CFD Simulation of Varying Fuel Jet Placement of Mach 2 Flow

By: Sara Broad

Advisor: Dr. Jeffrey Doom

Mechanical Engineering Department

South Dakota State University

April 29, 2022

Abstract

Supersonic flow is a concept that has been researched heavily for the past twenty years. It has many applications, with the most notable one being for the defense industry. This project specifically is based off a model that is being currently used for Air Force research. With supersonic flow, where the Mach number is larger than one, there has been continual research specifically on flameholders. Flameholders involve the discussion of the mixing, ignition, and combustion of the fuel that is released into the lower cavity of the scramjet. There is a current standard for the placement of fuel jets, but very little data as to why this is the best choice. The objective of this research is to prove that the current placement of the fuel jet is optimal and to use computational fluid dynamics software to visualize the fuel/air mixing of various nozzle placements. Five nozzles were created and ran in Star-CCM+ and conclusions were drawn by use of the passive scalar function. As final conclusions were made it was determined that the current fuel jet placement is optimal and similarities between it and other referenced findings was noted. Even though this project was constrained by time, there is an ending discussion on how this project could be continued for further research.

Table of Contents

Table of Figures

Introduction

In the past twenty years, research of supersonic flow has become increasingly popular with a variety of academic papers on the subject. Specifically, much research has been done on supersonic cavities and their flameholders. Flameholder research involves the mixing, ignition, and combustion of the fuel that is released into the lower level of the supersonic cavity.

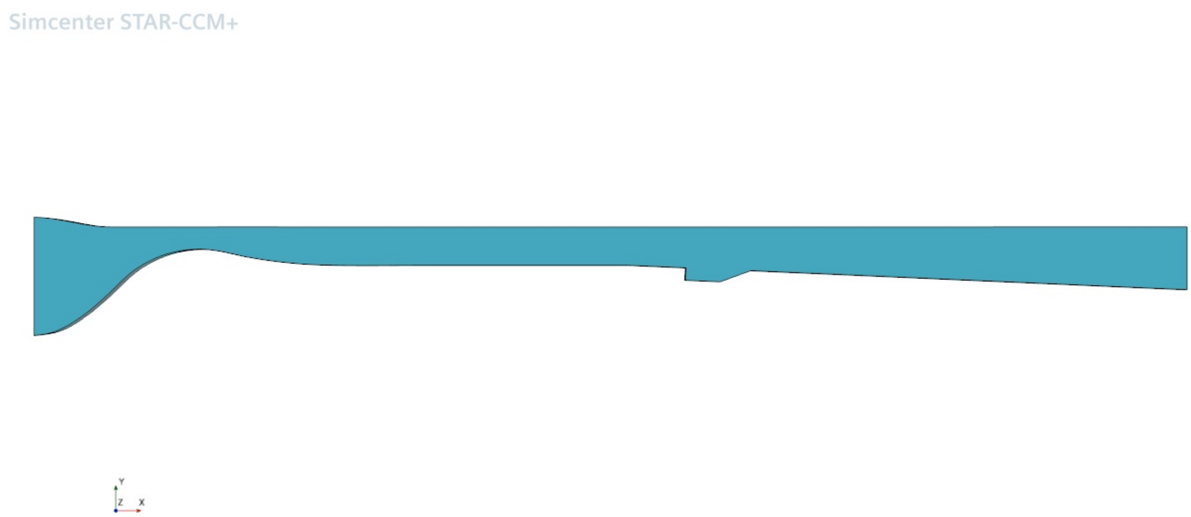

Figure 1: Supersonic Cavity Geometry

In brief, supersonic flow is achieved when the Mach number of the fluid being examined reaches a value of one or greater. Even though a high Mach number is almost always seen with turbulent flow, the transition that occurs from laminar to turbulent flow is deemed less efficient by an increase in the Mach number [8]. This transition efficiency is worth noting but will not be seen in this research since the air is entering the scramjet at a high velocity. To determine the Mach number, the velocity of the fluid at any point is divided by the speed of sound (represented with the variable c).

$$c = \sqrt{kRT} = 343\,\frac{m}{s} = 767\ mph$$

$$where: k\ is\ the\ ratio\ of\ specific\ heats\ for\ air, R\ is\ the\ ideal\ gast\ constant,$$

$$and\ T\ is\ the\ temperature\ in\ Kelvin$$

The speed of sound at standard air and temperature conditions is represented above.

$$M = \frac{v}{c}$$

$$where: c\ is\ the\ calculated\ speed\ of\ sound, and\ v\ is\ the\ velocity\ of\ the\ fluid$$

In the situation studied for this research, the Mach number reached Mach 2+ in the main chamber of the scramjet.

