ME 51 Lab 3 & 4: Fluid and Heat Transfer Analysis of a Grid Fin

Alex Talbot & Lera Anders

December 2020

1 Introduction

For this lab, the external air flow and heat transfer around a grid fin of a Falcon 9 rocket will be analyzed. The grid fin is a key component of the main engine booster that experiences extreme temperatures during flight, and controls the reentry and landing of the reusable vehicle. Studying simulations of the grid fin flow and heat transfer are necessary for understanding high points of pressure, temperature, and forces on the structure in order to achieve maximum efficiency during its deployment. Heat transfer simulations for the grid fin, in particular, are important for engineers to study as the reusable booster undergoes a temperature range of 100K to 1920K. The purpose of the CFD flow analysis in this lab is to study where the highest velocity gradient occurs on the grid fin. The locations of the highest velocity gradient are important for designing where the fin should have extra supports or how to more efficiently control the rocket by manipulating the angle of attack. These parameters depend on the velocity gradient as a high gradient will indicate points of rapidly changing speed, necessary for understanding high stress concentrations. The purpose of the heat transfer study is to observe the highest temperature recorded for the connecting part between the grid fin and the rocket body after a discrete change in time. This will indicate the region on the fin that heats the fastest and could indicate areas that potentially contain high thermal stresses. The point of attachment for the fin to the rocket experiences extreme temperature fluctuations during its life cycle. Understanding this will help guide appropriate material selection for engineers and provide designers with knowledge of weak points on the fin that may require reinforcement. Overall, engineers are motivated to study the flow and heat transfer of grid fins because of their importance to reusable boosters, which effectively lowers the cost of spaceflight travel.

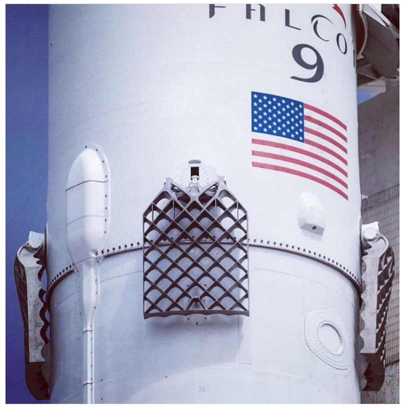

Figure 1: Grid Fin of Falcon 9

Figure 2: Burned Grid Fin of Falcon 9

2 Methods

2.1 Geometry

The geometry of the grid fin was modeled in SolidWorks and based on the geometry from G. A. Faza et al. For simplifications, the grid fin is a uniform thickness, rectangular shaped, and includes two extended bars to act as the attachment points to the rocket. The cross-hatched walls inside the rectangular box are defined by their diagonal length (80mm) and the wall thickness was approximated to be 3.50mm. The geometry is a scaled representation of an actual grid fin size. In comparison to the real structure, the geometry in this lab is generally accurate yet does not include the swept back angle on the top or the curved edges on the side walls. These simplifications will change the flow field and pressure slightly but will not cause a significant impact on the entire field. After designing the grid fin, a box was placed around the geometry for assigning boundary conditions in Comsol. The flow is predicted to be turbulent due to the intense changes in speed, so the model is 3D.

