MECH 410 Project Report

UVic Rocketry Hybrid Rocket Injectors: A Computational Fluid Analysis

Aidan Sarkozy - V00937139
Colm Molder - V00937879
Blaine Tubungbanua - V00918128

Dec 6th, 2023

Abstract

Working for the University of Victoria's Rocketry Team, this project focused on optimizing hybrid rocket injector orifice geometry using Ansys Fluent. Despite limitations in simulating nitrous oxide phase change, the study identified Coanda effects impacting flow distribution. Initial results guided optimization efforts, determining ideal orifice parameters and investigating impinging orifices and swirl angles. The ideal geometry was determined to be a hole diameter of 1.9mm, combined with a 15-degree impinging angle and a 22.5-degree swirl angle. Technical challenges in meshing and software navigation were overcome, offering suggestions for future work with alternate solvers, such as ANSYS CFX, to improve phase change modelling and enhance simulation accuracy. The outcome of the report provides valuable insights and recommendations for refining oxidizer injector designs for UVic Rocketry's Ramses-1 project.

Table of Contents

List of Tables 3
1 Introduction 4
1.1 Background 4
1.2 Problem Definition 4
1.3 Project Scope 4
1.4 Theory 5
2 Methods 6
2.1 Software Selection 6
2.2 Simulation Set Up 6
3 Discussion and Results 8
3.1 Initial Results 8
3.2 Optimization 10
3.3 Simulation Verification 12
3.3 Recommendation 14
4 Project Reflection 15
4.1 Technical Challenges 16
4.1.1 Navigating ANSYS 16
4.1.2 Meshing 16
4.2 Experience and Suggestions 16
4.3 Group and Individual Efforts 16
4.4 Future Work 17
5 Conclusion 18
References 19

List of Figures

Figure 1: Mule-1 Hybrid Rocket Engine 4
Figure 2: Schematic of orifice flow velocity problem 5
Figure 3: Coanda Effect [3] 6
Figure 4: 3D and 2D Models 7
Figure 5: 3D and 2D Mesh 7
Figure 6: Simulation Boundary Conditions 7
Figure 7: Mule-1 Velocity Simulation over 25ms 8
Figure 8: Mule-1 Velocity Simulation Steady State 8
Figure 9: Mule-1 Volume Fraction Simulation over 25ms 9
Figure 10: Mule-1 Volume Fraction Simulation Steady State 9
Figure 11: Stream Lines During Early Time Steps 9
Figure 12: Mule 1 3D results after 25ms -Volume Fraction Left -StreamLines Right 10
Figure 13: Ansys Workbench Parameter Workflow 10
Figure 14: Orifice Geometry Optimization 11
Figure 15: frame-by-frame flow oscillation at critical spacing 12
Figure 16: Impinging orifices. 12
Figure 17: Experimental Discharge coefficient vs pressure drop [5] 13
Figure 18: Experimental N2O injector test 14
Figure 19: Full 3D model of combustion chamber with optimal orifice geometry 15
Figure 20: Close-up image of optimal 3D injector geometry with meshing 15
Figure 21: Swirl Injectors - NO2 Volume Fraction over 25ms 15
Figure 22: Swirl Injectors - Streamlines 16

List of Tables

Table 1: Simulation discharge coefficients for velocity and mass flow 14

1 Introduction

The following sections will discuss the project's background, problem definition, and relevant theory.

1.1 Background

The University of Victoria's Rocketry team (UVR) is attempting to build a new hybrid rocket engine called Ramses-1. This rocket will be the successor to the current test build known as Mule-1. A hybrid rocket engine is propelled by the reaction of fuel and oxidizer, one in a solid phase, and one in a fluid phase [2]. Ramses-1 and Mule-1 make use of a cylindrical solid paraffin wax fuel with a hollow center to allow a vapourized nitrous oxide (N2O) oxidizer to interact with the fuel grain. The N2O is stored as a liquid in a tank, and when propulsion is desired, is released. The N2O then travels through small orifices which atomize the liquid, aiding its transition to a vapour phase. After the injection orifices, is a chamber called the pre-combustion chamber, which allows the N2O to vapourize before interacting with the fuel grain, located after the pre-combustion chamber. This layout is shown below.

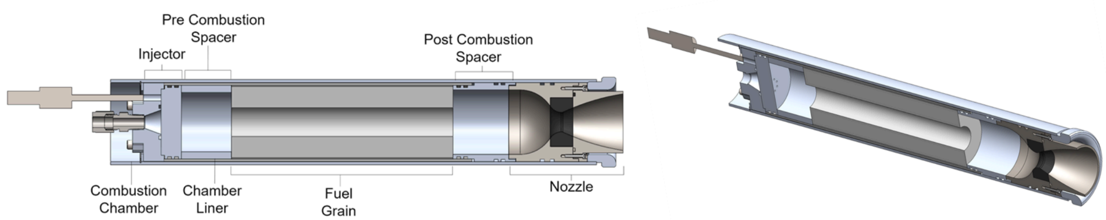

Figure 1: Mule-1 Hybrid Rocket Engine

1.2 Problem Definition

The goal of this project is to optimize the injector orifice geometry. Ideal orifice geometry successfully atomizes the liquid nitrous oxide and provides an even spread along the solid fuel grain. These factors are important to ensure stable combustion at a desired oxidizer-fuel ratio and fuel grain regression rate. Orifice geometry will be optimized by determining the ideal:

1. Orifice Length
2. Orifice Diameter
3. Orifice Angle
4. Orifice Swirl Angle

1.3 Project Scope

This project will aim to develop a model of N2O injection into the pre-described geometry combustion chamber. Due to limitations of Ansys Fluent, no atomization or phase change of N2O will be simulated. Optimization attempts will only focus on the injector geometry and only analyze a single borehole diameter consisting of eight orifices. The fluent simulations will be used to observe the effect of various geometry on the spread of gaseous N2O about the fuel grain while ignoring combustion effects that will eventually result from the interaction. This assumption is reasonable as simulations will only analyze the initial injection instances before combustion begins.