The completed research specifically explored how varying fuel jet placement on the supersonic cavity would affect the mixing and eventual combustion of the fuel. Currently, there is a standard placement for these nozzles, but very little research as to why it is the best option. To investigate this, the computational fluid dynamics software Star-CCM+ was used and five different nozzles were created on the model of the supersonic cavity given. One nozzle represented the current fuel jet placement, and the other four were used as tools for comparison.

To explore this topic, the potential mixing of the air and fuel (fuel/air ratio) will be visualized. Though it is possible to run a full combustion simulation in Star-CCM+, it takes a lot of time and energy to run each simulation. Visualizing the fuel/air mixing can draw the same conclusions in much less time, but more information on rich and lean supersonic combustion can be noted in the following references [15, 16].

Objective

Star-CCM+ [7, 17] was used for physics modeling of high-speed flows (Ma > 1) and visualization to observe how varying fuel jet placement on supersonic cavities alters the potential combustion and mixing of fuel. Proving that the current supersonic fuel jet placement is optimal could help form a better understanding of the processes of compressible mixing, a vital part of increasing the growing technologies that are using this process [5].

Literature Review

By reviewing some of the many peer-reviewed articles on supersonic cavities, a better base knowledge of the topic was developed. When looking at freestream air that is traveling at Mach 2, it has been documented that the wall temperature of the cavity would not lead to a significant source of error [13]. However, it is noted that as the flowfields become reactive, there could be sources of error worth noting in these walls [13]. Due to the fact that no combustion was simulated in this research, this source of error was deemed to be negligible.

Though there has been little research specifying why the current fuel injection area is best for supersonic flow, it has been reported that having normal or tangential injection has shown success [9]. As expected, research has concluded that having directly fueled cavities is optimal for injecting fuel into the combustion chamber [4]. This will be represented in nozzles 1-4, with nozzle 5 showing what would happen if the fuel was not injected in the combustion chamber. For this specific research the true fuel that would be injected into the supersonic chamber is unknown, but it can be assumed that the fuel would be ethylene or ethylene-like. Even though combustion is deemed out of the scope of this research, much research was reviewed to gain a better understanding on how others have approached ethylene injection in supersonic flow situations [1, 18].

Scramjet cavity flameholders have been researched and experimentally studied for supersonic flow in numerous ways. It was found that as the inlet is distorted, the fuel distribution within the cavity is affected [6]. This is interesting to note as the way the fuel is distributed has a large effect on the overall combustion of the fuel/air mixture. However, besides the placement of the fuel jet, this research will not cover distortion of the inlets. In terms of achieving good flameholding and stabilization, one of the best ways to do that is by the organization of a recirculation area where the fuel and air can mix before ignition [2]. This technique is the one that will be represented and seen in the research.

Star-CCM+

One of the reasons that supersonic flow has become a more popular research topic is due to the creation of computer aided engineering software. The specific software used in this project, Star-CCM+, is a multiphysics computational fluid dynamics (CFD) solver that is used to research and solve engineering problems in real-world conditions. CFD solvers allow engineers to visualize the analysis of fluid flow and how it changes based on the conditions and variables assigned that can be changed as needed by the engineer. This software uses the Navier-Stokes equations as a basis for solving the problem assigned, and it has been proven that CFD can accurately study and interpret turbulent flows [3]. The Navier-Stokes equations are referenced often in fluid dynamics, and there has been much research into defining boundary conditions for compressible and incompressible flows [14].

The software does have different approaches that are available for modeling turbulence. The two most noteworthy options are RANS (Reynolds-Averaged Navier-Stokes) and LES (Large eddy simulation). LES allows for larger three-dimensional, unsteady, turbulent motions to be represented, while RANS utilizes the governing equations to determine the variable fields as a mean value with a fluctuating component that can change for each iteration. While in some ways LES may allow for more exact results, it was found by Peterson [12] that using RANS can take advantage of the flow symmetries that occur in the supersonic cavity. For this reason, and because it is easier to run using this option, simulations using the RANS method were completed.

Review of Software

When starting a new project, it is always a good idea review the software being used and confirm the results that were given. Therefore, a simulation was ran with the given model and variables to ensure that Mach 2 flow was reached.

