LAB 06 – Rocket Nozzle

Antoine BERTRAND

November 12th, 2024

Colorado State University, Fort Collins, CO 80521

I. PURPOSE

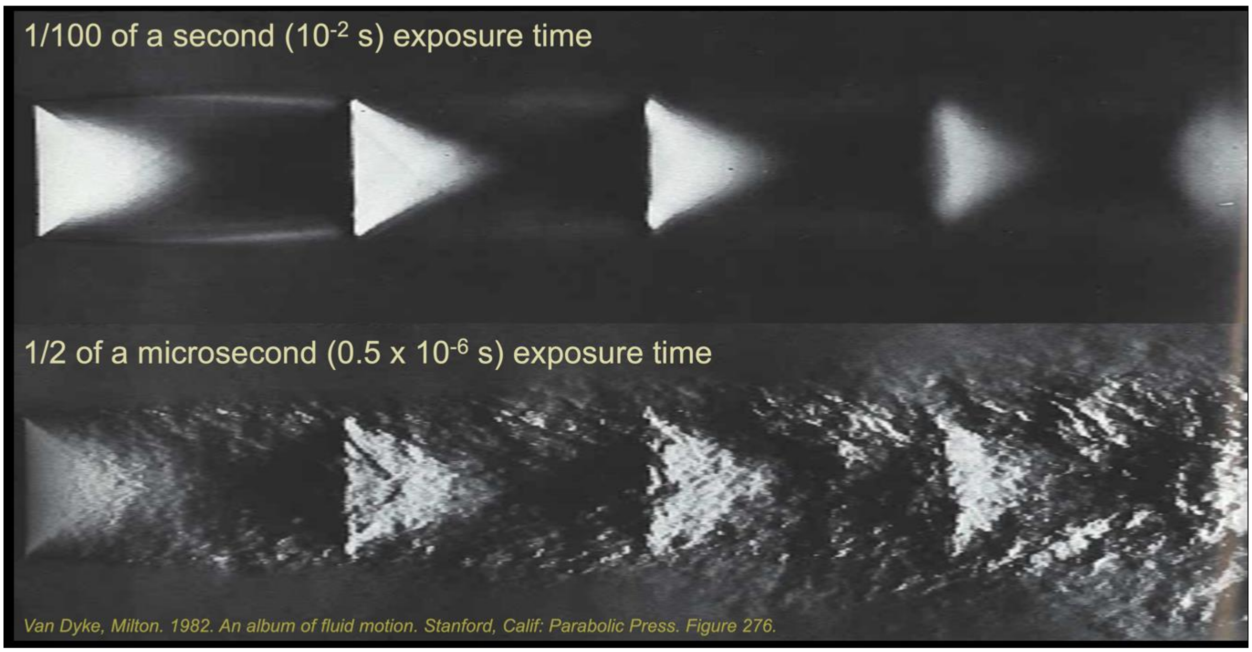

Figure 1 supersonic air jet ($Ma = 1.4$), captured at two different exposure times (top: 1/100th of a

In this lab, we explore the dynamics of compressible flow through a converging-diverging nozzle, a critical component in rocket engines and supersonic wind tunnels. The aim is to simulate and analyze high-speed flow conditions and phenomena such as shock waves, which occur at high Mach numbers, and to determine the nozzle's efficiency under varying conditions. By modeling three distinct cases—over-expanded, ideally-expanded, and under-expanded flow—the lab investigates how variations in back pressure affect the nozzle's performance.

The primary goals are as follows:

Compressible Flow Modeling: Use Fluent to simulate shock waves and high-speed flow features in supersonic conditions.

Thrust Coefficient Calculation: Quantify the nozzle's efficiency by calculating the thrust coefficient for each case, a critical performance metric in aerospace applications.

Comparison of Operating Conditions: Assess the nozzle's effectiveness in each flow condition to understand optimal and non-optimal performance and discuss real-world applications and limitations of the analysis.

Overall, this lab emphasizes understanding complex fluid behavior in nozzles and assessing how design parameters and operating conditions impact rocket and jet propulsion performance.

II. PROCEDURE

I begin by downloading and importing the provided nozzle mesh into ANSYS Fluent. I open Fluent in 2D, double precision mode, enable axisymmetric settings, and select a density-based solver for compressible flow. Next, I define the boundary conditions with an inlet pressure of 1000 kPa and temperature of 500 K, setting specific outlet pressures for each case: 60000 Pa for case G, 25000 Pa for case H, and 15000 Pa for case I. I configure the solution using the Third-Order MUSCL method, set the Courant Number to 2, and initialize using Hybrid Initialization.