1.4 Theory

The main driving factor that facilitates oxidizer mass flow is the pressurization of the N2O. By pressurizing the N2O upstream, the pressure reduction into the combustion chamber will cause N2O to flow into the combustion chamber. The magnitude of this mass flow is dependent on the cross sectional area of the orifice as well as the magnitude of the pressure drop from the N2O tank to the combustion chamber, while also accounting for pressure losses in the feed lines from the N2O tank to the combustion chamber, resulting in a pressure drop of $\Delta P = P_{pre-inj} - P_{CC}$, where the pressure drop is measured across the pre-injector and combustion chamber. A schematic of this can be seen in Figure 2:

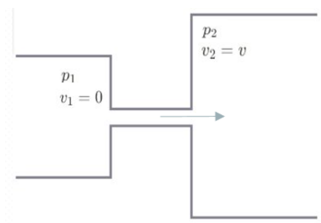

Figure 2: Schematic of orifice flow velocity and pressure drop

A common initial approximation for liquid injector mass flows is the Bernoulli equation. Assuming the initial velocity is zero, the outlet velocity can be obtained for the given pressure drop ΔP, and the resulting mass flow $\dot{m}$ can be determined for a given orifice area A.

$$p_1 + \frac{1}{2}\rho v_1^2 + \rho g h = p_2 + \frac{1}{2}\rho v_2^2 + \rho g h$$

$$v = \sqrt{\frac{2\Delta p}{\rho}}$$

$$\dot{m} = vA\rho = \rho Q = A\sqrt{2\rho\Delta P}$$

In reality, there are losses to the system, and the actual mass flow is reduced by wall-boundary interactions, as well as inefficient momentum changes at the orifice inlet. These losses are characterized by the discharge coefficient C_d.

$$C_d = \frac{\dot{m}_{actual}}{\dot{m}_{theoretical}}$$

A rocket injector could be a single large hole which would minimize the wall-boundary losses, but this would result in poor atomization of the N2O, yielding poor mixing with the fuel grain.

As will be seen later in this report, several notable fluid motion phenomena were observed in simulations. Fluid jets injected into an ambient environment are observed to be pulled toward a surface due to a process known as the Coanda effect. The Coanda effect results from entrainment of the ambient fluid into the injected fluids flow causing a velocity and pressure gradient. This low pressure surrounding the flow is filled by ambient fluid, however, if a surface is present then the flow itself will be attracted to the surface. The flow will then remain on the surface due to the ambient pressure [3]. As seen below, this effect can trap an eddy of ambient fluid above the flow.

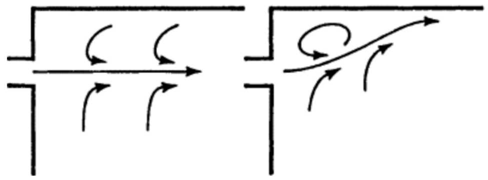

Figure 3: Coanda Effect [3]

2 Methods

The following sections will overview the software chosen for this task and the procedure of setting up simulations.

2.1 Software Selection

Due to the desired simulation involving complex fluid mechanics such as multiple phases, atomization, and fast dynamics, Ansys fluent was recommended by Dr. Dong as a much more capable CFD software than Siemens NX. Using the Ansys Workbench environment would also allow for efficient alterations in design, mesh, and simulation settings allowing for quick changes of parameters. The 3D fluid volume model that will be seen in the following section was designed in SolidWorks due to team familiarity and integration with UVR's current CAD standards. For 2D geometry, to allow faster alterations, Ansys Design Modeler was used within the Workbench environment.

2.2 Simulation Set Up

Both 2D and 3D parametric models were created for this project. 2D models will be used to quickly alter geometry and simulate results while 3D models will be used to verify the 2D results and test the final recommended configuration. The 3D model made use of symmetry to cut the model in half and simulate four of eight orifices while the 2D model was a slice modeling two orifices.

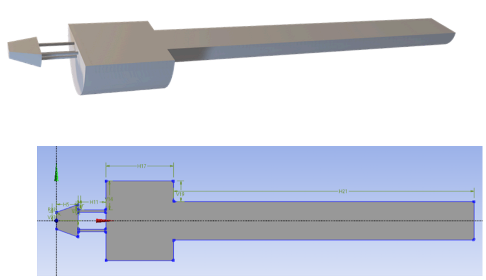

Figure 4: 3D and 2D Models

Determining the mesh for both 3D and 2D meshes was found to be critical to ensure the success of simulations. The 2D mesh made use of an all-triangle method, as this proved to provide the fastest meshing, important for quick geometry alteration. The 3D mesh used a Quadratic element method to provide more accurate final results. Both meshes used mesh refinement on the small diameter orifices as well as a growth rate of 1.2, high smoothing, and smooth inflation. These settings ensured the element size transition from the orifices was gradual, essential to ensure no floating point errors occurred during simulation. The 2D mesh used a max size of 1mm, while the 3D made use of 5mm. Variations in mesh size were attempted, but few effects on simulation results were noted.