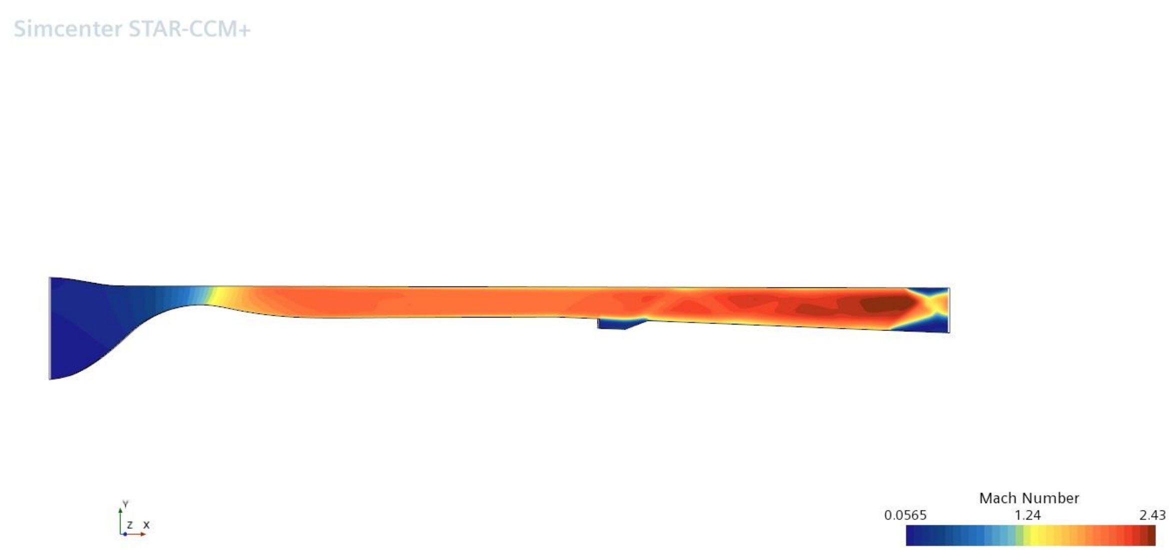

Figure 2: Mach Number

Once the first simulation was complete, more practice using the software was performed. By varying the given geometry using separate CAD software, simulations were run to see how the Mach number varied. The inlet and outlet of the supersonic cavity were changed to be smaller and then larger than the original value.