I then run the simulation for 1500 iterations, refining the mesh as needed to ensure accurate results in regions with high Mach numbers. After completing the simulation, I move on to post-processing, where I plot static pressure along the nozzle's axial position and generate Mach contours for each case. Finally, I calculate the thrust coefficient using the exit values of density, velocity, and pressure from each simulation to assess the nozzle's performance under each operating condition.

III. RESULTS

Case G (Over-expanded) : Pressure inside the nozzle drops to a value that is lower than the outside ambient back pressure p_b. To restore balance, the flow compresses outside the exit in a complex series of oblique shocks until the pressure matches p_b. Thus, in this case, $p_e < p_b$.

For this case I had to adapt the cells to have the residuals converging under 10^{-6} :

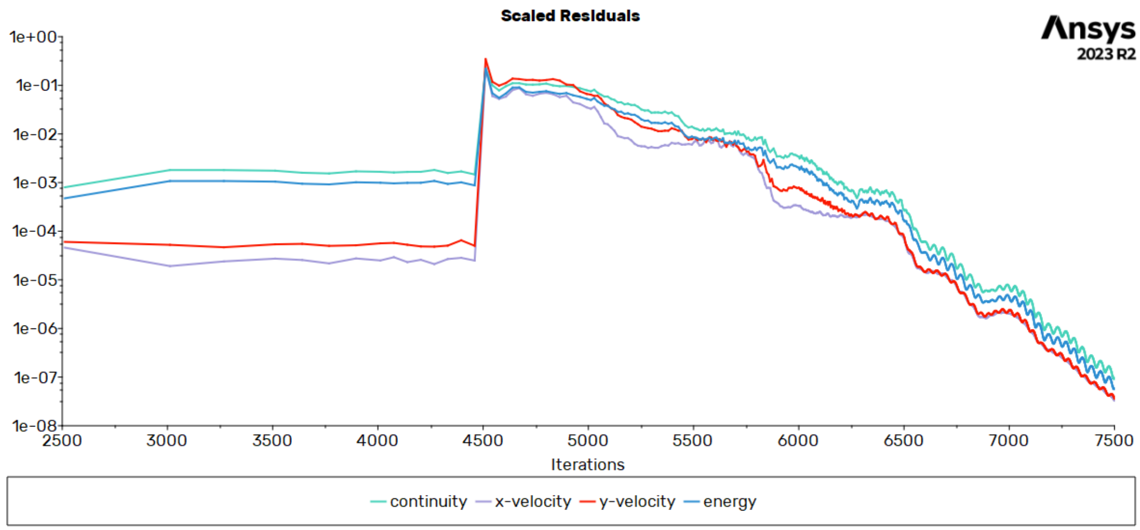

Figure 2 Scaled residuals for the case G

And the following Static Pressure vs. Axial Position plot :

