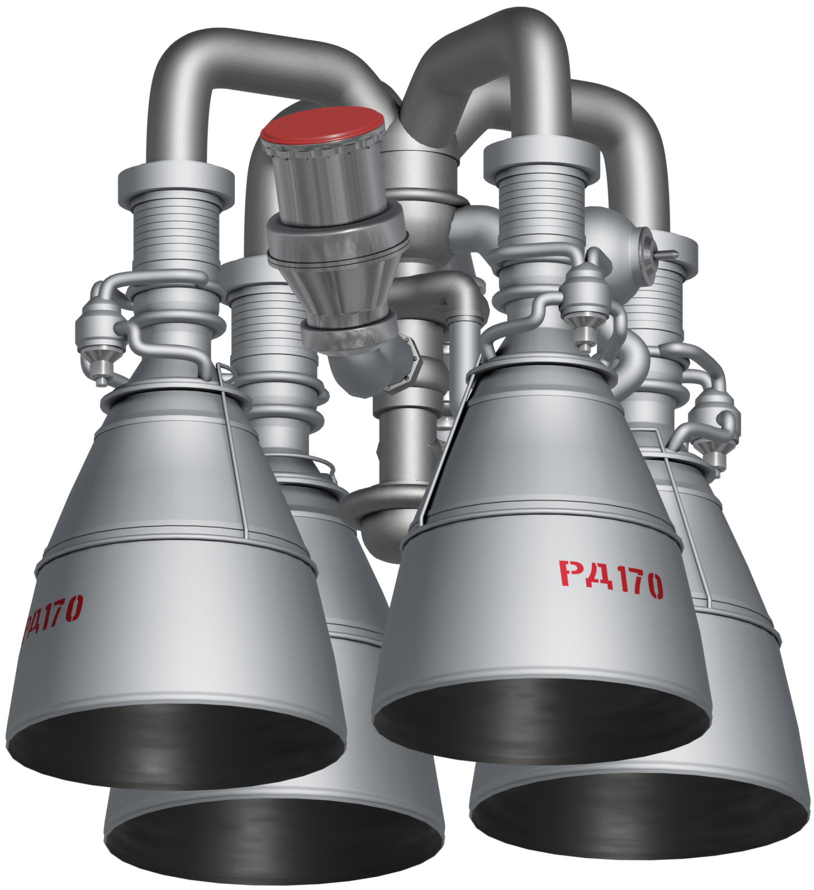
RD-170/180 Ansys Simulation



Dash Slamowitz, Lukas Wolf, Brandon Beckle
ME 495 Propulsion Project

Combustion Study

Kerosene and liquid oxygen (Kerolox) are injected into the combustion chamber of the RD-170/180 where they react explosively generating a high pressure. The working fluid is forced through a converging diverging nozzle to accelerate the flow to sonic conditions. The goal of this study was to model the combustion process and the acceleration of the flow of the fluid through the nozzle.

Mesh

We generated a mesh for the converging diverging nozzle of the RD-170/180. The dimensions of the combustion chamber, throat, and exit were acquired from internet sources. The general shape of the nozzle was estimated from photographs. The mesh has a refinement bias that favors the wall to provide higher resolution where viscous effects will take place.