$$Original\ Geometry: Inlet = 6.16\ in\ and\ Outlet = 3.27\ in$$

$$Small\ Inlet: Inlet = 3\ in\ and\ Outlet = 3.27\ in$$

$$Large\ Inlet: Inlet = 9\ in\ and\ Outlet = 3.27\ in$$

$$Small\ Outlet: Inlet = 6.16\ in\ and\ Outlet = 1.5\ in$$

$$Large\ Outlet: Inlet = 6.16\ in\ and\ Outlet = 6\ in$$

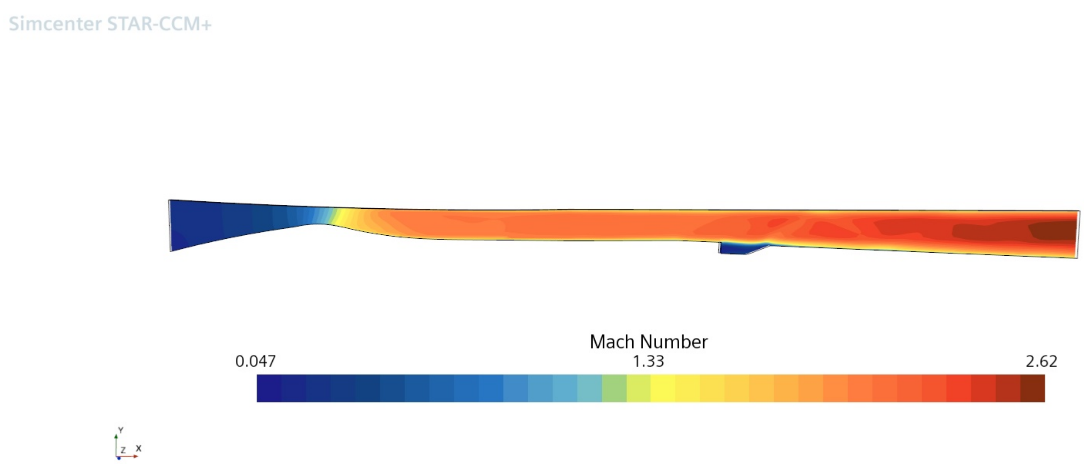

Figure 3: Small Inlet Mach Number

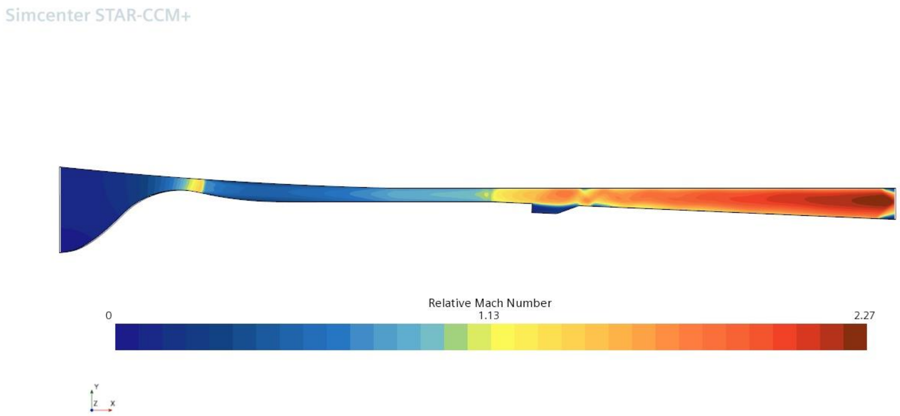

Figure 4: Large Inlet Mach Number

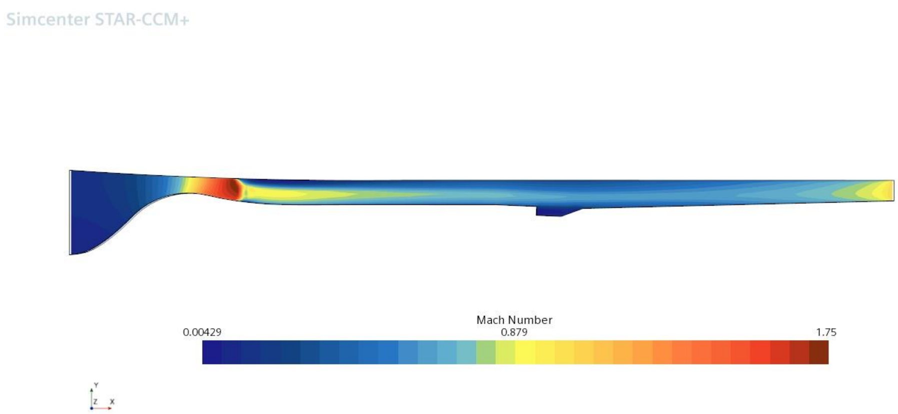

Figure 5: Small Outlet Mach Number

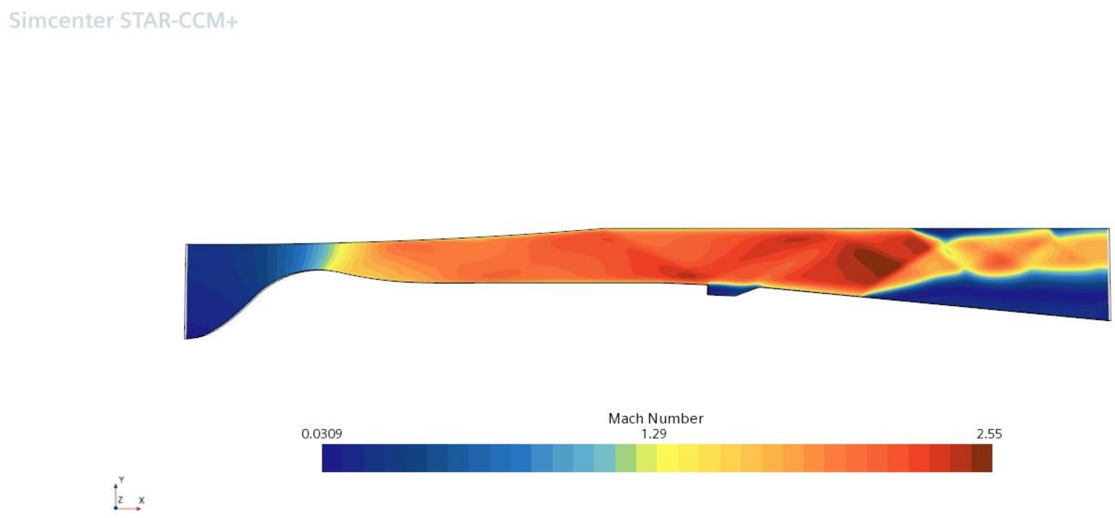

Figure 6: Large Outlet Mach Number

The simulations showed expected changes in Mach number and fluid flow based on the geometry changes. This helps conclude that the results found for the fuel jets that were simulated are accurate.