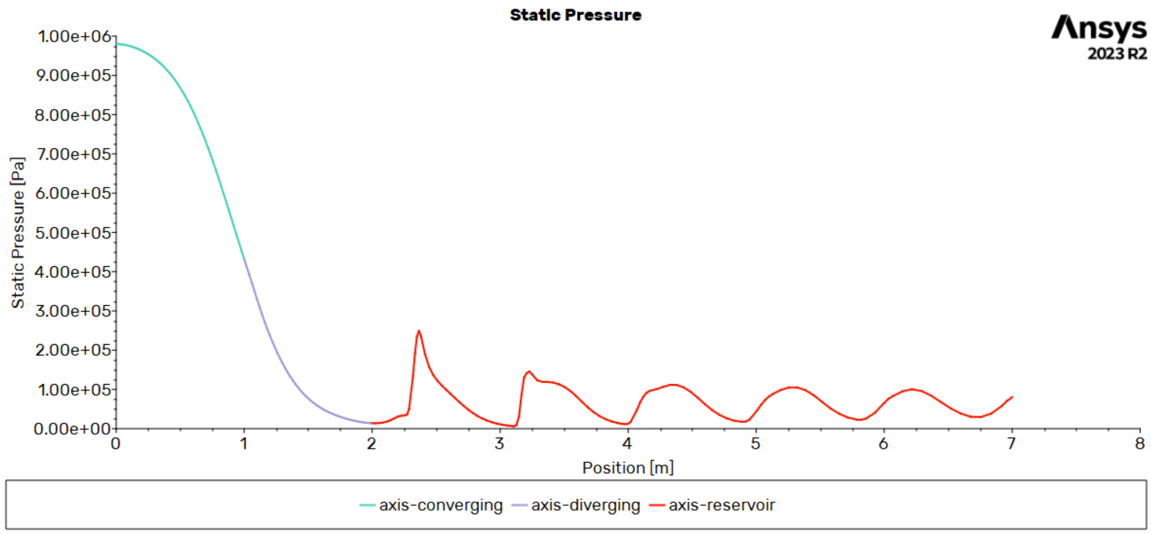

Figure 3 Static Pressure vs. Axial Position for the case G

It looks like we have the correct solution for this case. We can now create a new Mach Number contour to see how the nozzle reacts in this case :

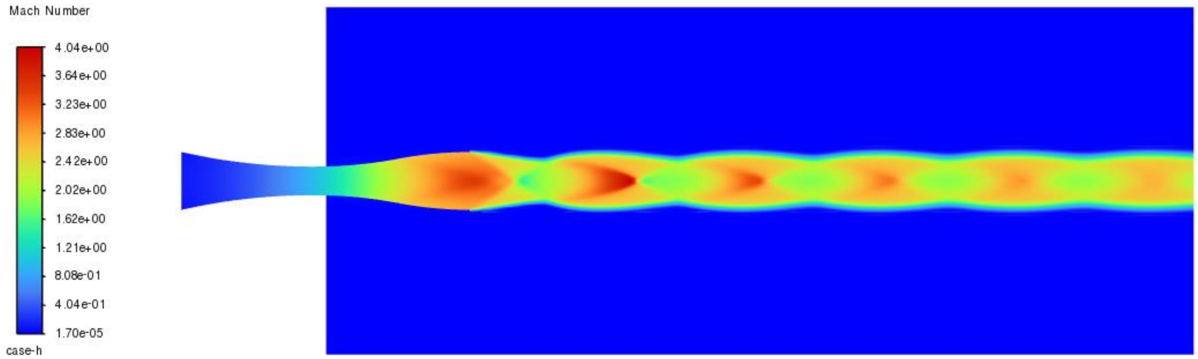

Figure 4 Mach Number contour for the case G

We can see that for this case there is a formation of oblique shocks until the pressure matches p_b.

Table 1. Variables calculated at the nozzle exit for case G

Variables	Values
Average of Density on Nozzle Exit	0.490642 [kg m^-3]
Average of Velocity on Nozzle Exit	773.134 [m s^-1]
Average of Pressure on Nozzle Exit	25536.1 [Pa]

Case H (Ideal expansion) : Back pressure p_b is lowered to a specific pressure. The diverging flow is entirely sonic, and the flow stays at the same Mach number even after the exit. This is the condition for a supersonic wind tunnel or an efficient rocket exhaust. Here, $p_e = p_b$.

For this case I didn't need to adapt the cells to have the residuals converging under 10^{-6} :