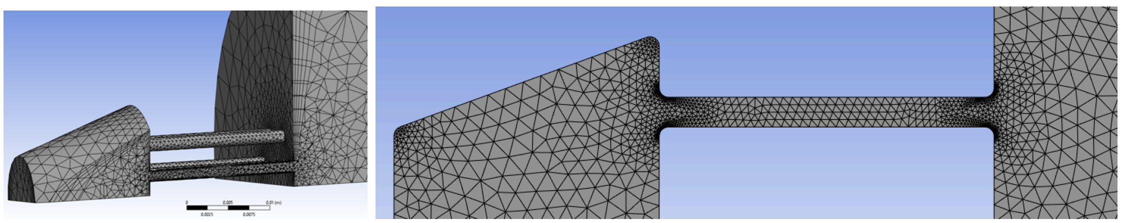

Figure 5: 3D and 2D Mesh

To study the spread of N2O and its interaction with the ambient air, a volume of fluids multiple-phase model was used in Fluent. To facilitate this mode, the model was pressure-based and a transient study was used to observe the changes over time. The boundary conditions of this simulation are seen below. The N2O inlet was set to the provided pressure of the tank and a temperature of saturated liquid N2O. To further increase simulation accuracy, viscous K-epsilon equations were used with a coupled solution method and all residuals were set to 1e-6. The simulation initialization was set to calculate from all zones with ambient starting conditions. Initialization was found to be important to capture initial injection dynamics.

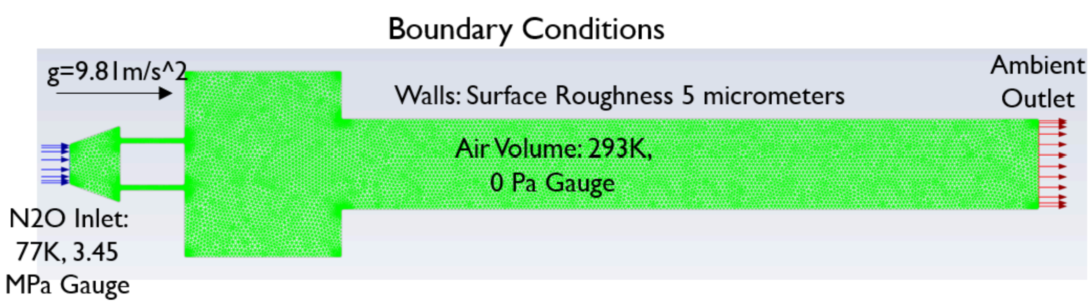

Figure 6: Simulation Boundary Conditions

3 Discussion and Results

The following sections will outline the initial results on the original Mule-1 geometry, then overview the optimization steps taken.

3.1 Initial Results

The initial velocity results on the Mule-1 geometry are seen below over the first 25ms. Steady-state results are seen in Figure 8.

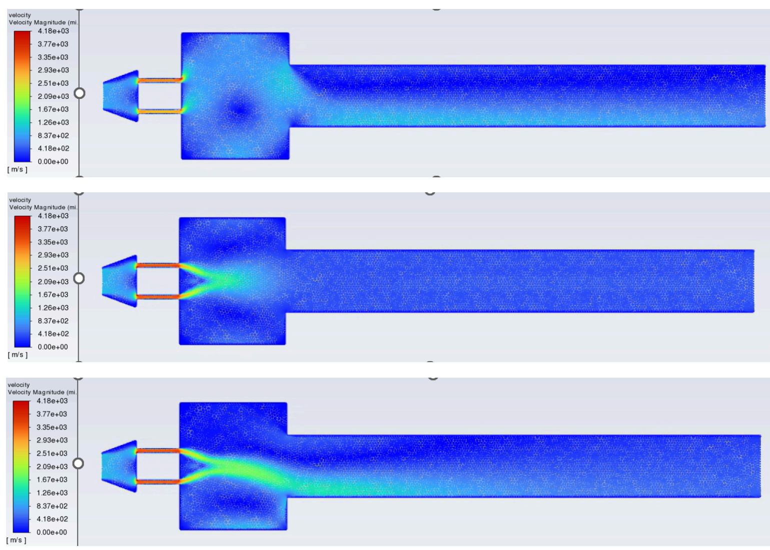

Figure 7: Mule-1 Velocity Simulation over 25ms

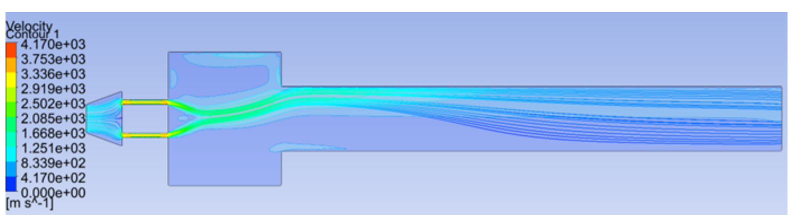

Figure 8: Mule-1 Velocity Simulation Steady State

The Coanda effect can be seen in the injected N2O flow being pulled to one side of the fuel grain. The side the injection tended to flow to would change depending on mesh or simulation settings, and was likely due to small asymmetries in the mesh. In reality, the direction the flow would tend to would result from manufacturing tolerances or other disturbances. Similar effects to Coanda can describe why the two high-velocity flows combine. As air between the streams is entrained, an area of low pressure is created, pulling the two streams together. The Coanda effect is not ideal, as the N2O is mostly being delivered to one side of the fuel grain.

As seen below in Figure 9, the volume fraction of N2O (where red is 100% N2O) is seen over the first 25ms with the steady state shown in Figure 10.

