ME 605 | Computational Fluid Dynamics

Project 3

Due: 11:59 pm, October 20, 2024

Instructions

1. You can choose any programming language of your choice.
2. Do not use any in-built or intrinsic functions as you are expected to write your computer program (including for solving the system of algebraic equations).
3. This is an **individual** project. While discussion among students is permitted, each student should write his/her own code and report. Each student should submit his/her individual project report and the code in the Google classroom.
4. Your report must consist of:
   (a) Problem statement
   (b) Mesh details and approach for discretization
   (c) Derivation and presentation of the final form of the discretized equations
   (d) Solution methodology
   (e) Results and discussion
   (f) Concluding remarks

   Note that an in-depth analysis and discussion of results is required.
5. The report must be prepared using WORD or LaTeX. Handwritten reports will not be accepted.

Project Statement

Fluid Flow through Converging-Diverging Rocket Nozzles

Background and Assumptions

Nozzles are devices which are used to increase the velocity of the flow. They find numerous applications; one such application that is relevant to this project is rocket propulsion. In this case, the products of combustion are expanded through a convergent-divergent (CD) nozzle to extremely high velocities (typically supersonic speeds) to generate thrust.

The goal of this project is computationally analyze fluid flow through converging-diverging rocket nozzles. To avoid excessive complexity, the following assumptions are invoked:

1. The flow is assumed to be quasi one-dimensional. In other words, flow properties are uniform across the cross-sectional area of the nozzle and therefore vary only along the axial direction.

2. The flow can be assumed to be inviscid, since the objective of the project is *not* to study the effect of walls on the fluid flow. Further, at high speeds, the convective terms in the governing equations are more important than the diffusive terms.

3. The fluid flow is assumed to be unsteady, laminar, and compressible.

4. The wall is assumed to be adiabatic. The radiative heat transfer is neglected.

Nozzle Geometry

A converging-diverging nozzle is to be considered. The nozzle geometry is described by the area function as given below:

$$A = 1 + 2.2(x - 1.5)^2 \qquad 0 \le x \le 3 \tag{1}$$

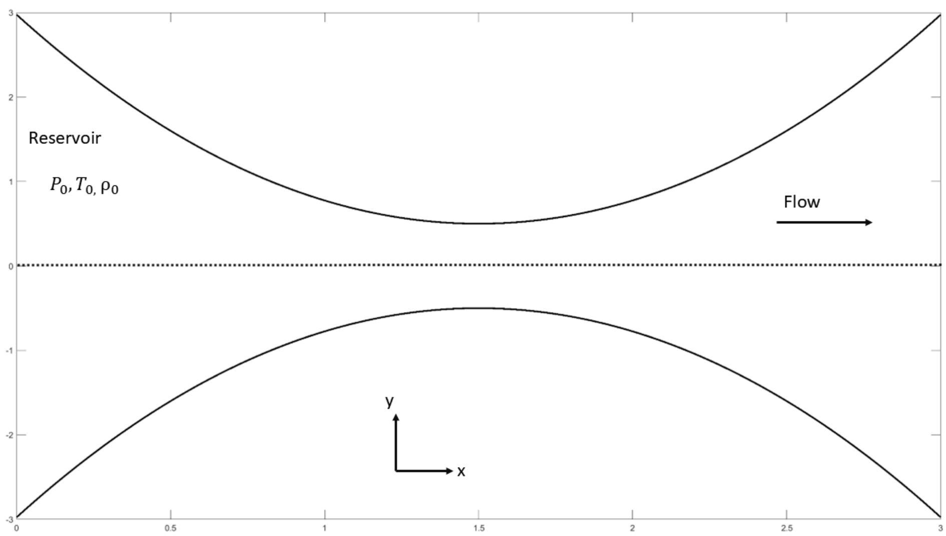

Figure 1: Converging-Diverging nozzle to be considered in the present study

Governing Equations

Since the flow is assumed to be unsteady, compressible, and inviscid, the time-dependent Euler equations must be solved. While the objective of the project is to compute the steady-state solution, the time-dependent Euler equations must be solved to ensure hyperbolicity of the governing equations across the entire Mach number regime of concern. The mass, momentum, and energy conservation equations for the quasi-one-dimensional flow are given below:

Continuity

$$\frac{\partial(\rho A)}{\partial t} + \frac{\partial(\rho A V)}{\partial x} = 0 \tag{2}$$

where ρ is the density, A is the cross-sectional area, and V is the velocity.

Momentum

$$\frac{\partial\left(\rho AV\right)}{\partial t}+\frac{\partial\left(\rho AV^{2}\right)}{\partial x}=-A\frac{\partial P}{\partial x} \tag{3}$$

where P is the pressure.

Total energy equation

$$\frac{\partial}{\partial t}\left[\rho A\left(e+\frac{V^{2}}{2}\right)\right]+\frac{\partial}{\partial x}\left[\rho AV\left(e+\frac{V^{2}}{2}\right)\right]=-\frac{\partial(PAV)}{\partial x} \tag{4}$$

where e is the internal energy.