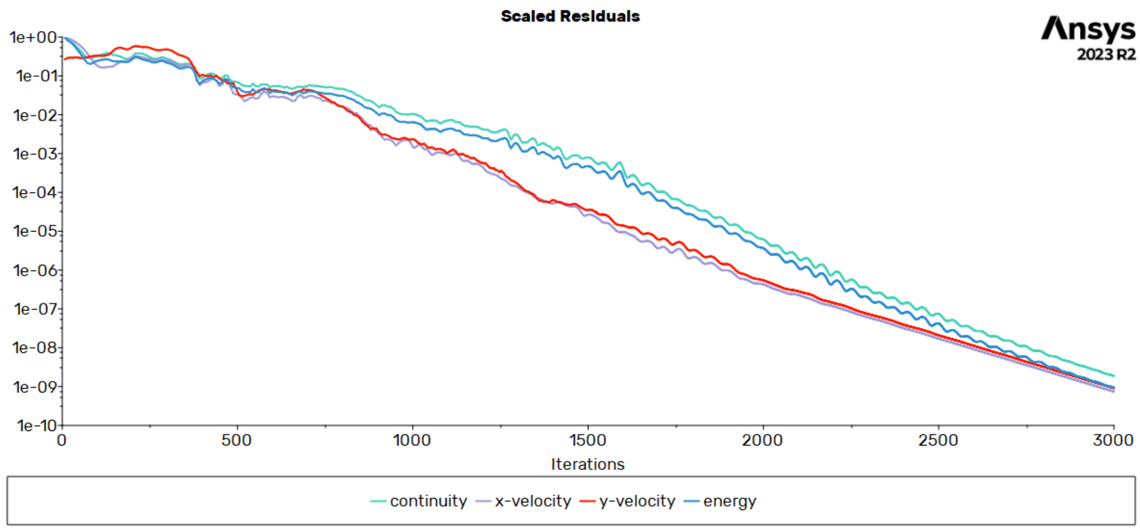

Figure 5 Scaled residuals for the case H

And the following Static Pressure vs. Axial Position plot :

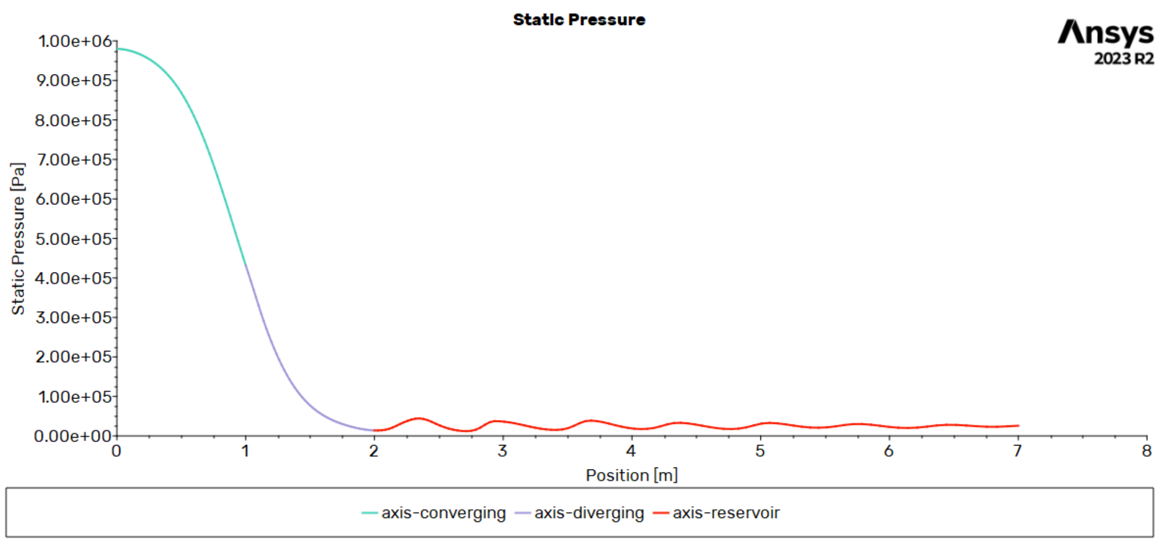

Figure 6 Static Pressure vs. Axial Position for the case H

It looks like we have the correct solution for this case. We can now create a new Mach Number contour to see how the nozzle reacts in this case :

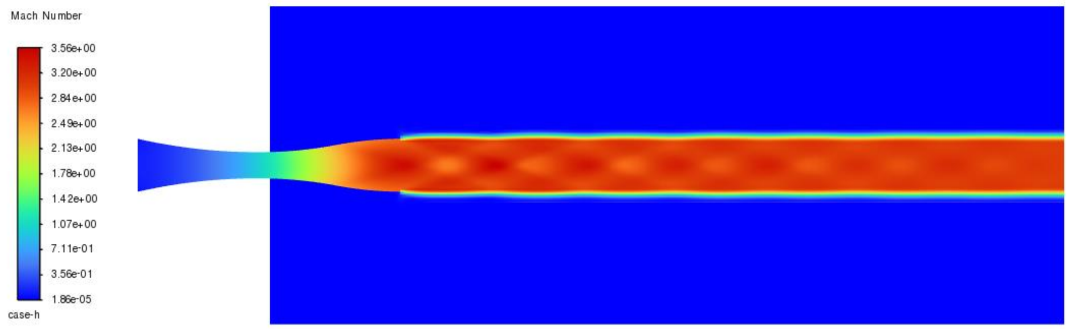

Figure 7 Mach Number contour for the case H

We can see that for this case there is no formation of shock, all flow is at the same Mach Number and the pressure shocks are way less aggressive compared to case G.