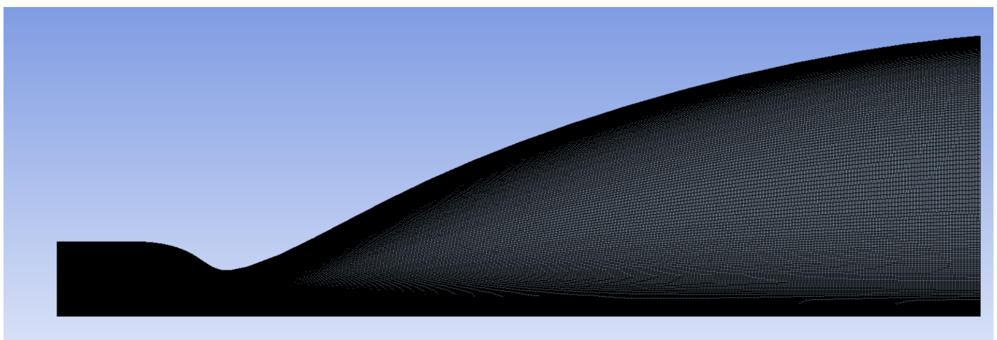


Figure 1: Combustion study mesh

Setup

The simulation was set up using the species transport model for combustion set to kerosene air. Alternating mass flow inlets for oxidizer and kerosene were arranged on the inlet side of the nozzle. The mass flow rates were set from technical data acquired for the engine in the correct 2.36:1 ratio. The nozzle outlet was set to a pressure outlet at atmospheric pressure (101325 Pa).

Results

The simulation did not produce acceptable results. The fuel and oxidizer did not mix thoroughly inside the combustion chamber resulting in incomplete combustion as seen in figure 2 below.

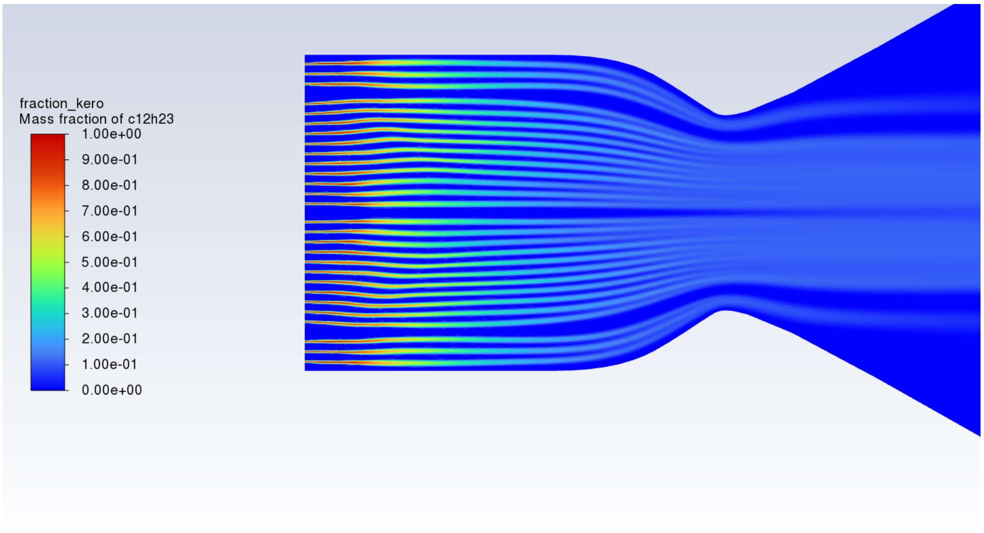


Figure 2: Incomplete combustion

The loss of energy associated with the incomplete combustion prevented the flow from reaching mach 1 at the throat resulting in a subsonic solution. As shown below in figure 3 the flow did not expand to fill the nozzle.