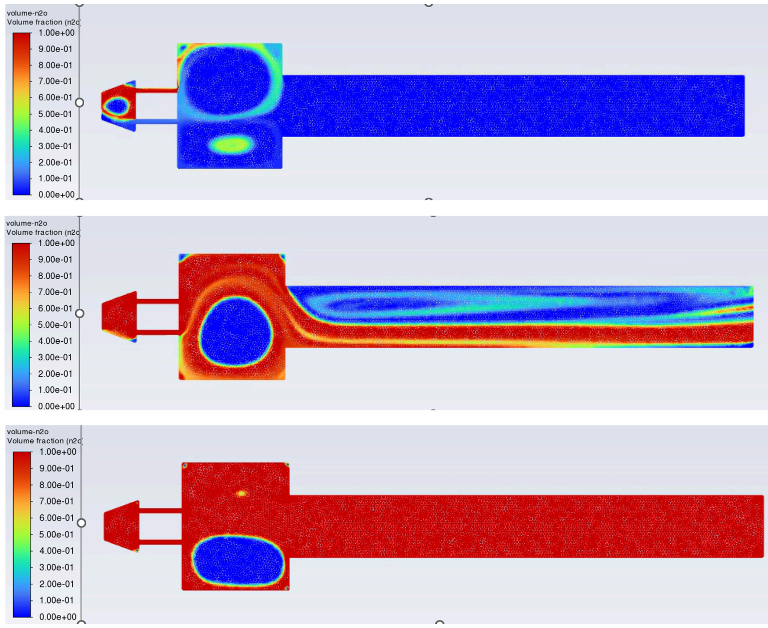

Figure 9: Mule-1 Volume Fraction Simulation over 25ms

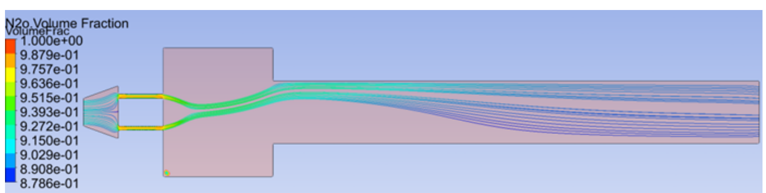

Figure 10: Mule-1 Volume Fraction Simulation Steady State

The Coanda effect is seen to trap an eddy of air in the pre-combustion chamber. This result was anticipated as seen previously in Figure 3. The air eddy can be seen forming in the streamline graphic below, taken during early time steps.

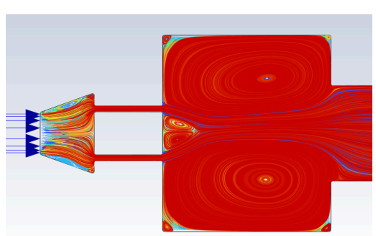

Figure 11: Stream Lines During Early Time Steps

The effect of an air eddy being trapped in the pre-combustion chamber is again not ideal. As seen in Figure 10, at steady-state the air is eventually released, however, this occurs slowly. As the air is released, it disrupts the N2O's interaction with the solid fuel grain.

To verify 2D results, a 3D simulation was run. As seen below, the Coanda effect is again present and traps an air eddy. This figure also shows the air beginning to escape, and interrupting the N2O interaction with the solid fuel grain.

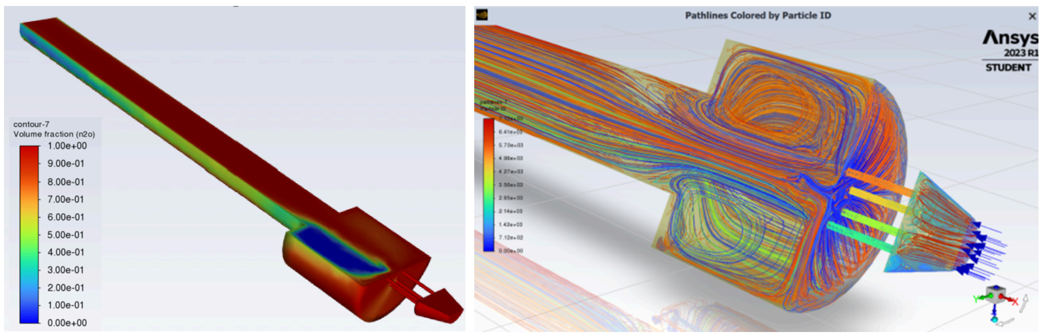

Figure 12: Mule 1 3D results after 25ms -Volume Fraction Left -StreamLines Right

The maximum velocity recorded in the 3D simulation was 2.68e03 m/s compared to the maximum velocity of 4.17e03 m/s seen in the 2D simulation. This discrepancy highlights the inaccuracies in 2D simulations as they don't account for turbulence and other fluid dynamics in all three dimensions.

3.2 Optimization

After completing the initial model, optimization of the orifice diameter, length, angle, and the potential for a swirl were investigated. To accelerate simulation time, a simplified 2D single orifice model was used. The simulation ran for 0.5 seconds, and the velocity plots were analyzed. Using the parameter set tools in Ansys workbench, an open-loop parameter optimization was conducted (Figure 13)

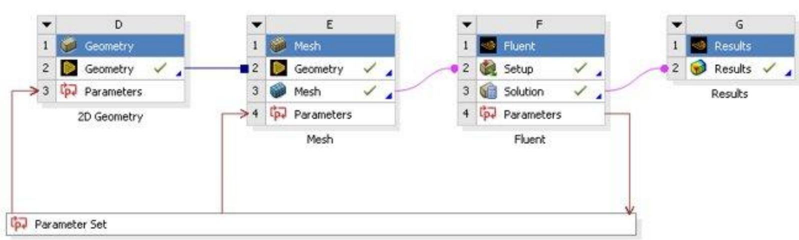

Figure 13: Ansys Workbench Parameter Workflow

Due to simulation issues discussed in section 3.3, formal quantitative conclusions could not be reached, and the following optimizations were based on qualitative observations of the N2O flow through the combustion chamber.

