MAE 185 Final Project Report

Janna Aboudaher | Joshua Cantrell | Christian Parra

In this report, air flow simulations for the vortex cooled rocket engine were performed, optimized, and analyzed. The vortex cooled rocket engine, Maelstrom, was a senior design project conducted by students at UC San Diego to determine the feasibility of a vortex cooled engine to adequately cool the engine walls while successfully mixing the fuel and oxidizer. The airflow simulations served as a verification that the engine geometry would perform as desired and prompted hands-on testing. In this project, simplified airflow simulations were conducted on the maelstrom engine geometry with slight modifications to improve performance. Vortex formation and wall cooling were studied via these simulations similar to the original Maelstrom team's simulation goals. A grid independence study was conducted to determine the optimal grid size for the fixed boundary conditions to gain physical results of a steady state simulation.

In this simulation, the pressure based solver was utilized as the flow is subsonic air and can be treated as an ideal gas and generally incompressible. The energy model is selected as "on" due to the selection of ideal gas for air density and the Sutherland model for air viscosity. The selection of ideal gas and the Sutherland viscosity are different from the selections made by the original Maelstrom team, and provide more physical velocity and temperature results that will be discussed in the results section. The realizable k-ε turbulence model was chosen based on research done by the original Maelstrom team. It was found that realizable k-ε is better suited for flows with vortices, rotation, and high turbulence as well as disregarding unphysical values for k and ε. This model was compared against k-ω and the team found more physical and consistent results using realizable k-ε.

The geometry was slightly modified to accommodate a larger plenum. To better simulate fairfield conditions, the plenum geometry was increased to have a diameter and length 20x that of the engine nozzle exit. This is a substantial increase in size from the given Maelstrom geometry, and allowed for better simulation of the exhaust jet. However, the larger plenum did pose interesting challenges for the mesh sizing, as high computational expense goes into fine meshes over large areas. To remedy this, local sizing was implemented for both the engine and plenum to allow for fine spatial resolution in the engine, specifically at the injectors and nozzle throat, as well as lower resolution throughout the plenum. Additionally, to better simulate the fuel (top) injector flow, the shower head geometry was chopped to avoid the difficult simulation of the right angle within the shower head manifold. It can be seen in the velocity streamline images that the fuel injector geometry was reduced to just the eight inlet holes to the chamber. This is sufficient for evaluating the vortex formation and thermal insulation of the walls. Should one require a simulation of this flow, isolating this geometry and running a more specific simulation would be more computationally efficient.

The boundary conditions selected for this simulation are based on those collected by the Maelstrom team through a prototype test. Different boundary condition types were tested, and velocity inlets / a pressure outlet were determined to provide the most physical and converged simulation. The velocity inlet boundary conditions were determined by the original Maelstrom team using a 3D printed fuel injector geometry, an air hose, and an anemometer to determine the velocity exiting the fuel injector holes. This value was set at 12 m/s, and due to the improvement to the viscosity model, was bumped to 15 m/s for this simulation. Additionally, the fuel inlet is set to 1500 K to simulate a hot central combustion column, requiring the wall to be insulated via the oxidizer vortex. The oxidizer injector velocity was determined by trial and error knowing its velocity must be less than that of the fuel injection for strong central column formation. This value was set to 8 m/s as this allowed for strong vortex formation without disrupting the central column. Hybrid initialization was selected and generally converged during initialization.

Grid study

	Level 1 - Coarse	Level 2 - Medium	Level 3 - Fine
Mesh			
Plenum LocalSizing	25	12.5	6.25
Engine LocalSize	4	2	1
Surface Mesh Size (Min/Max)	(0.5 / 50)	(0.25 / 25)	(0.125 / 12.5)
Boundary Layers	3	6	12