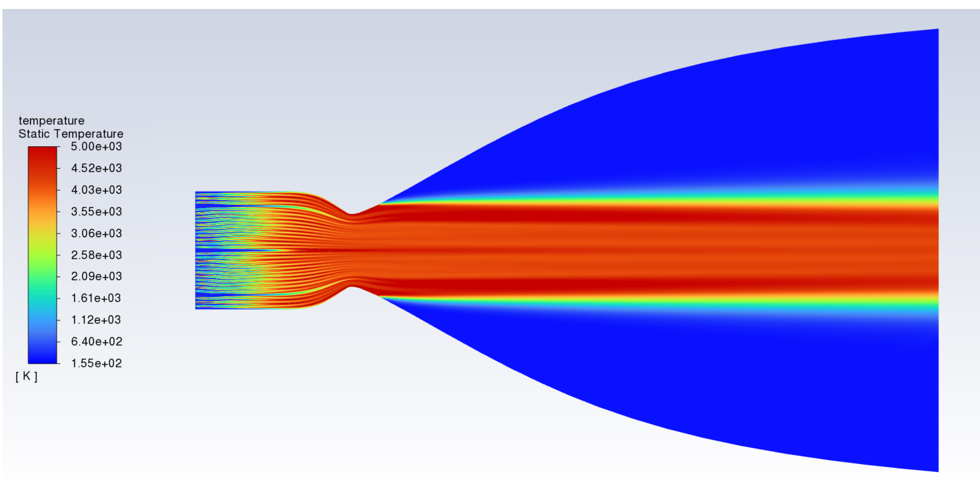


Figure 3: Failed expansion

Fuel injection systems for rocket engines are incredibly complex and details are highly classified. Improving our model of the fuel injection system is beyond our capabilities for this project.

Nozzle Flow Study

We decided to limit the scope of our nozzle flow simulation to just the flow of working fluid through the nozzle without the combustion process. The goal of this study was to analyze the acceleration of working fluid through the converging diverging nozzle from subsonic to supersonic speeds and compare the results with the analytical solution.

Mesh:

We generated a new mesh that was nearly identical to the combustion simulation, except for the reduction of combustion chamber length as it was no longer needed for the simulation.

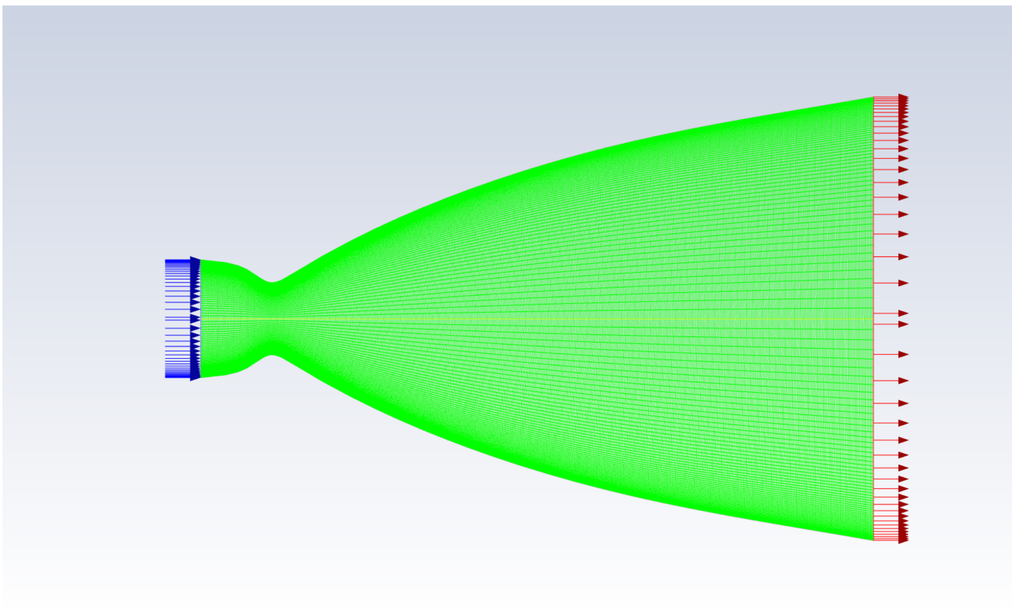


Figure 4: Nozzle flow mesh

Setup:

The inlet and outlets of the nozzle were modeled as pressure inlets and outlets respectively. The pressure on the inlet was set to the combustion chamber pressure — 24.7 MPa. The outlet pressure was set to atmospheric pressure — 101325 Pa — to simulate the back pressure associated with sea level operation. The simulation was solved using a density based solver with the fluid set to compressible ideal gas.

Results:

As shown in figure 5 below, the flow accelerated from zero velocity to greater than mach 7 through the converging diverging nozzle. The flow reaches mach 1 at the through which agrees with the analytical results. Using the isentropic flow equations for nozzles we computed the analytical results for exit mach number to be M_e = 5.5.