Note that the above equations are in the **conservative** form. It is referred to as the conservative form, since the equations describe evolution of mass, momentum, and energy, which are typically conserved. You should discretize the governing equations as given in the conservative form. You should **NOT** apply product rule to convert the equations into the non-conservative form in order to solve for primitive variables such as velocity, density etc. directly. The non-conservative form is highly susceptible to numerical instabilities and conservation issues and should be avoided in general. It is bound to fail in situations where there are shock waves in the solution domain, since the primitive variables change abruptly across the shock wave, while the conserved variables do not change.

Numerical Schemes

You should use the Finite Difference method (FDM) to discretize the governing equations. It is sufficient to use explicit method in this project. You should use MacCormack's scheme to ensure second order accurate discretization in both space and time. Since you will implement an explicit scheme, the Courant (CFL) number should be sufficiently small to ensure stability. Note that the CFL number should be calculated based on the maximum wave speed (Eigenvalue) for the problem.

Simulation Cases

The flow regime in a converging-diverging nozzle is dictated by the conditions at the inlet and the outlet. In a typical experiment, the nozzle is connected to a reservoir where stagnation conditions prevail. A flow is established when the pressure at the outlet (commonly referred to as back pressure) is lowered below the pressure in the reservoir (commonly referred to as stagnation/total pressure). For a given stagnation pressure, different flow regimes are established depending on the back pressure:

1. **Isentropic subsonic flow**: Here, the back pressure is only slightly lower than the stagnation pressure. The flow accelerates in the converging section and decelerates in the diverging section of the nozzle, as one would expect in an incompressible flow conserving the volumetric flow rate. The pressure follows an opposite trend. This is shown in Figure 2. Note the the Mach number is always lower than 1, ensuring subsonic conditions throughout the nozzle. As viscous effects are neglected and the walls are assumed to be adiabatic, the flow can be regarded as isentropic.

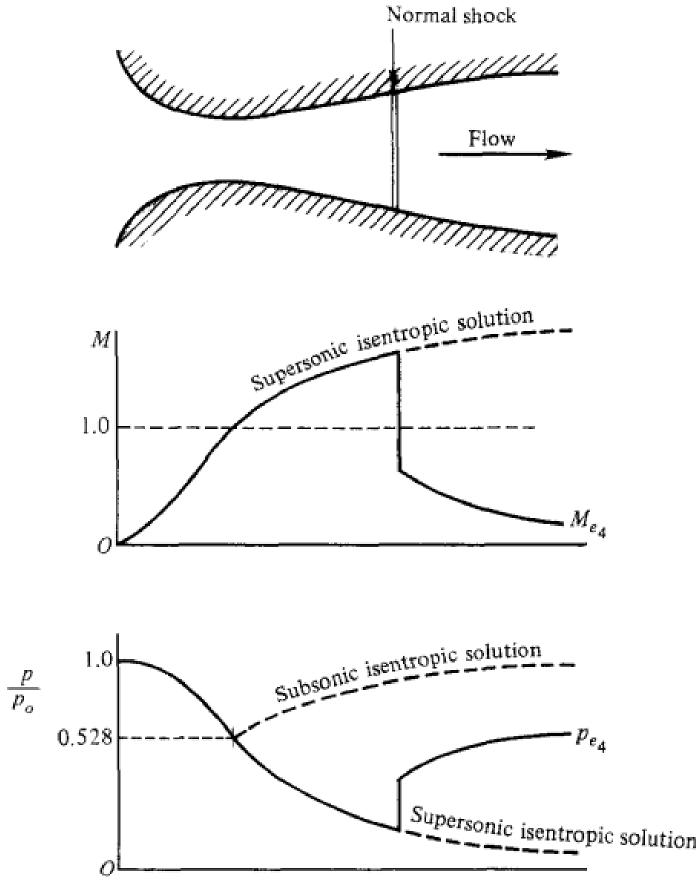

Figure 2: Flow regimes in a converging-diverging nozzle flow. From *Modern Compressible Flow* by John D. Anderson Jr.

2. **Isentropic subsonic-supersonic flow**: Here, the back pressure is significantly lower than the stagnation pressure. The flow accelerates in the converging nozzle, reaches sonic condition at the throat, and supersonic conditions at the outlet. The pressure decreases monotonically, as shown in Figure 2. As viscous effects are neglected and the walls are assumed to be adiabatic, the flow can be regarded as isentropic.

3. **Non-isentropic flow with shock wave**: Here, the back pressure is in between the above two cases. That is, it is not sufficiently low for the flow to be fully supersonic in the diverging section of the nozzle. To meet the unexpectedly high back pressure at the outlet, a shock wave is established in the diverging section. Across the shock wave, pressure jumps abruptly and the flow transitions from supersonic to subsonic conditions. The pressure evolution is shown in Figure 2. The shock wave renders the flow to be highly non-isentropic due to irreversible processes occurring inside the shock wave.