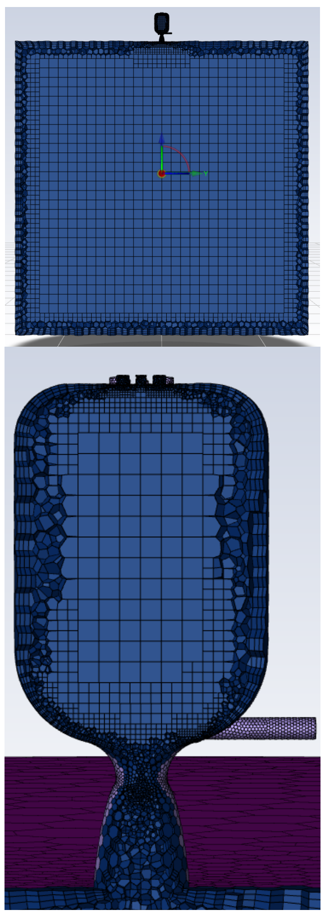
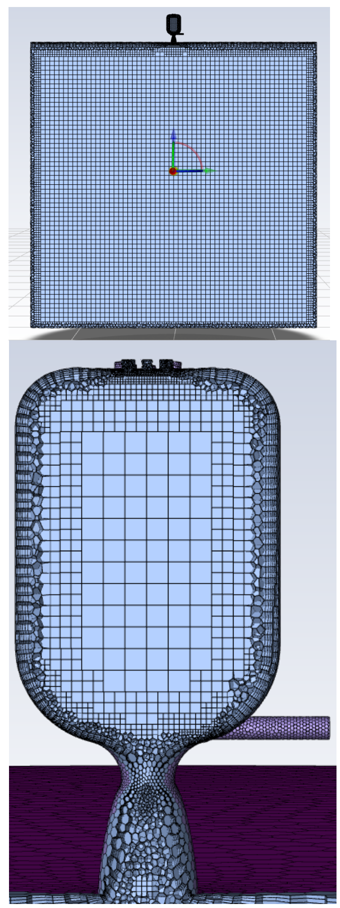
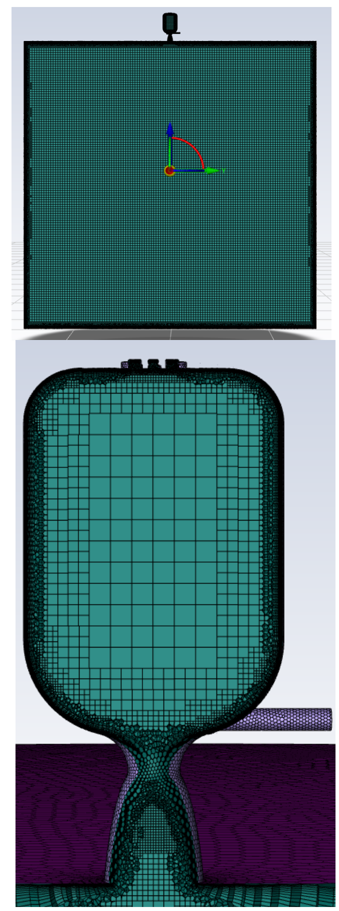
Maximum Cell Length	32	16	8
Number of cells	170,850	578,888	4,529,206
Maximum skewness	0.5615	0.5172	0.4847
Minimum orthogonality	0.2241	0.1566	0.0606
Screenshot of mesh			

Table 1: Parameters required to construct each mesh and their screenshots

A three level grid study was conducted at course, medium, and fine mesh configurations. The parameters of these mesh levels are presented in Table 1. All sizing quantities were halved for each progressing refinement level, while the amount of boundary layers was doubled after each refinement level. Local sizing was applied to both the plenum and engine, with target size of the plenum being much larger than that of the engine. This was done for two reasons. First because the results of the engine flow are the primary focus of the study and while the plenum is present in order to properly apply exit boundary conditions. Second, the engine geometry is complex and involves many small curvatures, requiring finer detail in order to accurately model flow behavior.