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

Eqn. 1

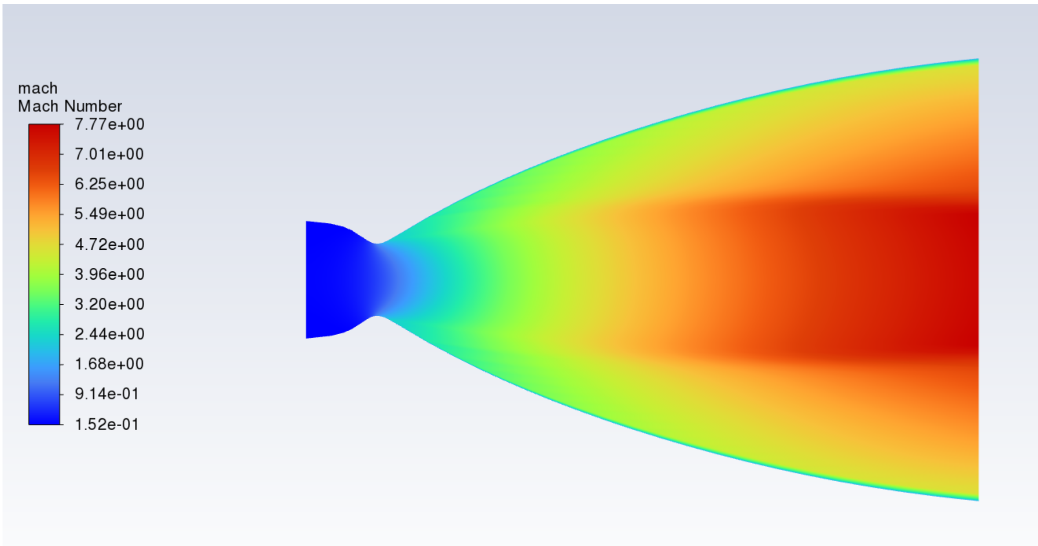


Figure 5: Nozzle mach number

The discrepancy between the analytical results and the simulation results is likely due to the formation of internal shocks that are visible in figure 5. The formation of internal shocks is confirmed by a result from a similar simulation shown in figure 6 and is caused by the fluid flow interactions with the walls of the expansion bell. These shocks are not considered by the isentropic flow equations.

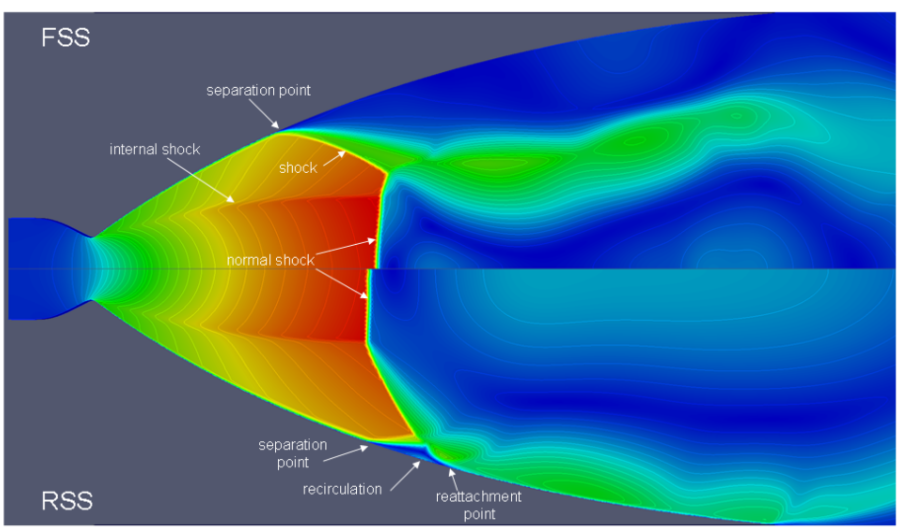


Figure 6: [Formation of internal shocks]

Jet Flow Study

With our success modeling the flow within the nozzle we decided to study the exhaust jet from the nozzle. The exhaust flow undergoes a series of shocks and expansion fans as it travels downstream. The purpose of this simulation is to study the shocks that form in the exhaust plume and the effect that the ambient pressure has on its flow.

