Final Project Proposal

Riley Jacobs

March 19, 2023

Abstract

In this final term project, I seek to analyze the effects of ramp angles and varying supersonic mach numbers for a compression ramp, particularly with respect to its treatment of oblique shocks. Numerically, I will implement a finite volume method utilizing Godunov's method with Roe's approximate Riemann solver as well as HLLE and HLLE+ schemes to approximate the numerical fluxes (superposition of Riemann problems) and solve the 2-D non-linear compressible, inviscid Euler equations.

1 Introduction

Over the past century, the ramp compression problem has been extensively studied through experimental, analytical, and numerical methods, owing to its simple geometry and easily defined boundary conditions. When a supersonic flow encounters an inclined surface, it changes direction to meet the solid boundary condition, resulting in flow compression and the formation of an oblique shock. This phenomenon is commonly observed in ramp-like or wedge-like structures, and for this project, I will consider a rectangular channel with a significantly larger depth than height, allowing for the flow to be modeled as two-dimensional. The objective of this project is to solve the steady, inviscid and compressible Navier-Stokes equations for this flow over a rectangular domain with varying ramp sizes. Figure 1, illustrates the geometry of the problem, including the necessary boundary conditions to achieve a steady-state solution. All flows investigated in this project will be supersonic, with the entrance to the domain considered as uniform flow velocity.

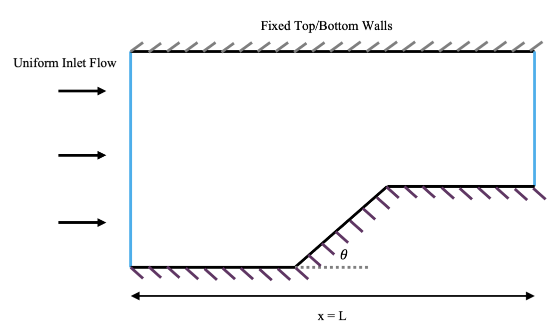

Figure 1: Supersonic Ramp Compression.

The results obtained through numerical simulation will be validated against the literature, experimental results and the analytical solution based on oblique shock theory.

2 Literature Review

The Navier-Stokes analysis of a supersonic flow over a compression ramp is a topic of special interest in the field of fluid mechanics and aerospace engineering. The study aims to analyze the behavior of a supersonic flow over a 2D wedge or ramp, which is an important component in many aerospace applications, such as supersonic aircrafts and missiles.

In one study, the schemes of ROE, HLLC, HLLC+, HLLE+, and HLLE++ were all implemented for an inviscid $M_\infty = 4$ flow over a two-dimensional 30 degree wedge [1]. Utilizing a van Albada flux limiter, it was shown that for the same number of iterations on both a medium and fine grid that HLLE++ produced the least density oscillation. That being said, HLLE+, HLLE++ and HLLC+ all provided accurate density behind the shock based on theoretical understanding. The Roe algorithm convergence improves with increasing discretization, but remains relatively error prone.

DeBonis et al. (2019) compiled eight numerical studies for a $M_\infty = 2$ flow over a 15 degree wedge using both structured and unstructured solvers [2]. They tested various combinations of spatial operators and flux limiters and investigated the Roe second and HLLE second methods with minmod limiters. Some small differences between the cases were seen up close to the region near the shock wave, and all solutions were relatively similar, except for the structured solver's Roe scheme, which produced a stronger post-shock oscillation.

In another study, Kolesnik, & Smirnov implemented an inviscid supersonic flow in a duct with a central wedge [3]. They assumed a inlet Mach number of $M_\infty = 3$ and achieved second-order spatial discretization using the MUSCL approach. They also extended their research to include implementing the Roe linearization scheme for comparison. They found that wall pressure breakdown occured in both schemes and this effect can be suppressed by introducing a limiter in combination.

Beyond the scope of this project, Chen et al. (2019) [8] conducted a numerical study of a supersonic flow over a compression ramp with different ramp motion modes, including ramp oscillation and ramp rotation. They used a high-order numerical method to solve the Navier-Stokes equations for a Mach 3 flow over a compression ramp. They found that the ramp motion had a significant effect on the flow field and the shock wave structure, and the ramp rotation mode was more effective in reducing the drag coefficient compared to the ramp oscillation mode.