The mesh quality for each refinement level was assessed using minimum orthogonality and maximum skewness. The maximum skewness decreased as refinement level increased, indicating that the cell shape became more regular and closer to ideal. The minimum orthogonality also decreased with finer refinement levels, reaching a minimum value of .0606 at for the fine mesh. This behavior is not desired as the minimum acceptable orthogonality level is .1, with perfect quality being one. This value indicates the amount of cells who have their coordinate axis aligned with that of the system coordinate axis, thus a low value introduces more room for calculation error during the flow simulation. This decreasing trend is not what was anticipated, and can potentially be attributed to complex geometry at the fuel injector and engine nozzle. As anticipated, the amount of cells present in the mesh increased as it became more fine. The amount of cells at the medium level was 3.4 times that of the course, and the fine level 26.5 times that of the course level. Table 1 summarizes these values and includes mesh images of the plenum and a more detailed view of the engine. The image of the coarse mesh includes many irregular cell shapes in the engine cells, while the medium and fine levels show improved cell shape uniformity and more regular cell shapes throughout the entire object.

Based on the mesh qualities discussed above and visual inspection, the medium mesh is most likely to provide accurate results, maximizing geometric detail, maintaining optimal orthogonality and skewness, and minimizing computational cost.

Discussion of Key Results

The primary results the team is looking for are the velocity streamlines of the air inlets as they flow through the engine, and the temperature contour of the two lines; the graphs of which are seen in Table 2. The team observed the formation of a vortex from the air entering through the four oxidiser injectors, with the velocity streamlines forming an upwards vortex. The air from the fuel line remains in a relatively straight path through the engine, which is easily observable from the temperature contour. This coincides with the expected behavior of the engine, where the cool oxidiser wraps around the fuel jet, providing both fuel mixing as well as insulation on the wall of the combustion chamber. It is important to note that there is no combustion occurring in this simulation, as to do so would take months to simulate, due to combustion being extremely complex.

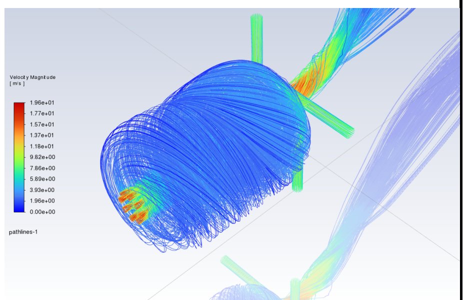
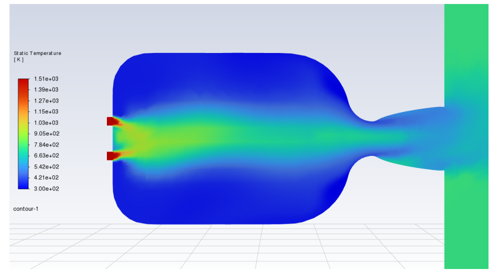
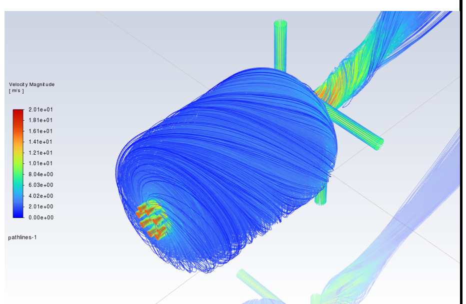
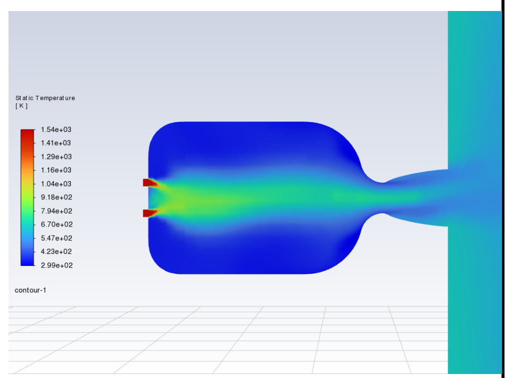
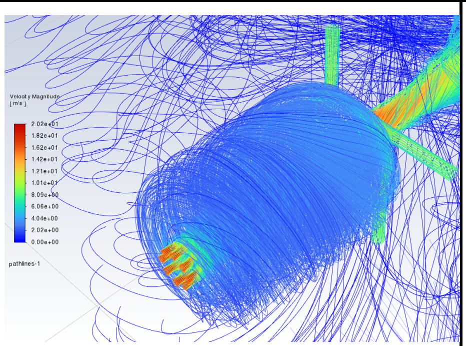
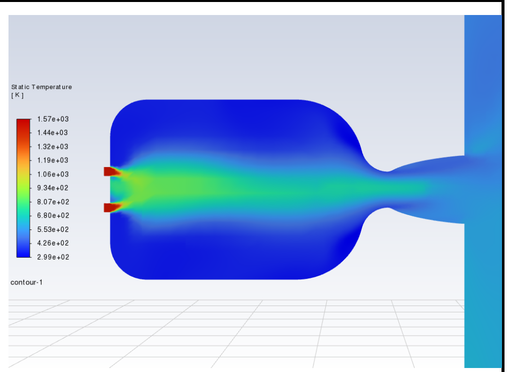
Mesh Level	Simulation runtime	Velocity Streamlines	Temperature Contour
1	126.31s		
2	437.785s		
3	3632.83s		