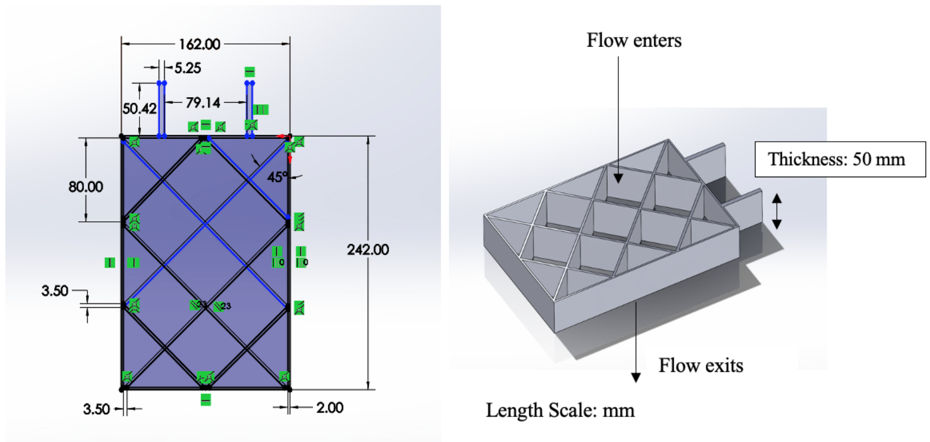

Figure 3: Geometry Dimensions

2.2 Mesh

We chose to use a coarser mesh because at first, when a normal mesh was created for the geometry, it consisted of around 4.5 million elements. The coarser mesh only consists of around 185k elements. The flow simulation with the coarser mesh took around 19.11 minutes to run, which is reasonable. The coarser mesh also gave accurate enough results for the purpose of this study. The coarser mesh was also used for the study with flow and heat transfer.