You are required to simulate all the above three cases in this project.

Boundary conditions

The following guidelines may be considered to ensure correct implementation of boundary conditions:

1. At the inlet, one can assume that reservoir conditions prevail. The pressure is equal to the stagnation pressure, density is equal to the stagnation density, and temperature is equal to the stagnation temperature. Velocity should NOT be set as zero!

2. For the isentropic subsonic flow case, the outlet pressure should be fixed as the isentropic static pressure for the specified γ, stagnation pressure p_0, and cross-sectional area at the outlet of the nozzle.

3. For the non-isentropic flow with shock wave, the outlet pressure should be taken to be equal to some intermediate static pressure between the outlet pressures corresponding to the subsonic and supersonic isentropic cases. It is recommended that you take a back pressure of 67.84 % of the stagnation pressure so that the normal shock is located in the diverging section of the nozzle.

4. Check the sign of the Eigenvalues at the inlet and outlet to decide the number of boundary conditions to be specified at the inlet and outlet for each simulation case. In other words, let characteristics guide the treatment of boundary conditions. If you are solving for N governing equations and characteristics dictate that the number of boundary conditions to be n, you should implement $N-n$ auxiliary conditions. The auxiliary conditions are commonly implemented using extrapolation techniques.

5. Since you are solving the conservative form of governing equations, the boundary conditions should also be implemented in terms of conserved variables instead of primitive variables.

Initial conditions

Since the goal of the project is to compute steady-state solutions, the initial condition can be creatively chosen to ensure convergence. You are encouraged to understand the evolution of flow variables for different simulation cases to arrive at intelligent initial conditions. For example, for the isentropic subsonic-supersonic flow case, it is known the density decreases monotonically with spatial coordinate. You could therefore assume a linear density profile (or piece-wise linear density profile) as an initial condition to ensure fast convergence. A similar approach can be adopted for other flow variables. Needless to say, using the analytical solution as the initial condition is NOT permitted.

Action Items

1. You will need to express the governing equations in the following form:

$$\frac{\partial Q}{\partial t} + \frac{\partial F}{\partial x} = S \tag{5}$$

where Q is the vector of conserved variables, F is the flux vector, and S is the source term vector. You will need to express F and S in terms of elements of Q. In your computer program, you should write flux and source term vectors in terms of conserved variable vector elements and not in terms of primitive variable vector elements. You are free to work with the dimensional form of the governing equations or non-dimensional form of governing equations.

2. You will need to write a computer program to solve the governing equations and compute the steady-state solution for the stated three cases: (1) isentropic subsonic flow, (2) isentropic subsonic-supersonic flow, (3) non-isentropic flow with normal shock wave in the diverging section of the nozzle. Take the specific heat ratio (γ) to be 1.4.

3. Plot the variation of the following quantities with axial coordinate and compare with the analytical solution:

   (a) Pressure (P/P_0)

   (b) Density (ρ/ρ_0)

   (c) Temperature (T/T_0)

   (d) Mach number (M)

4. Discuss the results in detail and provide physics based reasons to explain the trends.

Analytical solutions

Isentropic Flow Solutions

$$\frac{P}{P_0} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{-\frac{\gamma}{\gamma-1}} \tag{6}$$

$$\frac{\rho}{\rho_0} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{-\frac{1}{\gamma-1}} \tag{7}$$

$$\frac{T}{T_0} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{-1} \tag{8}$$

$$\frac{A}{A^*} = \frac{1}{M} \left[\frac{2}{\gamma + 1} \left(1 + \frac{\gamma - 1}{2} M^2\right)\right]^{\frac{\gamma+1}{2(\gamma-1)}} \tag{9}$$

$$C = \sqrt{\gamma R T} \tag{10}$$

where A^* represents the throat area and the subscript 0 refers to the reservoir state. C is the speed of sound.

Formulas for property change across the shock wave

$$\frac{P_{0_2}}{P_{0_1}} = \left(\frac{(\gamma+1)\,M_1^2}{(\gamma-1)\,M_1^2+2}\right)^{\frac{\gamma}{\gamma-1}} \left[\frac{(\gamma+1)}{2\gamma M_1^2-(\gamma-1)}\right]^{\left(\frac{1}{\gamma-1}\right)} \tag{11}$$

$$\frac{P_2}{P_1} = 1 + \frac{2\gamma}{\gamma+1}(M_1^2-1) \tag{12}$$

$$\frac{\rho_2}{\rho_1} = \frac{(\gamma+1)M_1^2}{(\gamma-1)M_1^2+2} \tag{13}$$

$$M_2 = \left\{\frac{1+\left[\frac{(\gamma-1)}{2}\right]M_1^2}{\gamma M_1^2-\frac{(\gamma-1)}{2}}\right\}^{1/2} \tag{14}$$

where the subscript 1 denotes the state before the shock wave, and the subscript 2 denotes the fluid state after the shock wave.