Mesh

We generated a new mesh that included a large field downstream of the nozzle to simulate the exhaust jet formed. The length of the rectangular field is 30x the nozzle exit radius and the height is 15x the nozzle exit radius.

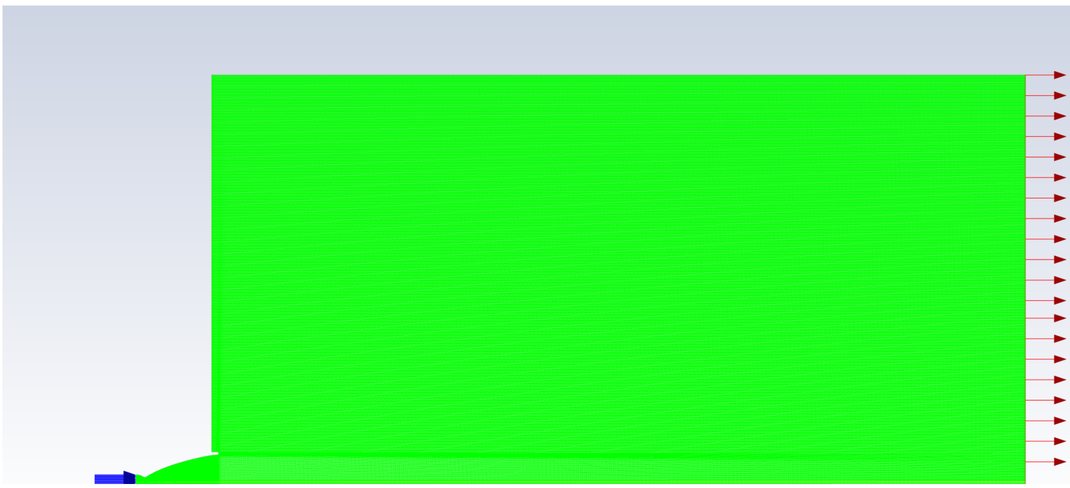


Figure 7: Jet flow study mesh

Setup

Boundary conditions were set up similar to the nozzle flow study. The inlet and outlets were pressure inlets and outlets respectively. For this study tested two operating conditions for the engine: sea level and vacuum operation. We set the outlet pressure equal to 101325 Pa to simulate the sea level operation and set it equal to zero to simulate vacuum operation. The pressure inlet remained at the combustion chamber pressure of 24.7 MPa.

Results

Getting acceptable results for this experiment was particularly difficult. The high mach numbers of the hypersonic exhaust jet proved to be very difficult for Ansys to model. Often the simulation would fail with residuals skyrocketing to greater than +1e20 (figure 8) producing nonsense results. Even successful simulations had low convergence values (figure 9).

