VILE

Vortex Induced Liquid Engine

Adam Duke and Michael Tomaso

Abstract

The growth of reusable space launch vehicles has led to a demand for improved rocket propulsion cooling systems which vary in degrees of complexity. Vortex Induced Liquid Engine (VILE) is a project to study and investigate cooling solutions that lead to higher engine efficiency and increased reusability for rockets. The research focuses on the design and thermal analysis to postulate a novel approach to vortex engine concepts to ensure the system's functionality and reliability is augmented by the ease of production to aid the growing field of space propulsion.

Background

Current day combustion chambers for rocket engines utilize a variety of cooling methods; most notable film cooling or regenerative cooling methods. Film cooling is the use of a thin layer of fuel on the surface of the combustion chamber that will act as a heat absorption layer that is expelled with the exhaust of the combustion products taking heat away from the walls of the combustion chamber. Regenerative cooling flows the cold fuel through the walls of the nozzle before being injected into the combustion chamber. Regenerative cooling takes heat from the combustion chamber walls keeping the temperature down. Film cooling is inefficient as excess fuel is required to cool the combustion chamber and nozzle through its entire burn time, thus increasing the necessary fuel mass. Regenerative cooling requires complicated plumbing and part design which leads to heavier engines and therefore a reduced thrust to mass ratio. VILE aims to improve the efficiency and cooling methods of combustion chambers.

Methodology

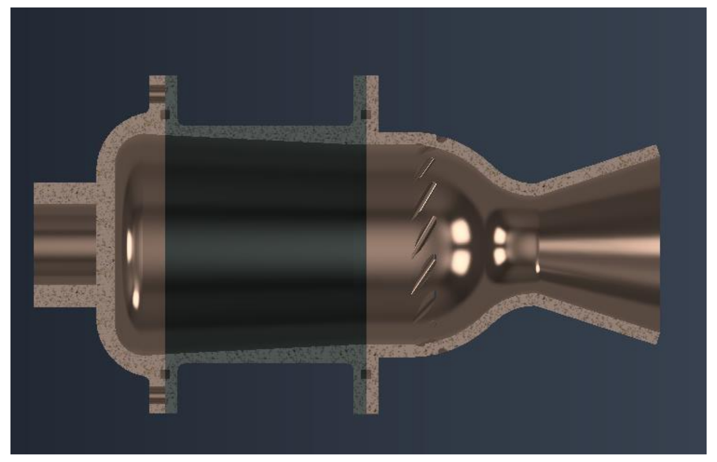
- Design a small liquid rocket engine using a predictive MATLAB model based upon theory and rocket fundamentals. Using the predictive model all design parameters are known for a CAD model to be constructed.
- Supporting hardware (sensor suite, DAQ systems, and plumbing) needs to be designed to allow for optimal testing conditions for the engine.
- Rapid prototyping and ANSYS Fluent CFD will be used to aid in the creation of better injector geometries and designs for vortex generation inside the combustion chamber.
- Final product will be integrated with sensor suite to enable data collection and analysis of vortex cooling capabilities

Hypothesis

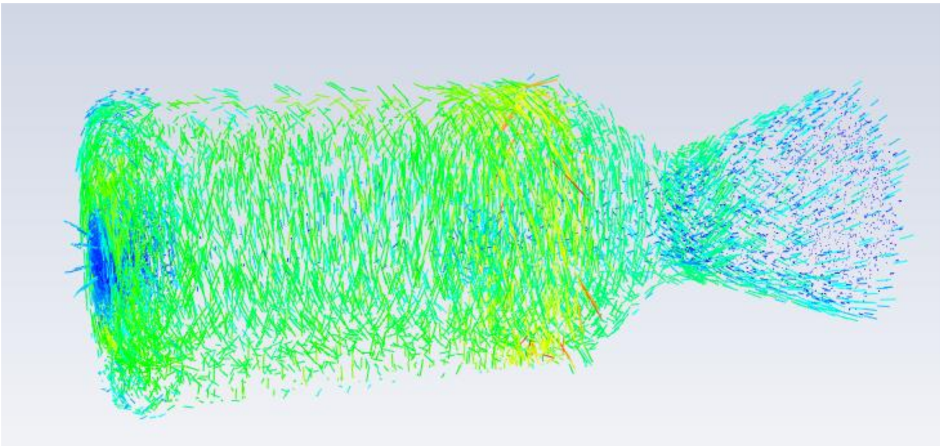
If we create a vortex that travels along the combustion chamber walls, we expect a significant quantity of heat to be absorbed by the vortex flow.

