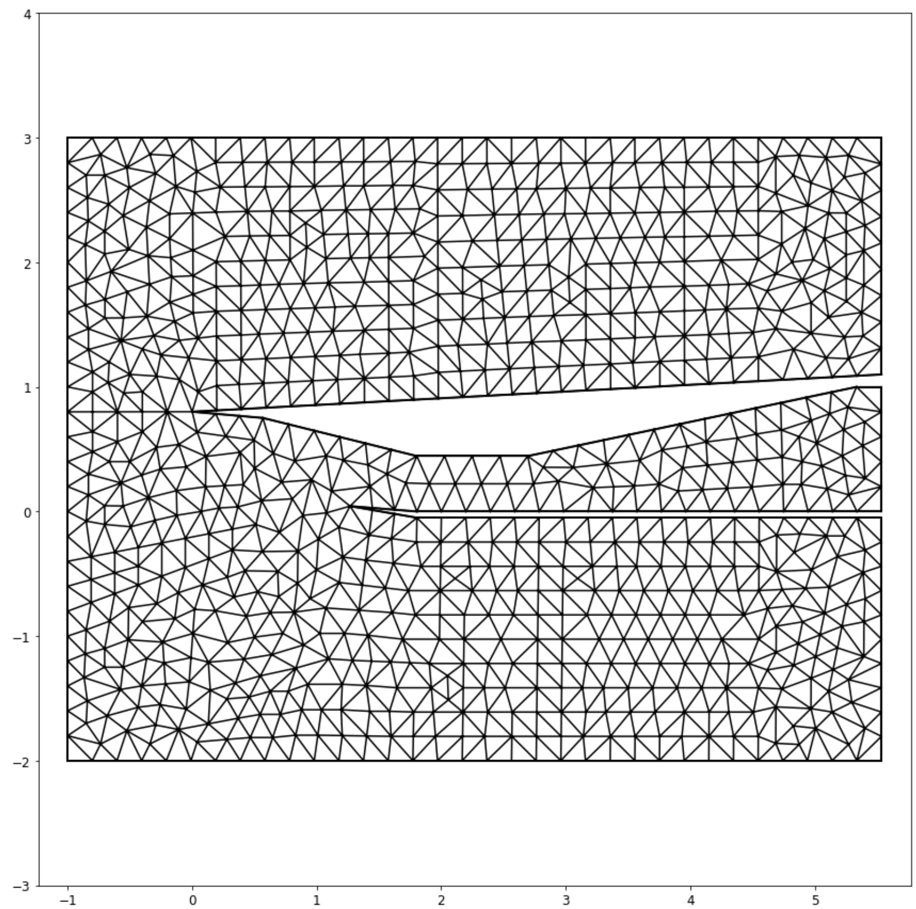


Figure 5: Coarse Mesh

5 Division of Labor

1. Sijian Tan: Roe flux implementation, Mach/pressure contour plot generation, mesh adaption, ANSYS validation
2. Kaiqun Peng: CAD, mesh generation, total pressure recovery implementation, mesh adaption
3. Cheng Zhang: Github version control, FVM implementation, convergence analysis

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.

[8] Dassault Systèmes. Solidworks. URL https://www.solidworks.com/.

[9] Program Development Company. Gridpro. URL https://www.gridpro.com/.