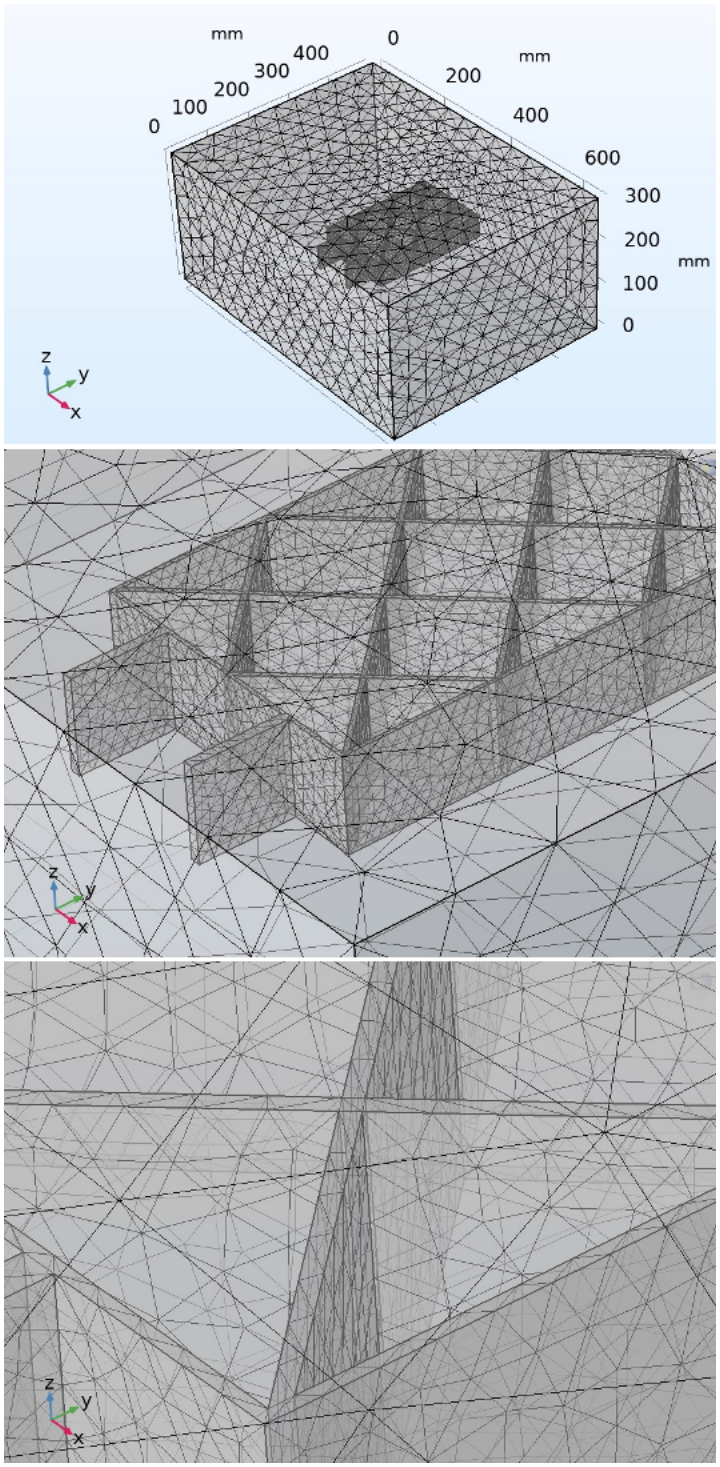

Figure 4: Three Views of Mesh: Overall, Fin, Wall

2.3 Flow Study

2.3.1 Boundary Conditions

In Comsol, the boundary conditions were applied to the bounding box. Air was set as the working fluid. The bottom of the box was set to an inlet with a flow velocity of Mach 1.5 or 514.5 m/s [3]. The turbulence intensity was set to 0.05% (0.0005) due to the still air atmosphere [4]. These values were justified by the sources included. The top of the box was set as the fluid outlet and the other sides of the bounding box were set to be open boundaries. Open boundaries were selected as they support both inflow and outflow conditions [2]. The grid fin geometry surfaces were set to interior walls. The flow was predicted to be turbulent based on the intense and frequent changes in speed, with a turbulence intensity value of 0.005. To confirm this assumption, the Reynolds number was calculated to be around 10.36 which supports a turbulent flow.

Reynolds Number Calculations:

$$U_{avg} = Velocity_{inlet} = 514.5\frac{m}{s}$$

$$\rho_{air} = 1.225\frac{kg}{m^3}$$

$$D = 242mm = 0.242m$$

$$\mu_{air} = 1.48x10^{-5}\frac{m^2}{s}$$

$$Re = \frac{\rho * U_{avg} * D}{\mu} = \frac{1.225\frac{kg}{m^3} * 514.5\frac{m}{s} * 0.242m}{1.48x10^{-5}\frac{m^2}{s}} = 10305643.6 = 10.3^6$$

2.3.2 Computation

As the flow is turbulent, the turbulence model was chosen to be $k - \epsilon$. This model performs well for complex geometries in external flow situations, such as a grid fin with external air flow. Also, the k-ϵ model has a good convergence rate and is used for industrial applications. After running the solution, the computation time was recorded to be 19 minutes and 11 seconds.