Table 1. Variables calculated at the nozzle exit for case H

Variables	**Values**
Average of Density on Nozzle Exit	0.472338 [kg m^-3]
Average of Velocity on Nozzle Exit	757.996 [m s^-1]
Average of Pressure on Nozzle Exit	24223.7 [Pa]

Case I (Under-expanded) : Pressure inside the nozzle is higher than the outside ambient air pressure. The flow compresses inside the nozzle and expands at the exit in a complex series of oblique shocks until the pressure matches p_b. Thus, in this case, $p_e > p_b$.

For this case I didn't need to adapt the cells to have the residuals converging under 10^{-6} :

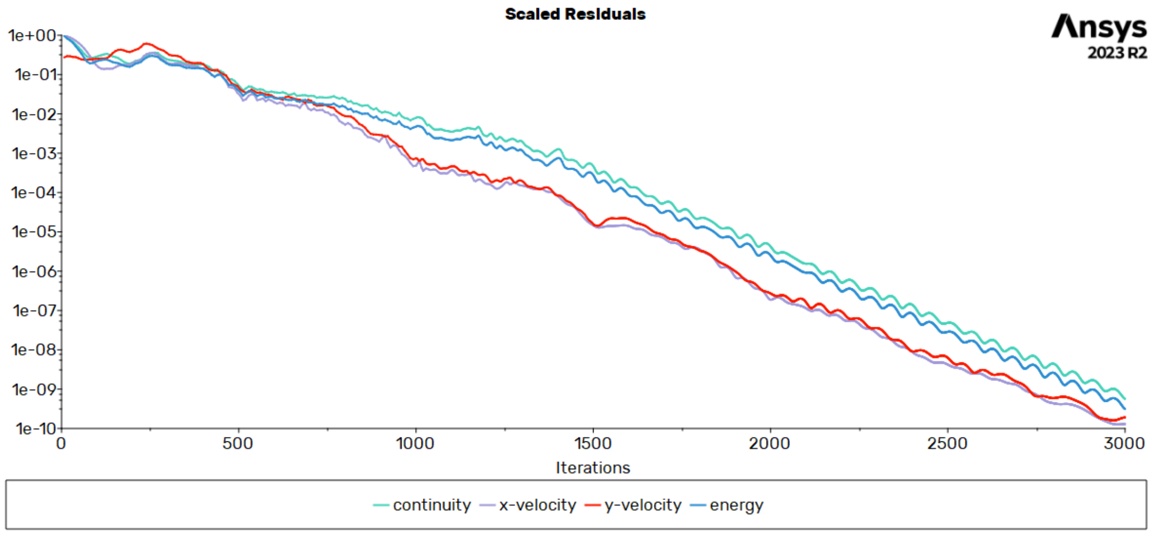

Figure 8 Scaled residuals for the case I

And the following Static Pressure vs. Axial Position plot :

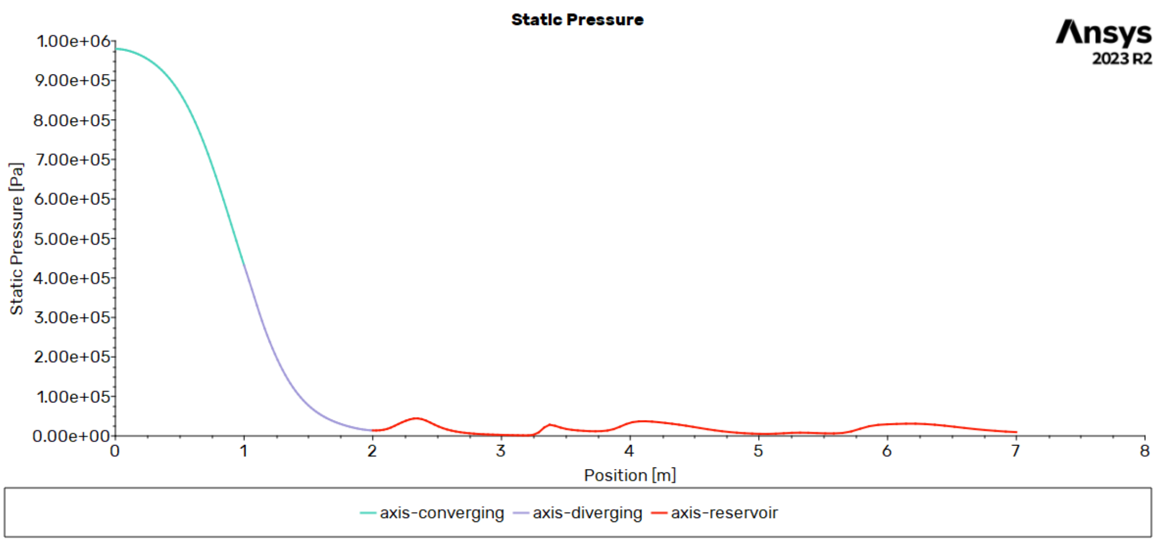

Figure 9 Static Pressure vs. Axial Position for the case I

It looks like we have the correct solution for this case. We can now create a new Mach Number contour to see how the nozzle reacts in this case :