In conclusion, various numerical methods have been employed to investigate shock compression interaction in supersonic flows, with the ultimate goal of understanding the behavior of the flow field and optimizing the design of aerospace components.

3 Scope and Goals

My primary objective is to simulate the conserved Euler equations across the entire domain over a specified period of time. However, during post-processing, I will specifically compare my results across the wedge to existing data in the literature to analyze the behavior of the shock wave that is generated at the step and how it propagates through the tunnel, creating an expansion fan downstream of the ramp.

To accomplish this goal, I have chosen to implement three discretization methods: Roe's linearized Godunov's scheme, HLLE (Harten-Lax-van Leer-Einfeldt) and Park and Kwan's HLLE+ flux function. All of these are well-established finite volume methods that can accurately handle shocks and discontinuities in the conserved Euler equations. Implementing all three schemes will allow for a good point of comparison of the performance of each of the algorithms for problems with strong shocks that are not aligned with the grid.

4 Problem Setup

1. Governing Equations

   For high speed flow, in the regime of supersonic and hypersonic flow, the flow is considered inviscid. Therefore, the Navier-Stokes equations can be simplified by neglecting the viscosity term ($\mu = 0$) and thermal conductivity term ($k = 0$). This simplification results in the Euler equations, which describe the motion of inviscid fluids. In two dimensions, the non-linear hyperbolic Euler equations can be written as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \underline{u}) = 0$$

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \underline{u} \otimes \underline{u}) = -\nabla P + \rho \underline{b}$$

$$\frac{\partial \rho E}{\partial t} + \nabla \cdot (\rho \underline{u} H) = \rho \underline{b} \cdot \underline{u} + s$$

   Without body forces or heat addition, the Euler equations can be rewritten as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \underline{u}) = 0 \tag{1}$$

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \underline{u} \otimes \underline{u}) = -\nabla P \tag{2}$$

$$\frac{\partial \rho E}{\partial t} + \nabla \cdot (\rho \underline{u} H) = 0 \tag{3}$$

   To compile the equations, define the state vector $\mathbf{U} = \mathbf{U}(x, y, t)$, the flux vector $\mathbf{F} = \mathbf{F}(\mathbf{U})$ in the x direction and flux vector $\mathbf{G} = \mathbf{G}(\mathbf{U})$ in the y direction. Once again, define H so we can write H as $H = E + \frac{P}{\rho}$. Then the two-dimensional, unsteady Euler equations read

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}}{\partial x} + \frac{\partial \mathbf{G}}{\partial y} = 0$$

this means

$$\mathbf{U} = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ E \end{bmatrix}, \mathbf{F} = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ u(E+p) \end{bmatrix}, \mathbf{G} = \begin{bmatrix} \rho v \\ \rho uv \\ \rho v^2 + P \\ v(E+p) \end{bmatrix}$$

combining into one flux vector

$$\vec{F} = F_x + F_y = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ u(E+p) \end{bmatrix} \frac{\partial}{\partial x} + \begin{bmatrix} \rho v \\ \rho uv \\ \rho v^2 + P \\ v(E+p) \end{bmatrix} \frac{\partial}{\partial y}$$

where $\rho([\frac{m^3}{g}])$ is the density, $\rho u([\frac{kg}{m^2 s^1}])$ and $\rho v([\frac{kg}{m^2 s^1}])$ are the x and y components of the fluids momentum per unit volume, P ($[\frac{N}{m^2}]$) is the pressure and E ($[\frac{j}{m^3}]$) is the total energy per unit volume. This means we have five equations (1 vector, 2 scalar) and six unknowns ρ, u, v, P, E, H. The following equations are then required for closure of the Euler equations and implementation in numerical simulation.

$$E = \rho(\frac{|\mathbf{u}|^2}{2} + e) = \rho(\frac{|\mathbf{u}|^2}{2} + C_v T) \tag{4}$$

$$|\mathbf{u}|^2 = (u^2 + v^2)/2 \tag{5}$$

$$p = \rho RT = (\gamma - 1)\rho e = (\gamma - 1)(E - \rho|\mathbf{u}|^2/2) \tag{6}$$

Despite their simplicity, the unsteady Euler equations enable us to exhibit the key behaviors of compressible fluids at high Mach numbers such as the formation of shocks.