The first optimization was orifice length and diameter. A L/D ratio of 5 was selected based on literature, and different orifice diameters were tested to maintain this constraint (Figure 14). Examining the results, the 1.9mm variation was selected for its downstream flow characteristics. This was selected to move the tests forward but must be revisited once the model is improved.

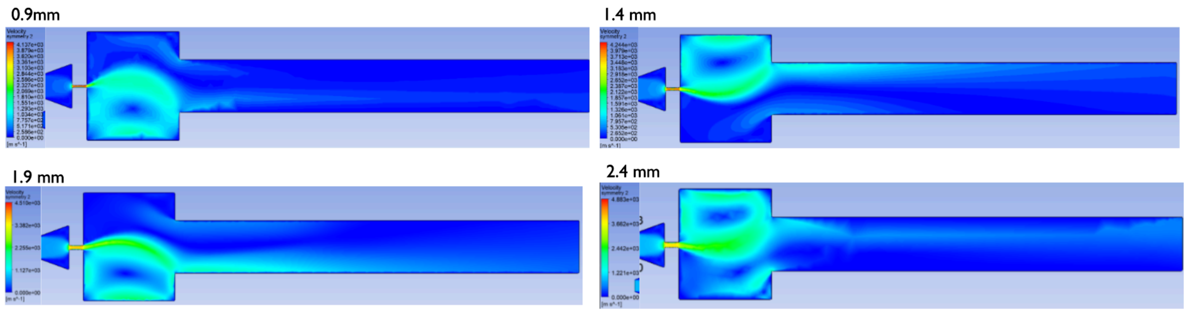

Figure 14: Orifice Geometry Optimization

The effect of angled orifices was also investigated, specifically a 15-degree hollow cone configuration, according to values in literature [4]. With this configuration, the spacing between the orifices was investigated. With the orifices close together, spaced 5mm apart, the flows converged, as expected. At 10mm apart, the flows oscillated (Figure 15), as the two flows fought to entrain each other, and at 20mm, the flows separated completely. These oscillations are undesirable and should raise concerns for the rocketry team due to their potential to cause variations in pressure. If these pressure oscillations escalate significantly, they could lead to the failure of the pressure vessel, posing a safety risk. Additionally, such oscillations can cause fluctuations in mass flow, ultimately impacting the injector's ability to deliver the intended oxidizer-fuel ratio, thereby reducing the overall performance of the system.

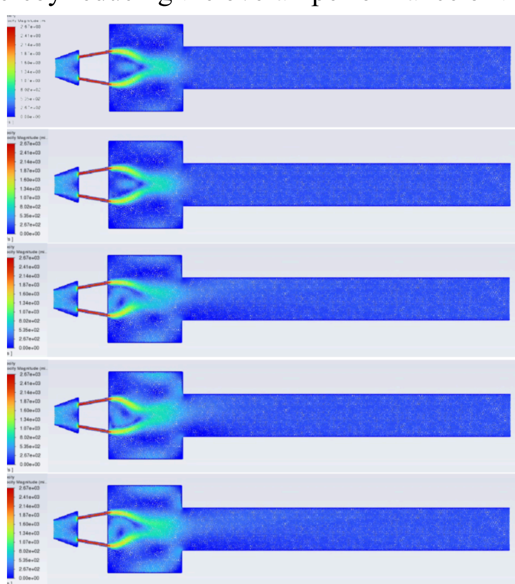

Figure 15: Flow oscillation at critical spacing over 30ms

Impinging orifices were also examined (Figure 16), as the literature says it improves particle breakup as streams collide [4]. This is a common configuration in industry, but due to its manufacturing difficulty, the benefit must still be weighed. Unfortunately, the current model doesn't give insight into this phenomenon. The inclusion of discrete phase modelling (DPM) during simulation is left as future work, which would allow tracking of individual particles, and particle size.

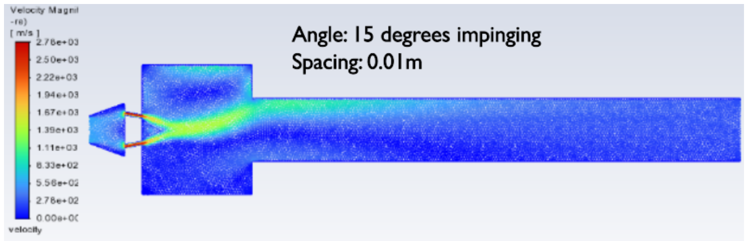

Figure 16: Impinging orifices.

The effect of a swirl angle was also investigated, for its potential to combat the Coanda effect, improve distribution along the fuel grain, and increase fuel grain regression rates [5]. This was not possible in the 2D simulations and was attempted in the 3D simulation discussed in section 3.4.