Methodology

Mesh Values

To achieve accurate results, a mesh must be created in the CFD software. A broken or poor mesh will yield inaccurate results. The mesh used had the following properties:

$$Enabled\ Meshing\ Models: Prism\ Layer\ Meshing, Surface\ Wrapper,$$

$$Surface\ Remesher, Extruder, Trimmer$$

$$Mesh\ Base\ Size: 0.001\ m$$

$$Prism\ Layers: 4$$

$$Nozzles\ Base\ Size\ Percentage: 15$$

Figure 7: Mesh Scene

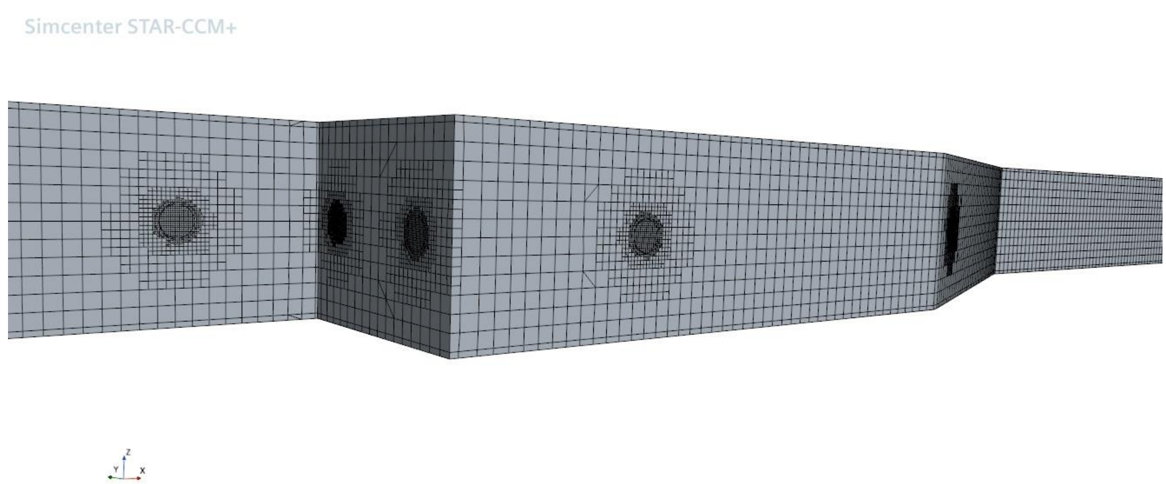

Figure 8: Nozzle Mesh

As can be seen in Figure 7, the mesh for the entire supersonic cavity was created to be very fine. However, since the main area of study was the nozzle inlets created, Figure 8 shows how the mesh was developed to be much more refined in those areas. This allows for a more accurate simulation to be ran in less time.

Physics Values

The variables for pressure, temperature, and velocity had already been determined by the advisor of this project. Additional information on the supersonic cavity is given by this reference [10]. The following values and physics models were used:

$$Physics\ Models: Steady - state, coupled\ flow, ideal\ gas,$$

$$passive\ scalar, k - omega, turbulent$$

$$Pressure: 0\ Pa$$

$$Static\ Temperature: 300\ K$$

$$Inlet\ Total\ Pressure: 483000\ Pa$$

$$Inlet\ Total\ Temperature: 589\ K$$

$$Reference\ Pressure: 101325\ Pa$$

$$Maximum\ Allowable\ Temperature: 5000\ K$$

Analysis and Results

All the nozzles were simulated as velocity inlets with a velocity magnitude of 100 m/s. While in the real-world the fuel flowing through the nozzles would be ethylene, it was modeled as air since the two fluids are very similar in density. The passive scalar function was used since it shows the mixing of the fuel and air while considering the entire supersonic cavity system, but it will not show the interference of the main cavity in the visualization.