2. Mesh

   To simulate the flow over a ramp, commonly found on high-speed vehicle surfaces or inlets to engines, a preliminary mesh has already been generated in Matlab. The mesh refinement process has been undertaken to ensure that the mesh is adequately refined to accurately visualize and analyze the development of the oblique shock at the base of the ramp. The following figures depict the refinement process, with Figure 2 showing the baseline mesh with basic geometry.

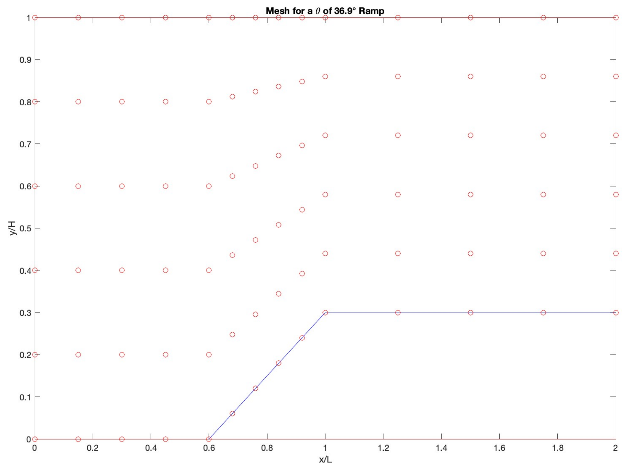

Figure 2: Preliminary Mesh in Matlab.

The mesh refinement process has resulted in significant changes from the "basic" mesh. To better capture the rapid changes in flow properties closer to the wall, a scaling or "stretching" has been applied in the y direction to create a greater number of cells. Additionally, to further refine the mesh domain, each square control volume has been divided into triangular control volumes.

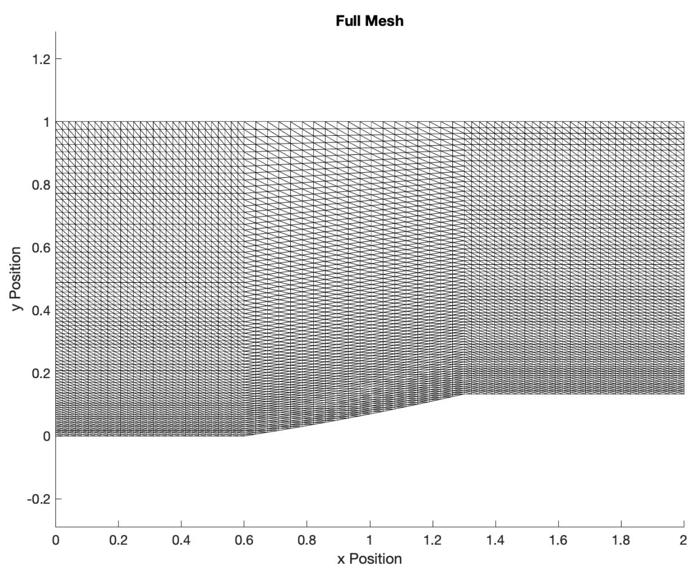

Figure 3: Overview of the refined mesh to show greater detail and resolution - especially along the ramp and aft of where the oblique shock will occur.

The current mesh consists of three major regions: pre-shock, ramp, and post-shock. Based on schlieren images and physical reasoning, it is expected that the solver will depict an oblique shock at the beginning of the ramp compression and an expansion fan as the flow turns back at the end of the ramp. This process is visualized in real life with a schlieren image, as shown below.