3.3 Simulation Verification

To verify the simulation, the original goal was to compare the simulation C_d with values found in literature (Figure 17). Solving the expected mass flows yielded the results in Table 1. Due to limitations of Ansys Fluent's modelling capabilities, unreasonable results were obtained. Ansys Fluent was unable to simulate the phase change of N2O from liquid to gas. Instead, the model assumes N2O as gas for the entire simulation, despite being pressurized above its saturation pressure. Liquid-vapour phase changes are available for other materials like diesel, but for N2O, modelling the phase change would require inputting custom user models. This leads to an increased mass flow due to the gaseous properties of N2O when pressurized beyond its saturation point, causing a volume that is unrealistically smaller than the incompressible liquid volume. While comparing the velocity results, which are somewhat less dependent on density, they exhibit closer alignment. However, this discrepancy results in a discharge coefficient greater than 1, which is unrealistic. Addressing this issue presents an opportunity for future work, potentially utilizing Ansys CFX, which is better equipped to handle phase changes, boiling, and cavitation in simulations.[7].

Table 1: Simulation discharge coefficients for velocity and mass flow

2D Model Results	
Pre-injector pressure Post-injector pressure Pressure Drop	$P1 = 3.21\ kPa$ $P2 = -1.49kPa$ $\Delta P = 4.69kPa$
Theoretical Massflow	$\dot{m} = 0.00663\ kg/s$

Simulation Massflow Discharge Coefficient	$\dot{m} = 7.0427\ kg/s$ C_d =1062
Theoretical Velocity Simulation Velocity Discharge Coefficient	$v = 2179.24\ m/s$ $v = 3763\ m/s$ $C_d = 1.73$

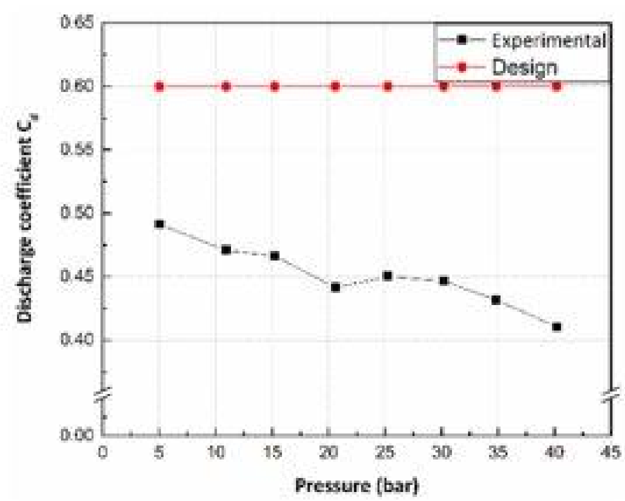

Figure 17: Experimental Discharge coefficient vs pressure drop [5]

Despite these challenges, the model remains valuable for simulating the behaviour of N2O as it travels through the combustion chamber, under the assumption that the oxidizer exits the orifices in a vapour state. A comparison between this model and an experimental injector test detailed in literature (Figure 18) revealed a similarity in behaviour. The model closely mirrored the experimental setup, demonstrating the occurrence of the Coanda effect observed in physical rocket injector experiments.

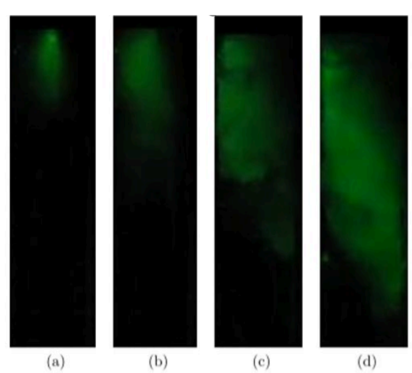

Figure B.8: Injector (5) images at (a) $\Delta P = 5$ psi (0.03 MPa), (b) $\Delta P = 50$ psi (0.34 MPa), (c) $\Delta P = 100$ psi (0.69 MPa), and (d) $\Delta P = 200$ psi (1.38 MPa).

Figure 18: Experimental N2O injector test

3.3 Recommendation

Based on the findings from the 2D analyses, it was established that an orifice diameter of 1.9mm, combined with a 15-degree impinging angle, would provide optimal flow dispersion of the oxidizer within the combustion chamber.

Subsequently, this specific geometry was incorporated into a 3D model of the injectors to gain a comprehensive understanding of how the different streams would interact with one another. A swirl angle was also included in the model as existing research suggests its potential to enhance combustion stability and distribution along the fuel grain [5]. The full 3D model which includes this orifice geometry appears in Figure 19, and a zoomed-in image focusing on the orifices with the generated mesh is shown in Figure 20.

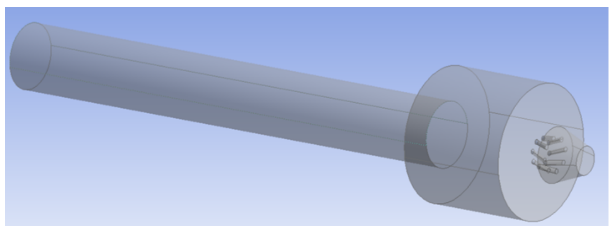

Figure 19: Full 3D model of combustion chamber with optimal orifice geometry

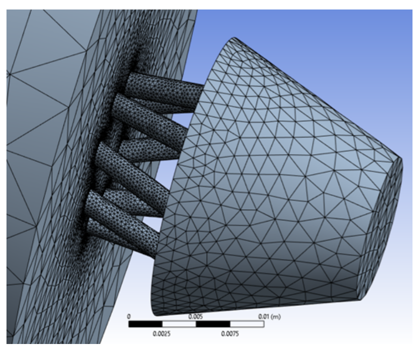

Figure 20: Close-up image of optimal 3D injector geometry with meshing

Through the same simulation settings and inputs used in the 2D models, an animated representation of the flow pattern was generated. This animation is shown over 4 images in Figure 21 and illustrates the pressurization of the injector inlet upon the introduction of nitrous oxide into the space. Subsequently, the gas is expelled from the injectors, evenly distributing along the fuel grain's length.