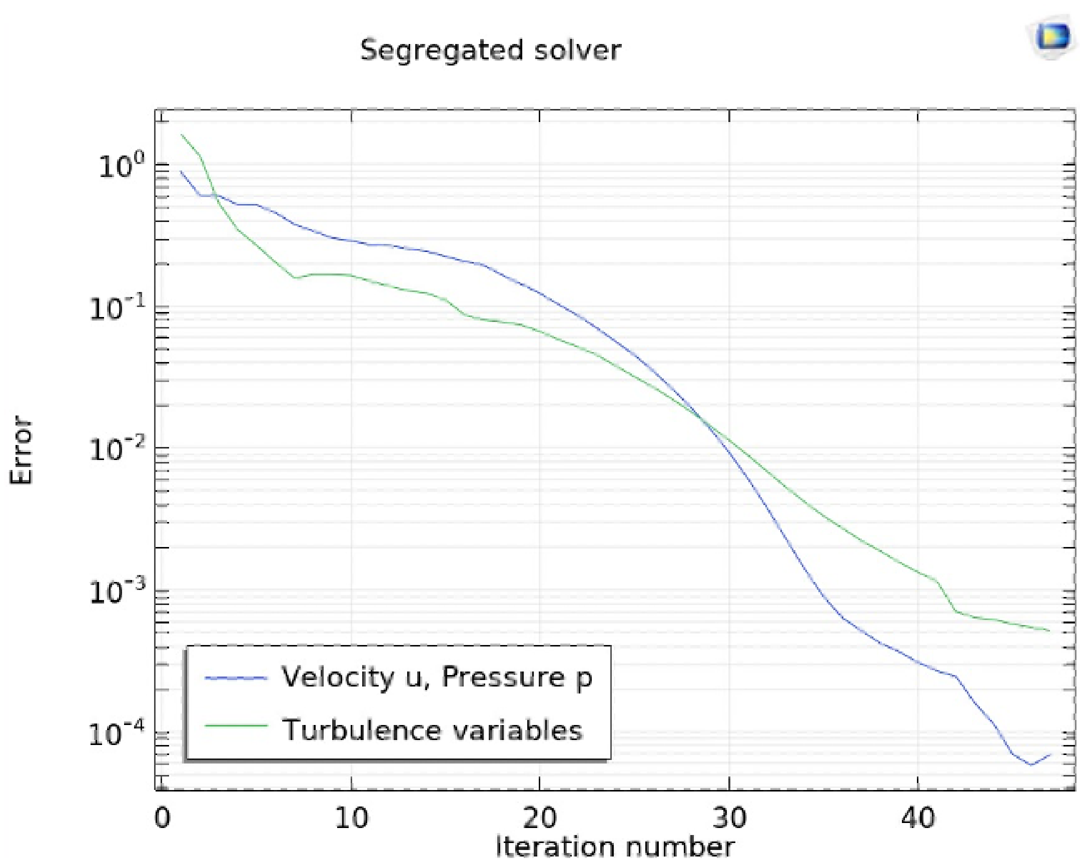

Figure 5: Convergence Plot for Flow Study

2.4 Heat Transfer Study

2.4.1 Boundary Conditions

In Comsol, boundary conditions were set for both the bounding box and the fin. The bounding box was set to be the region of fluid flow and the fin was set to be solid. The material for the fluid was set to air and the material of the fin was set to titanium. Initial temperatures were given for both the fin (100k) and the fluid (450K)[1]. The inflow was set to be coming in through the bottom face of the bounding box and the outflow was set to the top face. The remaining walls of the box were set as open boundaries. For this time dependant study, parameters for time also had to be set. Initial time was set to 0 seconds, the step was set to 0.01 seconds, and the ending time was set to 0.02 seconds.

2.4.2 Computation

The heat transfer model used in the simulations was Heat Transfer in Solids and Fluids. This model was chosen based on its use for industrial applications and our previous flow setup using the k-ϵ model. Solids and Fluids was selected as the study contained compressible flow of air for the fluid and a titanium structure for the solid. Additionally, a time dependent study was chosen since the rocket booster induces a positive heat source during landing that increases the temperature of the fin over time. The time dependant study was necessary in order to see any change in temperature of the fin from its initial state. The time intervals of the study were chosen based on trial and error in order to find a time that showed the fin displaying temperature profiles during the transitional period while various parts of the fin were at temperatures between the initial temperature of the fin and the initial temperature of the fluid. Without the appropriate time interval, no apparent temperature gradient could be seen. Because the turbulent flow setup was time independent, the heat transfer study had to be run separately from the flow study. The convergence plot is shown below in Figure 6, and the computation time was 6 seconds because only heat transfer was modeled but with the same flow setup.