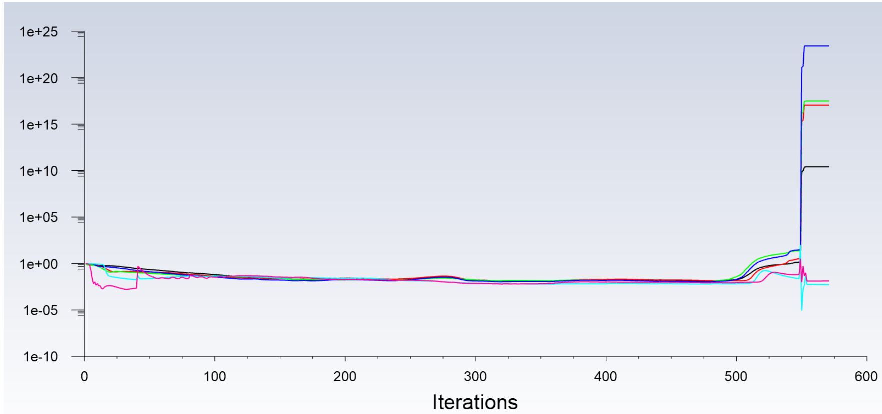


Figure 8: Simulation failure

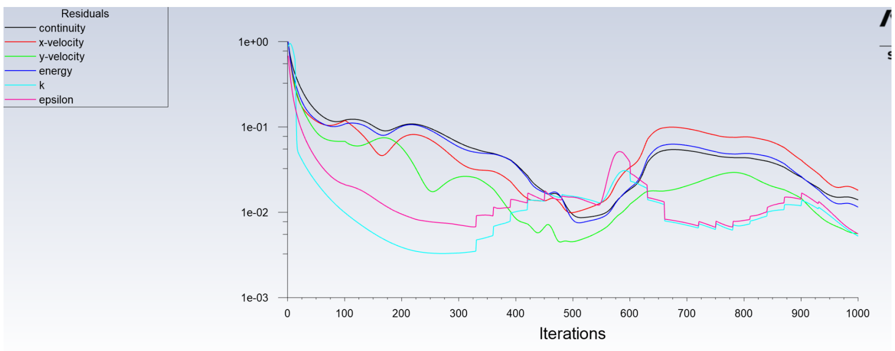


Figure 9: Low convergence

We enabled simulation steering to guide the solver towards a hypersonic solution. Using simulation steering greatly improved the stability of our simulation and we were able to produce useful results.

The mach contour plot for sea level operation (figure 10) is an example of slightly overexpanded flow. As the flow leaves the nozzle it collapses back into itself with overlapping shocks and expansion fans that form mach diamonds.

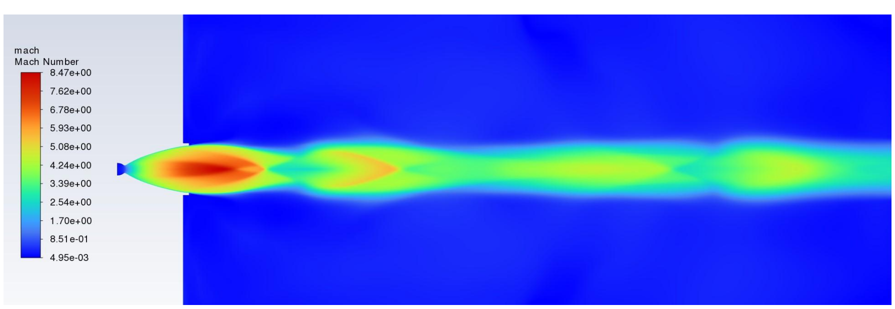


Figure 10: Mach contours for sea level operation

The flow for vacuum operation (figure 11) is clearly underexpanded. The flow rapidly expands radially past the walls of the expansion bell upon exiting forming a series of expansion fans.

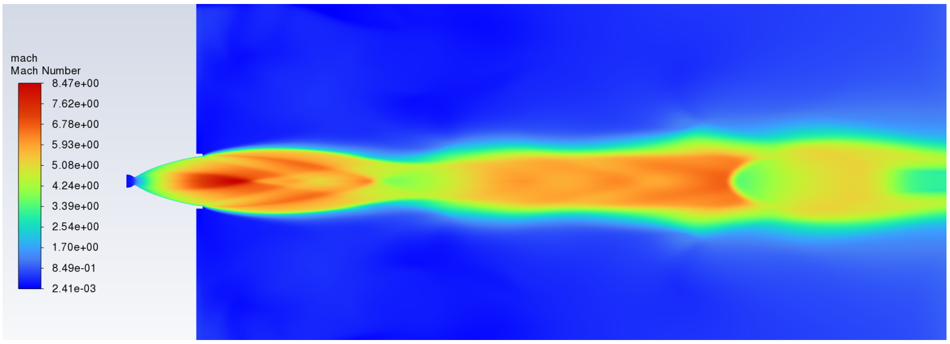


Figure 11: Mach contours for vacuum operation

The jet flow study produced valuable results that furthered our understanding of the formation of shocks in jet flows at super and hypersonic speeds. Simulation stability proved to be a difficult issue to overcome and we believe that the simulation has significant room for improvement to produce more usable and accurate results.