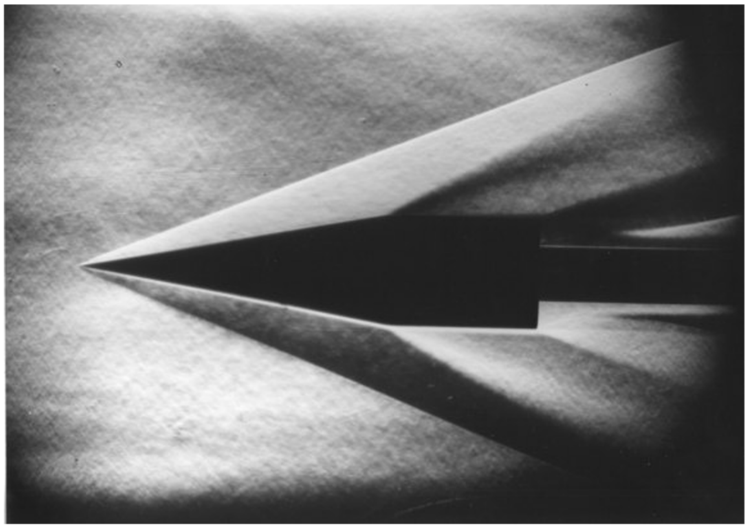

Figure 4: Schlieren image of a model resembling a supersonic aircraft body undergoing supersonic flow.

3. Grid and Boundary Conditions

   The boundary conditions for the velocity field are relatively straightforward. The inlet will ensure a uniform velocity in the x-direction, while the top and bottom boundaries will be set to zero to ensure that the no-slip and no-through boundary conditions are satisfied. Reflecting boundary conditions will be applied along the walls of the mesh. Unlike incompressible flow, there is no need to solve Poisson's equation for pressure, but we only need to account for pressure using the state equation. In summary, the "wind tunnel" will contain a uniform flow in the x-direction with density $\rho = 1$, pressure $P = 1.4$, and varying mach numbers depending on comparison to literature analysis.

5 Chosen Numerical Scheme

The literature review highlights that most compressible Euler equation solvers use structured grids to solve the Navier-Stokes equations with the finite volume method. In this method, the advective terms are solved using the popular Godunov-type (upwind) method, where the inter-cell numerical fluxes are calculated by solving the Riemann problems using reconstructed values of the primitive variables at the cell interfaces. It has been found that finite volume schemes are better than flux splitting approaches due to their superior resolution of discontinuities [5]. Thus, I have chosen to implement the superior finite volume method to simulate my ramp compression problem. The solution is then advanced in time using an explicit upwind forward Euler method due to its simplicity and ease of numerical implementation. Depending on computational cost, I might add a second step using RK2, but that has yet to be verified.

In his 1959 paper, Godunov [6] outlined his first order finite volume method. It can be described as follows:

$$u_i^{n+1} = u_i^n - \frac{h}{\Delta x}(f^n_{i+\frac{1}{2}} - f^n_{i+-\frac{1}{2}})$$

where u_i^n is defined as:

$$u_i^n = \frac{1}{\Delta x}\int_{x_{i-1}}^{x_{i+1}} U(x, nh)\, dx$$

and $f^n_{i+\frac{1}{2}}$ is defined as:

$$F^n_{i+\frac{1}{2}} = \frac{1}{\Delta t}\int_{t_n}^{t_{n+1}} f(U(x_{i=1/2}, t))\, dt$$

Godunov observed that at each cell interface the numerical flux can be described as a Riemann problem. This means that the Riemann problem at every time can be solved explicitly in terms of waves, emanating from each interface. This form is the standard form of a finite volume scheme and can be solved explicitly for the scalar euler conservation laws.