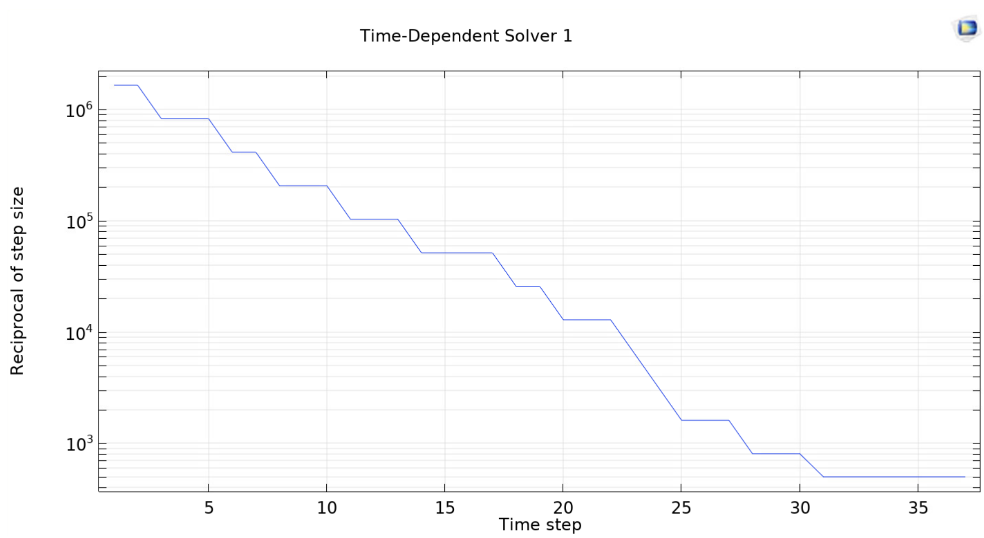

Figure 6: Convergence Plot for Heat Transfer

3 Results and Validation

3.1 Results

3.1.1 Flow Study Results

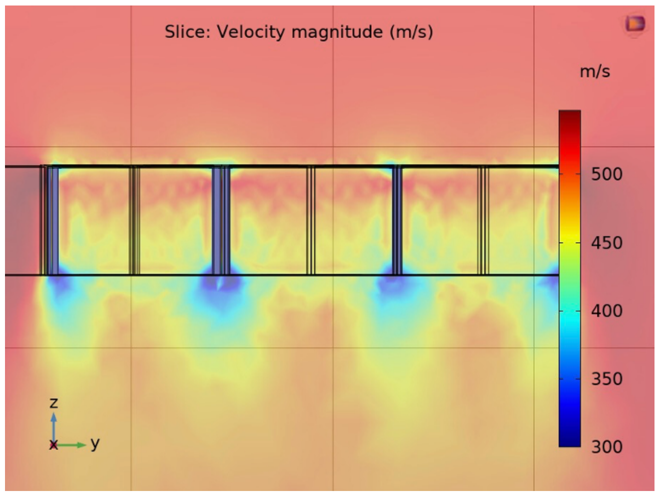

Figure 7: Side View (ZY) of Velocity Magnitude

In the figure, the flow is coming from the top and exiting through the bottom in the negative Z direction. This plot shows how the flow field slows from being disturbed by the grid crossings. The highest velocity gradient is at the top edge as it is the leading edge. In this view, four points show major changes in the flow field, demonstrating a high velocity gradient at the top edge. The bottom points have a propagating disturbance in the flow because there are three intersecting walls inside the geometry. The three other vertical lines do not have as great a flow change because there are only two intersecting walls.

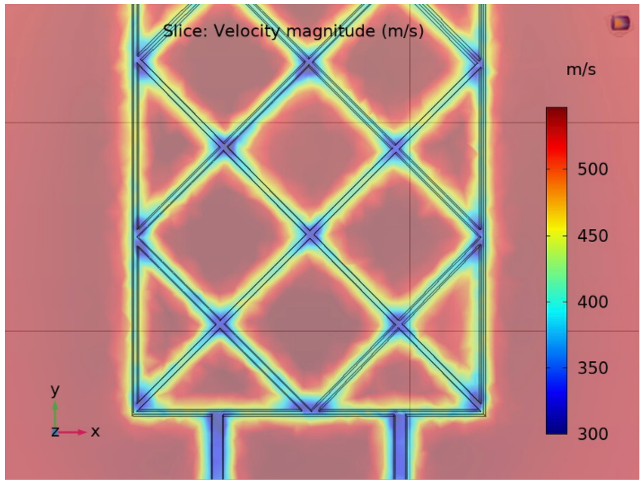

Figure 8: Top View of Velocity Magnitude

This view indicates that the intersecting nodes of the walls experience high stresses as they have larger areas of material blocking the flow.

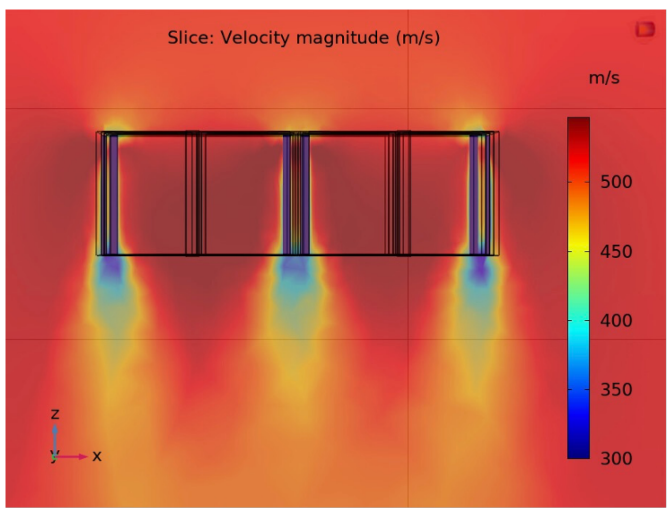

Figure 9: Side View (ZX) of Velocity Magnitude

This figure shows a slice of the grid fin in the middle with three intersecting nodes.