Current Work

- Currently VILE is undergoing analysis for its MATLAB model and associated version update for the CAD model. Focus is on the injector and feed systems for the fuel and oxidizer. As well the supporting hardware for data collection and analysis.
- CFD analysis is being done to visualize the different behaviors that various injector designs have on the flow within the combustion chamber.
- New injector geometries are being conceptualized and designed in CAD to improve fuel oxidizer mixing for further atomization of the fuel.



CAD model



ANSYS Fluent flow simulation

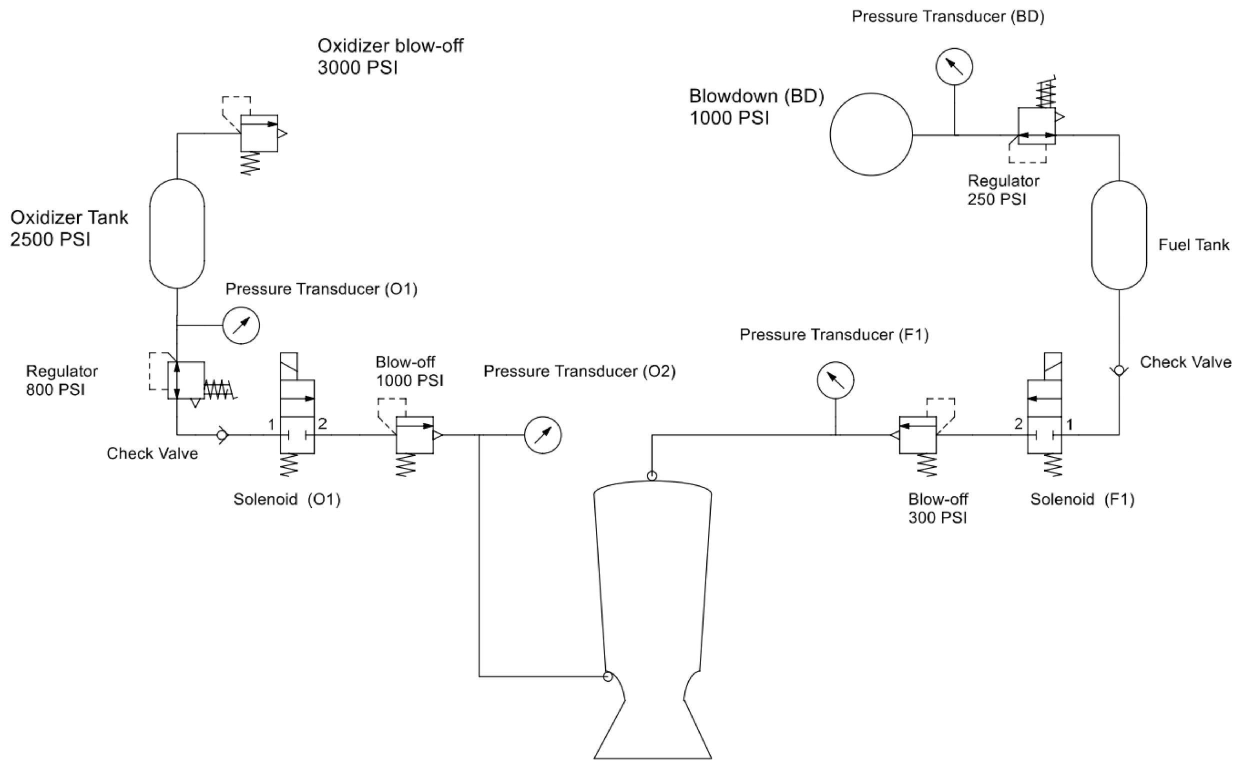
Ground Systems Control Box

Future Work

- Model combustion process in ANSYS Fluent to see if CFD can aid in the optimization of the engine components and vortex creation.
- Run hot fire tests to collect temperature data from combustion process and the vortex cooling methods.
- An analytical model should be created to help predict the boundary layer heat transfer rate. The analytical model will have to be supplemented with experimental data to further optimize the injector design to insure proper vortex generation.

Acknowledgments

We would like to thank the College of Arts and Sciences for providing students the opportunity to conduct undergraduate research in the Engineering Physics Propulsion Laboratory. We would also like to thank Dr. Sergey Drakunov for his mentorship.



Plumbing and Instrumentation Diagram