Table 2: Velocity Streamlines and Temperature Contours of fuel line and oxidiser line of Vapor Cooled Engine, separated by different mesh size levels

The fidelity of the graphs scales with mesh quality. The coarse mesh provides coarse streamlines, as well as abrupt color changes in the temperature contour. This makes sense, as the course mesh had a very fast simulation time, due to the low number of cells to compute through. On the level 2 medium mesh, the velocity streamlines are much more defined, with the formation of the vortex in the combustion chamber looking very clear, and the exit jet stable. The temperature contour is also smoother, especially at the nozzle where the flow exits and turns into the jet. In the level 3 fine mesh, the velocity streamlines behave as expected in the engine, but diverge from the expected in the plenum. The streamlines create a short jet at the nozzle exit, but then diverge towards the plenum wall outlets, creating the erratic image seen in column three of Table 2. While these are slow moving streamlines, the movement is extremely irregular and messy, and can be attributed to low orthogonality and overall mesh quality. In terms of the temperature contour, the fuel temperature is more rigorously defined and is not propagating as far out of the central column as the level 1 and 2 simulations. In terms of mesh quality and the time it took, the level 2 mesh would be a sufficient simulation as the level 3 simulation took 8 times as long as the level 2 one, for providing similar results.

Conclusion

In this study, an airflow simulation was conducted for an airflow prototype of a vortex cooled rocket engine. The motivation of simulating this engine geometry is to understand if this alternative oxidizer inlet orientation is capable of producing a vortex to shield the walls of the combustion chamber from the heat of combustion. The team chose to investigate the vortex formation and thermal insulation properties of the flow via velocity streamlines and temperature contours. Some amendments were made to the initial design, increasing the plenum size to better simulate the fairfield conditions at the exit, and the fuel injector was chopped at the shower head manifold to increase simulation efficiency. The pressure based solver, energy model, realizable k-ε, and sutherland viscosity were chosen as assumptions for this simulation as they are valid for low speed, incompressible flows.

Boundary conditions were selected based on data collected by the Maelstrom team via hands-on testing of the injector plate. Trial and error was used to verify the type of boundary condition for both inlets and outlets. Using a mesh refinement study, the team verified the boundary conditions, mesh quality, and anticipated solution. Hybrid initialization was run each solution time in order to reset initial conditions and ensure convergence. The plenum was included in the simulation to better simulate the exit boundary conditions, as having the pressure outlet at the nozzle exit enforces an unrealistic value inside the chamber. The plenum size was increased from the given geometry to be 20x the nozzle exit diameter, and the pressure outlet was placed at the plenum end and walls to allow for the formation of a more physical jet.

The team found that a vortex is present using the given injector geometry and the vortex only mixes with the fuel line at the fuel injector, maintaining vortex strength along the chamber wall. Additionally, the team was able to simulate cooling of the combustion chamber wall from a 1500 K fuel line using the vortex, indicating feasibility of this design for its intended use. To improve these simulations, a transient simulation indicating flow properties and cooling over time would be beneficial. Local sizing can be optimized to be specific to injectors and nozzle throat to attempt to remedy the low orthogonal quality. Finally, calculating the true combustion temperature of the propellants and simulating combustion is the optimal simulation to prove feasibility of the engine, however, significantly more computing power is needed for such a simulation.