3.1.2 Heat Transfer Results

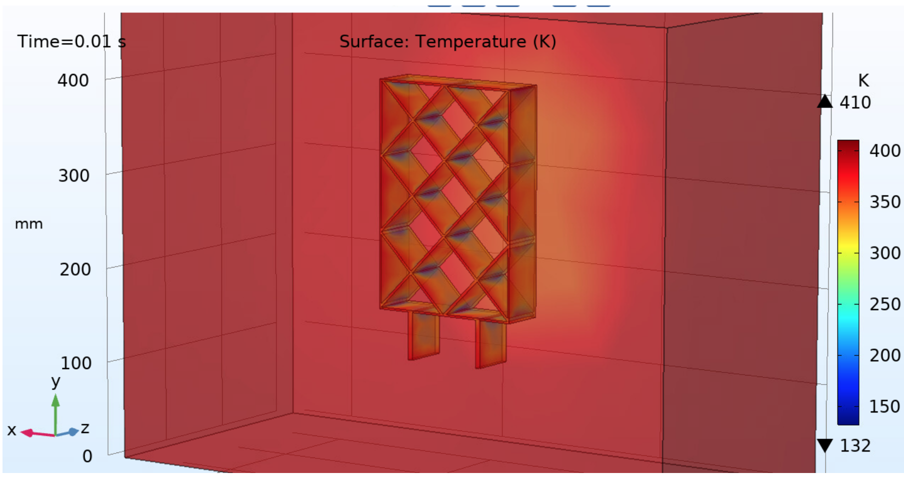

Figure 10: Wide Field View of Surface Temperature Profile for Grid Fin

This figure depicts the general temperature profile for the grid fin. The notable features are the leading edge, where the maximum surface temperature occurs, and intersections of the grid walls, where the lowest temperatures can be seen .

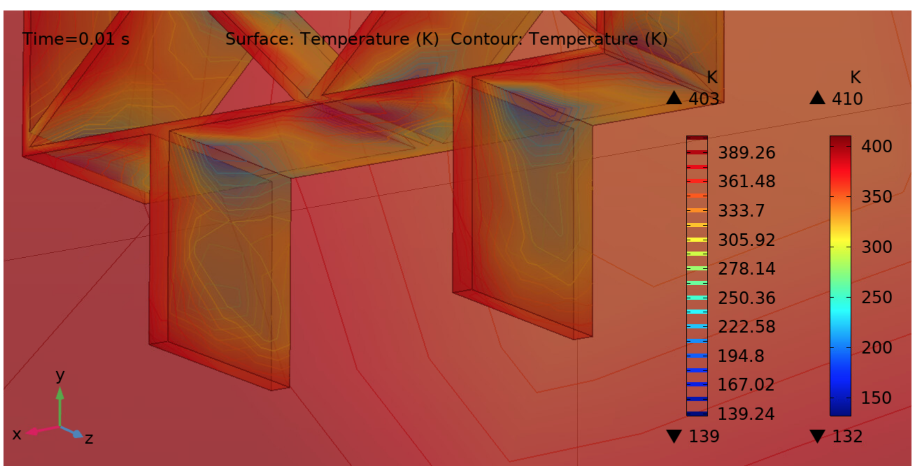

Figure 11: Surface Temperature Profile and Contour of Connecting Piece

This figure illustrates the contours and surface temperature around the connecting section between the grid fin and the rocket body. The key observations are how the surface temperature is hottest at the leading edge of the face and how the contour lines show that the "T" connection creates a permeating temperature disturbance.