In order to use the Godunov scheme, I will begin by replacing the exact solutions of the Riemann problem with two approximate Riemann solvers. The first linearized solver is Roe. Roe solves the nonlinear equations by linearizing them. The finite volume scheme with a Roe flux is implemented by the following formula:

$$F^n_{i+\frac{1}{2}} = F^{Roe}(U_i^n, U_{i+1}^n) = \begin{cases} f(U_j^n) & A_{j+1/2} \geq 0 \\ f(U_{j+1}^n) & A_{j+1/2} \leq 0 \end{cases}$$

This particular linearized solver fails at resolving rarefractions, but as my particular problem consists of a single wave traveling to the right (or to the left depending on the sign of A), the Roe scheme should be sufficient to capture the shock phenomena present in the problem [7]. I will additionally implement the Harten-Lax-van Leer-Einfeldt (HLLE) as well as HLLE+ flux function method as it is rather simple and efficient. This particular scheme keeps only the largest and smallest characteristics, averages intermediate states in-between.

$$\vec{U}_{HLLE} = \begin{cases} f(U_j^n) & S_L \geq 0 \\ f(U^*) & S_L \leq 0 \leq S_R \\ f(U_{j+1}^n) & S_R \leq 0 \end{cases}$$

where $\vec{U^*}$ is defined as

$$\vec{U^*} = \frac{S_L(U_{i+1}^n) - S_L(U_i^n) - (\underline{F_R} - \underline{F_L})}{S_R - S_L}$$

making the flux for the HLLE scheme computed as follows:

$$F^n_{i+\frac{1}{2}} = F^{HLLE}(U_i^n, U_{i+1}^n) = \frac{S^+F(U_i^n) - S^-F(U_{i+1}^n)}{S^+ - S^-} + \frac{S^+S^-}{S^+S^-}(U_{i+1}^n - U_i^n)$$

As proposed by Park and Kwan, the wave speed estimates from the HLLE scheme with the velocity estimate from Roe's scheme can form the HLLE+ flux function numerical scheme [8]. This is described as followed

$$S_R = max(\hat{\lambda}_2, (u+a)_R)$$

$$S_L = max(\hat{\lambda}_2, (u-a)_L)$$

$$|\bar{u}| = |\hat{u}|$$

Additional constraints such as the CFL condition as well as slope limiters will be investigated to help ensure numerical stability and accuracy especially around high density changes like the shocks [7].

6 Timeline

The following is a preliminary schedule to accomplish the necessary steps of this final project.

Dates	Timeline
03/12-03/18	Perform additional literature review, project setup and create mesh
03/19-03/25	Begin developing solver and work on understanding FV methods
03/25-04/01	Continue to debug and generate initial results
04/02-04/08	Post processing of the results and perform stability and convergence analysis
04/09-04/15	Produce an outline and first draft of written report
04/16-04/22	Finalize all written report details and figures
05/01	Turn in final report

References

[1] Burning, Nichols, R. H., & Tramel, R. W. (2009). *Addition of Improved Shock-Capturing Schemes to OVERFLOW 2.1.*

[2] DeBonis, J. R. (2011, January 06). Mach 2.0, 15 degree wedge. Retrieved March 16, 2023, from https://www.grc.nasa.gov/WWW/wind/valid/wedgeM2/wedgeM2.html

[3] Kolesnik, & Smirnov, E. M. (2018). Some aspects of numerical modeling of inviscid supersonic flow in a duct with a central wedge. *Journal of Physics. Conference Series*, 1038(1), 12133–. https://doi.org/10.1088/1742-6596/1038/1/012133

[4] S. Park, C. Lee, & K. Kang. (n.d.). Navier-Stokes analysis of a supersonic flow over moving compression ramp. In *39th Aerospace Sciences Meeting and Exhibit. https://doi.org/10.2514/6.2001-568*

[5] Toro. (2009). *Riemann solvers and numerical methods for fluid dynamics*: *a practical introduction / Eleuterio F. Toro. (3rd ed.)*. Springer.

[6] Sergei Konstantinovich Godunov. “A difference method for numerical calculation of discontinuous solutions of the equations of hydrodynamics”. In: *Matematicheskii Sbornik* 89.3 (1959), pp. 271–306.

[7] LeVeque. (2002). *Finite Volume Methods for Hyperbolic Problems.* Cambridge University Press.

[8] 3Park, S.-H. and Kwon, J., “An Improved HLLE Method for Hypersonic Viscous Flows,” AIAA-2001-2633, June 2001.