VIPER

Vortex Integrated Propulsion Experimental Rocket

Anthony Mellone, Grant Matthews
Nick Korotunow, Shreyas Madhvaraju

SHAPING THE FUTURE OF AEROSPACE

Abstract

Traditional rocket engines require the use of cooling methods in order to prevent system failure through the melting of the engine material. Regenerative cooling, the traditional form of engine cooling, adds weight and ultimately cost to the already complex engine. The Vortex Induced Propulsion Experimental Rocket (VIPER) project, associated with Embry-Riddle's American Institute of Aeronautics and Astronautics division, is designing a bi-propellent liquid rocket engine that will not require standard methods of cooling. A swirling combustion process is obtained through unique injection of ethanol (fuel) and nitrous oxide (oxidizer) into the combustion chamber. The swirl field centralizes the intense temperatures of combustion towards the vertical axis of the combustion chamber and creates a boundary layer of cooler gas at the engine wall. This leads to the engine walls maintaining a temperature below critical proven using Computational Fluid Dynamics (CFD) in ANSYS. With these preliminary results, only the throat of the engine will require regenerative cooling and will be accomplished through the circulation of the oxidizer. Further CFD runs utilizing multi-phase conditions will have to be performed to better simulate actual engine operating conditions. CFD results will be verified using cold flow testing where flow geometry can be recorded and analyzed. Refinement of these results will lead to an efficient and sustainable method of cooling, built for use on reusable rockets in an ever-growing, fast-paced aerospace industry.

Methodology

- Design a liquid rocket engine using MATLAB, rocket theory fundamentals, and model validation using CATIA.
- Fuel and Oxidizer determined based on cost, safety and reliability utilizing NASA CEA. All constant values featured in computations are found using theory based initial assumptions.
- ANSYS Fluent CFD is used to simulate flow to prove theory and observe behavior as a verification method for future tests.
- Cold flow test to be performed to physically observe behavior of the flow, verify flow geometry to compare with CFD results, and ensure proper orientation of injector ports.
- Manufacture final engine using Copper-Nickel alloy and perform necessary hot fire tests to ensure system requirements are met.
- Manufacture launch vehicle for the implementation of all systems
- Conduct fist launch after verified testing

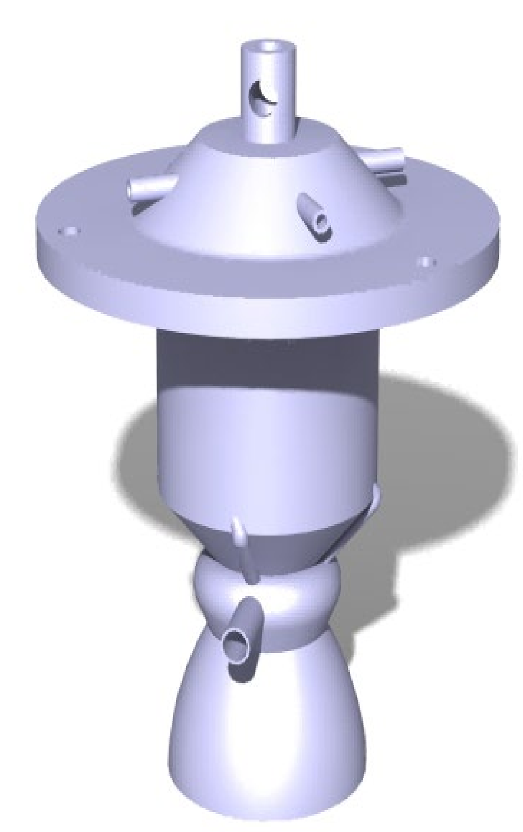
Project Background

In 2016, Sierra Nevada Corporation (SNC) was awarded a patent for an engine apparatus that would product a vortex flow field. As of 2019 this patent expired. This technology, however, is still being used by SNC for various programs. Interest in this new form of propellant injection that would lead to the mitigation of extraneous cooling methods drove Project VIPER to creation. Project VIPER is motivated to prove the effectiveness of such an engine through successful design, and integration into a flight capable launch body.