3.2 Validation

3.2.1 Flow

The journal paper that was referenced for validation studied grid fins using CFD simulations. A comprehensive introduction of grid fins and an evaluation of the geometry, mesh, and results were discussed, making this source a good point of comparison. The paper evaluated the swept angle effects on grid fins' aerodynamic performance, mainly studying a 0 degree, 15 degree forward, and 15 degree backwards angle of attack. In the paper's setup, the turbulence model was k-ϵ, and the bounding domain was a quadrilateral prism with a mesh element count around 3.5 million [3]. The CFD results of the paper focused on Mach number and pressure contour plots for the different angles. For the purpose of this lab, the Mach number contour plots of the 0 degree angle will serve as a validation of our results that are focusing on the highest velocity gradient. As seen in figure 10, similar intersecting nodes have high velocity gradients compared to figure 9b. One key quantitative difference between the paper and our simulation is the size of the flow separation. The journal paper has a noticeably smaller flow separation compared to the larger flow separation seen in figure 9b, this is a result of the thicker walls in our geometry. Taking this distinction into account, our flow and results seem reasonable.

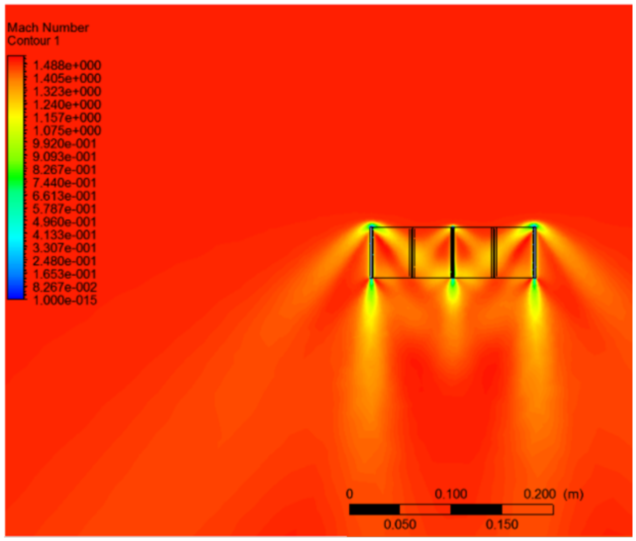

Figure 12: Velocity Gradient (Mach Number) Plot of the Flow Study Journal Paper [3]

3.2.2 Heat Transfer

The literature paper, "Fluid-Thermal-Structure Coupled Analysis for Grid Fins for Hypersonic Flight Vehicle", was used as the validation resource to check the reasonableness of our heat transfer results. This paper was chosen as it included a thermal study of grid fins and analyzed the temperature gradient of the structure, focusing on regions with a high gradient as that effects the strength and stiffness of the grid fin. For this paper's setup, a 200 second time-dependent simulation was conducted with the fin material set as a niobium alloy with an initial temperature of 293.15K. The flow field was set to a Mach number of 6, and similar to the previous validation paper, varying angles of attack from 0^0 to 15^0 were also tested. However, there were minimal details regarding the setup, so assumptions regarding the heat transfer model and an incoming temperature were made for this particular case. The flow setup was determined by the first validation paper, so those parameters were consistent and did not replicate the heat transfer study. As such, the temperature profile results were scaled for this study and the reasonableness of our data was evaluated based on the gradient's location.

From the heat transfer results, the gradients were in similar locations. The leading edge of the fin had the maximum surface temperature. Both the validation graph, Figure 13, and our graph, Figure 11, have similar temperature profile shapes near the "T" section where the connecting piece merges with the grid fin. These geographic similarities indicate that our data and results are reasonable enough to be accepted. The higher temperature scale for the validation paper is responsible for the greater surface temperature at the leading edge compared to the results from this study. The resultant discrepancies are insignificant enough that we can accept the new heat transfer study.

