Rocket Nozzle CFD Optimization

By Suhani Gupta

Abstract

This project focuses on modeling and simulating different types of rocket nozzles to evaluate how different geometries can affect supersonic flow and thrust performance at both a sea-level altitude and variations of altitudes. Using Solidworks and Solidworks Flow Simulation for computational fluid dynamics, as well as Python for analytical calculations, this study aims to compare multiple nozzle profiles and identify an optimized configuration for rocket design.

Methodology

- Using Solidworks, 3 different designs with different geometries were crafted, with the following design considerations kept in mind:
    - A straight-cone nozzle: For the uttermost simplification and comparison to a variety of more complex curvatures featured in other designs, I decided to create a nozzle that featured a cross-section with no curves, in order to observe a drastic difference and understand how curves and geometry influence the aerodynamics at hand.
        - The sizing and proportions aimed to keep the integrity of a nozzle in regards to scale but miniaturizing it for 3D prints in the future. We maintained a wall thinness of 2mm for 3D printing as well as inlet and exit diameters of 40 mm, as well as a 20 mm exit diameter for a 2:1 ratio, ensuring a choked flow and the ability to accelerate to mach1 flow (via m = pAV)
    - A bell/deLaval nozzle: This geometry was designed to explore the difference between a straight shape and a curved shape. This featured a curved cross-section, unlike the straight-cone nozzle.
        - Some key design considerations included the converging and diverging angles; we decided on 25 for both after observing similar models that ranged from 25-40. We kept the throat fillet at a radius of 2-3 mm after observing aerospace literature that recommended a small circular arc upstream of the throat to avoid mesh/flow separation issues during CFD and reduce structural stress.
    - Rao-optimized nozzle: Lot's of nozzle studies and aerospace literature dives into the Rao-optimized nozzle geometry as a big contender for rocket nozzles.
        - The main design consideration for the Rao nozzle was fitting the nozzle's contour with Rao's method. Using the throat radius r_t, exit radius r_e, exit length L, convergence θ_c and divergence half-angle θ_d, we divided the contour into an initial circular arc and a parabolic section. We used the following parabolic equation:
            - $y(x) = r_t + (r_e - r_t) \cdot (\frac{x}{L} - 0.5 \cdot (1 - cos(\frac{\pi x}{L})))$
- Then, using Solidworks Flow Simulation to run compressible flow simulations with ambient pressure at sea level altitude. After obtaining heatmaps for mach number,

pressure, and velocity, we continued to run simulations at altitudes of 10km and 20km with their responding environmental pressures. Along the way, we recorded the mass flow rate, exit velocity, and exit pressure for each nozzle geometry at each altitude.

- Finally, I developed a Python script and used the recorded mass flow rate, exit velocity, and exit pressure, as well as the already calculated exit area, to calculate thrust with the following equation.
  - $F = \dot{m}V_e + (P_e - P_a)A_e$
  - I also used the Python script to generate a summary table of the data and create graphs of the nozzles at sea-level altitude as well across the different altitudes.

Results

Our contour plots captured shock-structures and supersonic expansion as a result of choked flow, and showed how smooth bell contours minimize flow separation in comparison to conical altitudes. In terms of performance, the optimized bell nozzle achieved higher thrust both at sea-level and at high-altitude conditions, out performing both the deLaval nozzle as well as the straight-cone nozzle.