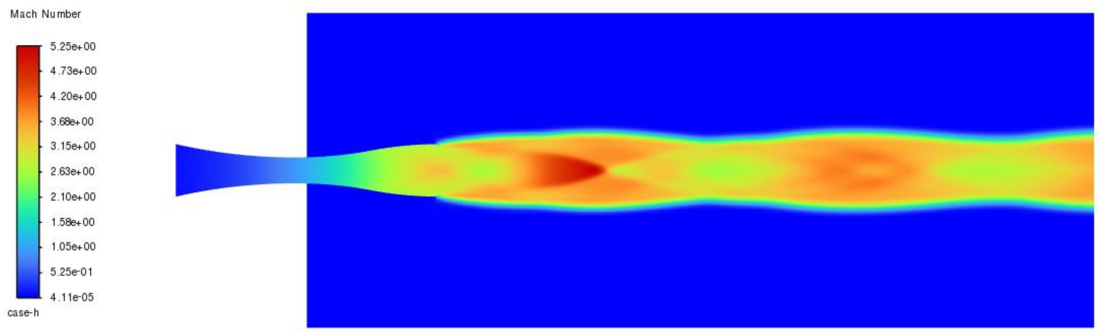

Figure 10 Mach Number contour for the case I

We can see that for this case there is a formation of oblique shocks.

Table 1. Variables calculated at the nozzle exit for case I

Variables	Values
Average of Density on Nozzle Exit	0.467133 [kg m^-3]
Average of Velocity on Nozzle Exit	758.033 [m s^-1]
Average of Pressure on Nozzle Exit	23888.2 [Pa]

IV. DISCUSSION

The thrust coefficient for a rocket nozzle is defined as :

$$C_{thrust} = \frac{F_{thrust}}{A_t p_0}$$

where A_t is the nozzle throat cross-sectional area, p_0 is the nozzle inlet pressure, and the thrust force is :

$$F_{thrust} = \dot{m}u_e + (p_e - p_b)A_e$$

Here, u_e is the nozzle exit velocity, p_e is the exit pressure, p_b is the ambient back pressure, A_e is the nozzle exit cross-sectional area, and the mass flow rate out of the nozzle is :

$$\dot{m} = \rho_e u_e A_e$$

With :

$A_e = 0.12566371 \text{ m}^2$

$A_t = 0.03141593 \text{ m}^2$

$p_0 = 1{,}000{,}000\ Pa$

Table 1. Variables calculated at the nozzle exit for case I

Case	p_b [Pa]	$\dot{m}$ [kg/s]	F_{thrust} [N]	C_{thrust}
Case G (Over-expanded)	60000	47.6682665	32523.0961	1.0352423
Case H (Ideal expansion)	25000	44.9914163	34103.3136	1.08554219
Case I (Under-expanded)	15000	44.4977985	33730.7997	1.0736847

As expected, Case H (ideal expansion), where the exit pressure matches the back pressure, is the most efficient configuration for this type of nozzle jet engine. In this scenario, no shock waves are present.

The limitation of a 2D axisymmetric analysis can be the following :

- Shock waves in real nozzles can exhibit complex, three-dimensional structures, particularly in off-axis or turbulent conditions. The 2D model captures basic shock phenomena but lacks the fidelity to represent intricate shock interactions fully.
- If the nozzle experiences uneven back pressures or side loads (e.g., due to crosswinds or non-uniform exhaust flow), a 2D axisymmetric model won't capture these variations, which can affect thrust vectoring and stability in real-world conditions.
- Secondary flow phenomena, such as eddies shedding that can occur in 3D flow, are not captured in a 2D model, which may result in a less accurate prediction of performance for certain operating conditions.

V. REFERENCE

1. White, F. M., & Xue, H. (2003). *Fluid mechanics* (Vol. 3). New York: McGraw-hill.