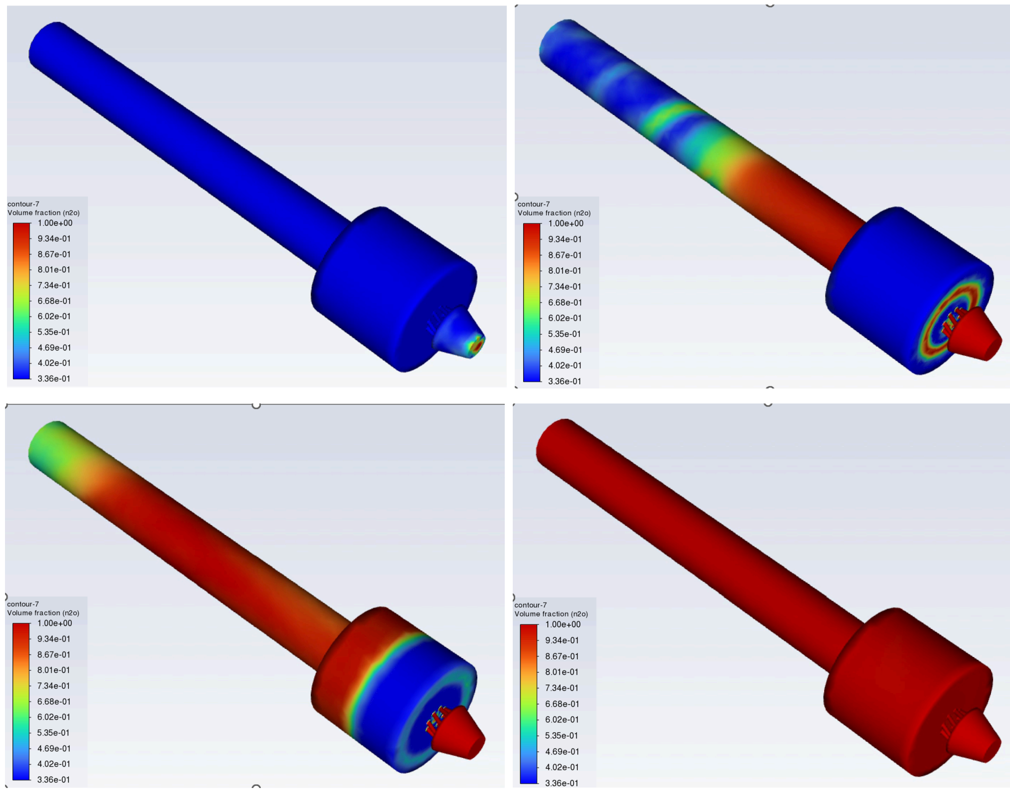

Figure 21: Swirl Injectors - NO2 Volume Fraction over 25ms

Finally, a streamline plot was implemented to showcase the vortex flow pattern of the injected oxidizer. Figure 22 showcases this phenomenon and it should be noted that this pattern effectively counteracts the Coanda effect, resulting in a notably uniform distribution of the oxidizer within the chamber. Additionally, there is no air eddy trapped in the pre-combustion chamber as there was in figure 9.

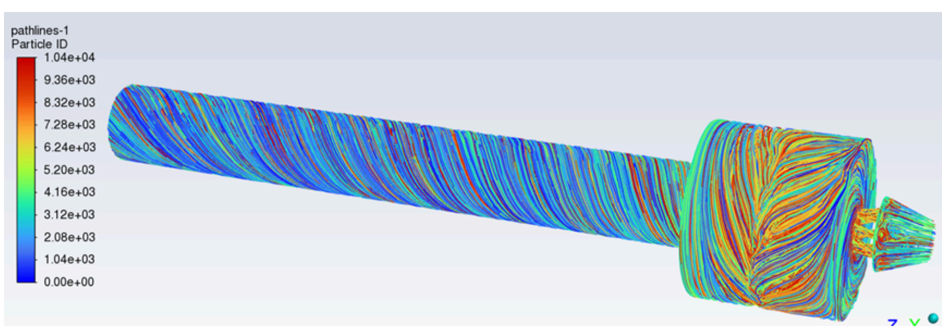

Figure 22: Swirl Injectors - Streamlines

4 Project Reflection

The following sections will give an overview of the problems overcome and the lessons learned while completing this project. A breakdown of how the workload was shared is also detailed.

4.1 Technical Challenges

There were significant issues that the group faced in the process of determining the optimal orifice geometry of a hybrid rocket engine.

4.1.1 Navigating ANSYS

The first issue that arose when presented with learning a new software such as ANSYS was gaining familiarity with the menus and settings. It took many YouTube tutorials and time spent reading the ANSYS manuals to know where specific tools were and how to use them in order to achieve the project goals. When there weren't any online resources available to assist with specific questions, trial and error was used as well as consulting with the course TA's. This was seen when it was determined that a multi-phase model should be used, but the type of the multi-phase model was not obvious. It was only after trying all the subtypes of multiphase models, and getting inconsistent results, that it was decided to use the volume of fluid method.

4.1.2 Meshing

Obtaining a high-quality and usable mesh was likely the most difficult and time-consuming challenge the group faced. Many days were spent running CFD simulations that failed and threw a "floating point exception". It was known that this error occurred due to an improper mesh, but it took many alterations to the mesh size and settings to determine that mesh inflation was the direct culprit. The mesh size was growing too quickly as the geometry changed from the orifices to the pre-combustion chamber. Once a constraint was set for the inflation, the error was resolved and the simulations could be run.