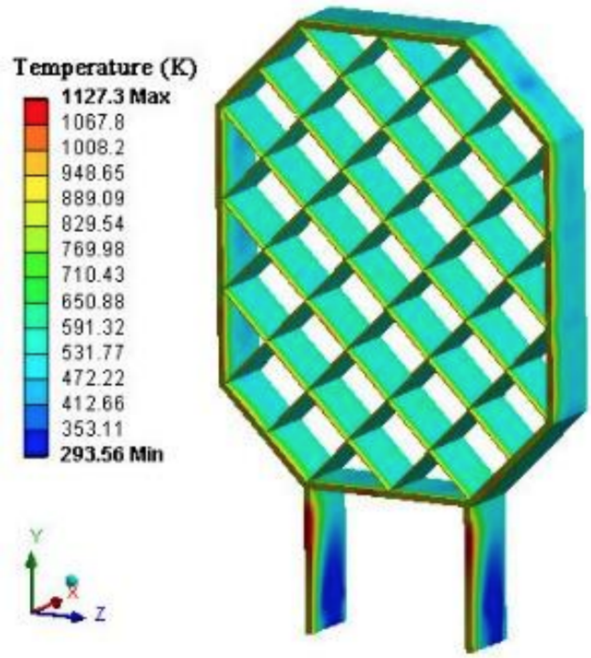

Figure 13: Temperature Profile Plot of the Heat Transfer Study Journal Paper [5]

4 Conclusions and Future Work

4.1 Flow

Our simulation showed that the velocity gradient has the highest magnitude near the nodes of the leading edge of the fin. This makes sense because the leading edge of the fin is a flat face that is perpendicular to the oncoming flow. The nodes are the locations on the leading edge with the largest area, therefore causing the most flow disruption. The significant flow disruption causes a quick drop in velocity and a high velocity gradient. This is useful information to know as an engineer because the leading edge would therefore encounter the highest stresses and would be an area of concern and possible failure.

The biggest challenge we faced conducting this simulation was creating the model with the correct flow boundaries. We ended up creating a bounding box that contained a cutout in the shape of the grid fin. Also, at first, we used normal mesh instead of coarser which took too long to run and eventually required cancelling and re-meshing.

If we were to redo this study, we would focus on the pressure fields instead of the velocity fields. We would look for the areas of highest pressure instead of the areas of highest velocity gradient. The areas near the fin with the highest pressures would show where the part encounters the highest stresses. The plots of the pressure field would be easier to interpret because the colors on the plot would show the areas of highest pressures. In this study, the area of highest velocity gradient is where the colors on the plot change the fastest, not where the plot is a certain color.

4.2 Heat Transfer

The heat transfer results demonstrate that the maximum surface temperature experienced by the connecting region of the grid fin and the rocket body after 0.01 seconds was 410K and this occurred near the leading edge of the connecting piece. This indicates that this region of the connecting part of the fin is the fastest to heat up. This result makes sense because the heat flow is moving in the positive Z-direction and first makes contact with the structure at that face, as shown in Figure 10. The pressure in the fluid at this leading edge would be the greatest because the leading face of the fin is causing the fluid to change directions. This region of higher pressure and disruption of flow allows for a greater heat flux across this leading edge of the fin. For engineers, it is important to know that this region of the fin heats up the fastest because this is where the highest thermal stresses would occur which could potentially lead to structural failure.

With this portion of the study, we struggled the most with making appropriate assumptions for the boundary conditions of the simulation. As there are many factors that occur in reality that we were unable to simulate all at once (e.g. density variations through the atmosphere, compressible flow, heat generation due to friction, etc.), we were forced to make many assumptions. Furthermore, we made assumptions for the initial temperatures of the fluid and the fin based on findings in other journal papers, which contributed to possible ranges for these temperatures.

If we were to redo this study, we would add additional parameters and boundary conditions in order to make the study more precise and accurate. To demonstrate, we would add heat generation due to friction. The high speed of fluid flow over the fin surface and the significant area of the fin that is perpendicular to the flow suggests that heat generation due to friction would likely be significant and alter our results. Although the heat transfer data may contain error, the new flow data from this study adds important information about the velocity profile around the inner walls of the grid fin to the engineering community.

References

[1] Franziska Zilker B. Sc. Aerothermal analysis of re-usable first stage during rocket retro-propulsion. pages 16–18, 2018.

[2] ComsolDocumentation Open Boundary. doc.comsol.com/5.5/doc/com.comsol.help.heat/heat-ug-ht-features.09.063.html.

[3] Faza G. A. et al. Study of swept angle effects on grid fins aerodynamics performance. pages 1–10, 2018.

[4] M Sitzmann H Riedel. In-flight investigations of atmospheric turbulence. *Aerospace Science and Technology.*

[5] Weihua Zhang Ke Peng Shengze Li, Zhenyu Jiang. Fluid-thermal-structure coupled analysis of grid fins for hypersonic flight vehicle. 2015.