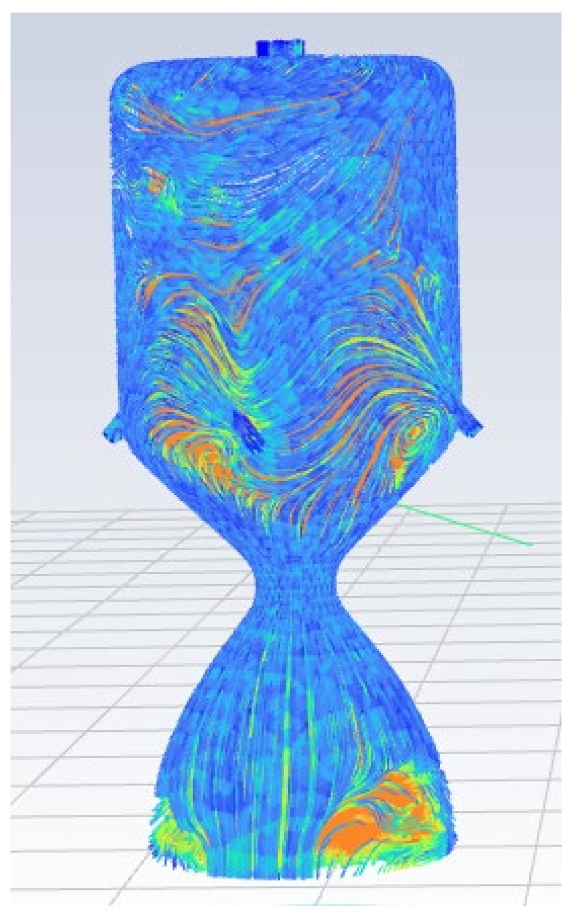
To visualize the combustion temperature field of the vortex flow, ANSYS Fluent is used on a mockup engine design.

The preliminary CFD model presents the following:

- Hottest regions of flow remain centralized and do not extend outwards towards the engine wall
- Engine wall temperatures remain below critical value, denoted by BLUE color zone in the CFD model



CAD Model



ANSYS Fluent CFD Model

Thrust	7995 N (5:1, 150 kg)
ISP	287 s
Total mass flow rate	2.840 kg/s
Fuel mass flow rate	0.738 kg/s
Oxidizer mass flow rate	2.100 kg/s
Oxidizer to Fuel ratio	2.850
Throat Area	8.630 x 10^{-5} m^2
Throat Diameter	0.0105 m
Chamber Area	7.783 x 10^{-4} m^2
Chamber Volume	4.470 x 10^{-4} m3
Length Chamber	0.5700 m
Diameter Chamber	0.0315 m

Engine Dimensions/Performance

Current Work

Current efforts are accomplished through four sub teams:

Propulsion Team
- Leading engine design efforts from engine dimensioning to propellant feed system
- Leading CFD efforts to verify flow pattern and operational temperature conditions
- Researching injector geometry and requirements to match designed flow rates and injection sites

Structures Team
- Leading CAD design for geometry sets provided by the propulsion team
- Working towards performing FEA analysis on various structures to verify structural integrity

Controls Team
- Leading efforts for parachute deployment system to ensure the reusability of the launch vehicle and engine
- Responsible for other tasks such as fin design for rocket body

Data and Command Handling Team
- Leading the design of software and hardware to collect all in flight data
- Will work with other sub teams to learn what in flight sensors are needed and will devise ways to collect and store this information

Future Work

- Refine CFD Flow to fix slight backflow and observe thrust produced, backchecking with design conditions
- Calculate mass flow rate required for fuel and oxidizer pumps into combustion chamber to cause accurate pressure difference for vortex flow to be generated.
- Perform cold flow on prototype engine to crosscheck vortex flow with CFD results and observe potency of vorticity.
- Construct final prototype of engine, capable of hot flow; perform hot fire test and observe cooling potential of vortex engine in comparison to design condition and regularly cooled sample engine.

Acknowledgments

This project would not be made possible without the great leadership of our sub team leads:

Nicholas Juganu (Structures), Anthony Mellone (Propulsion), Joshua Jones (Controls), Andres Ramirez (Data and Command Handling)