4.2 Experience and Suggestions

In the future, it would be recommended to have a very concrete understanding of the system that is meant to be simulated before a model is made. The output of a CFD analysis is the product of its inputs. If a simulation is improperly set up, the output will be inaccurate and lead to false conclusions. Knowledge of the system and the use of hand calculations allows for the simulation to be verified, which is important if any conclusions are to be drawn or design choices are made based on the output of a CFD analysis. In the case of this project, it would have been useful to verify that the model used to simulate flow through the injectors was able to handle a phase change. It was discovered that the volume of fluid model could not process phase changes as determined by the mass flow comparison of the CFD to the hand calculations. While having enough understanding of the system to recognize this error was important, setting up the correct model initially would have saved a lot of time and led to more accurate conclusions.

4.3 Group and Individual Efforts

The following section will overview the main contributions of each team member.

Colm was responsible for the determination of ideal mesh and Fluent settings used for simulations. Colm performed in-depth research and practice with the Ansys environment including WorkBench, DesignModeler, SpaceClaim, and Fluent. Once ideal settings were determined, Colm ran initial simulations and determined the best post-processing graphics and animations to analyze to gauge results. Colm also worked with Aidan to create 2D parametric models used to examine the effect of angled injectors.

Blaine managed the project, determining the relevant parameters to focus on, conducting research for the project gathering the theory required to integrate the injector with the rest of the engine, as well as preliminary hand calculations to determine initial values for the injector to iterate on, as well as validation of the simulation. Blaine also ran 2D 2-orifice angle optimization simulations investigating the effect of angled orifices with parameterized models, using the meshing and simulation settings optimized by Colm.

Aidan was responsible for the creation of the 3D models utilized throughout this project. The initial model was based on Mule-1's geometry and designated as a baseline to compare further simulations. Aidan also created 2D models of the rocket combustion chamber with a single orifice and ran ANSYS Fluent simulations to determine the optimal geometry for the orifice. The simulation settings were based on the settings Colm discovered while running previous CFD analyses.

4.4 Future Work

Major discrepancies between the theoretically calculated mass flow and the CFD mass flow result from the model's inability to model the phase change of N2O without custom user-inputted fluid models. Creating these models, or using alternate CFD solvers such as ANSYS CFX or employing new models in ANSYS Fluent like DPM could result in more accurate results.

One potential avenue for improvement involves enhancing the phase change and cavitation analysis capabilities by employing ANSYS CFX, which would allow for improved simulation of the phase change as well as the resulting boiling and cavitation that occurs throughout the orifice [7], improving the accuracy of the interactions between the N2O and the orifice. Furthermore, applying discrete phase modelling (DPM) in Fluent would allow for insights into the effects of impinging flow configuration, and particle flow out of the orifices.

Additionally, the current model could be improved by incorporating a diffuser at the end of the fuel grain to better model the increase of pressure in the combustion chamber. This modification would better replicate real-world conditions and refine the accuracy of the simulation, capturing the effects of the pressure increase as the gas enters the combustion chamber.

5 Conclusion

This project focused on optimizing hybrid rocket injector orifice geometry for UVic Rocketry's Ramses-1 project through computational fluid analysis using Ansys Fluent. Despite limitations in simulating N2O phase change, the study identified Coanda effects impacting flow distribution. The project scoped the ideal orifice parameters and investigated impinging orifices and swirl angles, determining an optimal geometry of a 1.9mm hole diameter, a 15-degree impinging angle, and a 22.5-degree swirl angle.

Future work was recommended, emphasizing the need for improved phase change modelling, exploring alternative solvers, employing discrete phase modelling (DPM), incorporating diffusers, and refining simulation setups. These efforts aim to enhance the accuracy of injector geometry recommendations for UVic Rocketry's Ramses-1 project.

Ultimately this report provides an overview of CFD's implementation in rocket injectors and highlights the potential to further develop more robust models using alternative tools. Initial simulations provide a qualitative analysis of injector flow through a cylindrical body and present UVic's Rocketry Team with a starting point for which to test their latest rocket engine.

References

[1] University of Victoria, 'MECH 410', Zuomin Dong, Siyang Steven Liu, Chon Him Lawrence Wong

[2] 'Hybrid Rocket Propulsion - Propulsion 2 - Aerospace Notes'. Accessed: Dec. 04, 2023. [Online]. Available: https://aerospacenotes.com/propulsion-2/hybrid-rocket-propulsion/

[3] 'Coanda Effect - an overview | ScienceDirect Topics'. Accessed: Dec. 04, 2023. [Online]. Available: https://www.sciencedirect.com/topics/earth-and-planetary-sciences/coanda-effect

[4] B. Waxman, "An Investigation of Injectors for Use with High Vapor Pressure Propellants with Applications to Hybrid," Stanford University June 2014,

[5] M. Bouziane, A. E. M Bertoldi, D. Lee, P. Milova, P. Hendrick, M. Lefebvre "Design and Experimental Evaluation of Liquid Oxidizer Injection System for Hybrid Rocket Motors", in the 7th European Conference for Aeronautics and Space Sciences, 2017.

[6] S. Alam, 'ANSYS CFX: Everything to Know | Explore the Future of Engineering: 3D Modeling, CAD and More', Sunglass. Accessed: Dec. 04, 2023. [Online]. Available: https://sunglass.io/ansys-cfx/

[7] M. Passarelli, Launch Canada Community Forum, University of Toronto, Jan 23, 2021 (Registration required to view)