Nozzle 1

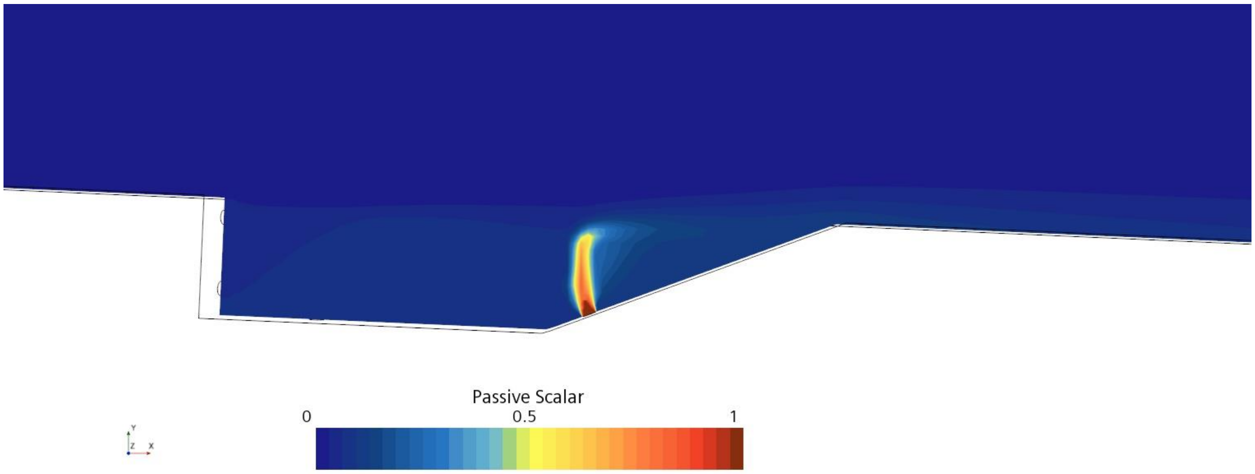

Figure 9: Passive Scalar Nozzle 1

This nozzle represents the current real-world fuel nozzle placement. Note that the fuel concentration and distribution is showing peak recirculation near the bottom wall. This observation was also noted by the following reference [11]. This similarity in the CFD simulation gives confidence in the accuracy of this research.

Nozzle 2

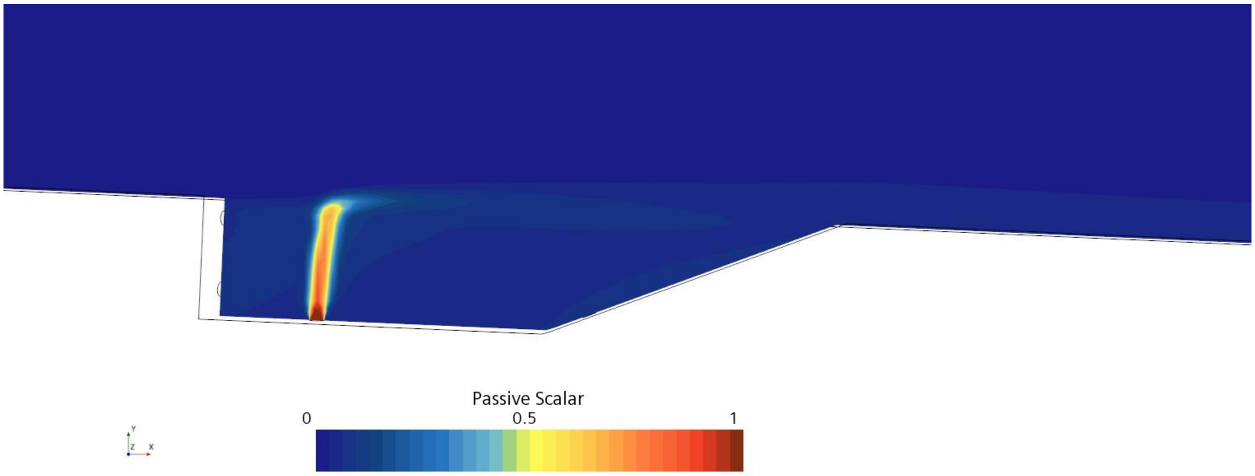

Figure 10: Passive Scalar Nozzle 2

Nozzle 2 shows fuel flowing into the cavity from the bottom section of the cavity. You can see very poor circulation at the bottom right of the cavity, which leads to a very poor fuel/air mixture.

Nozzle 3

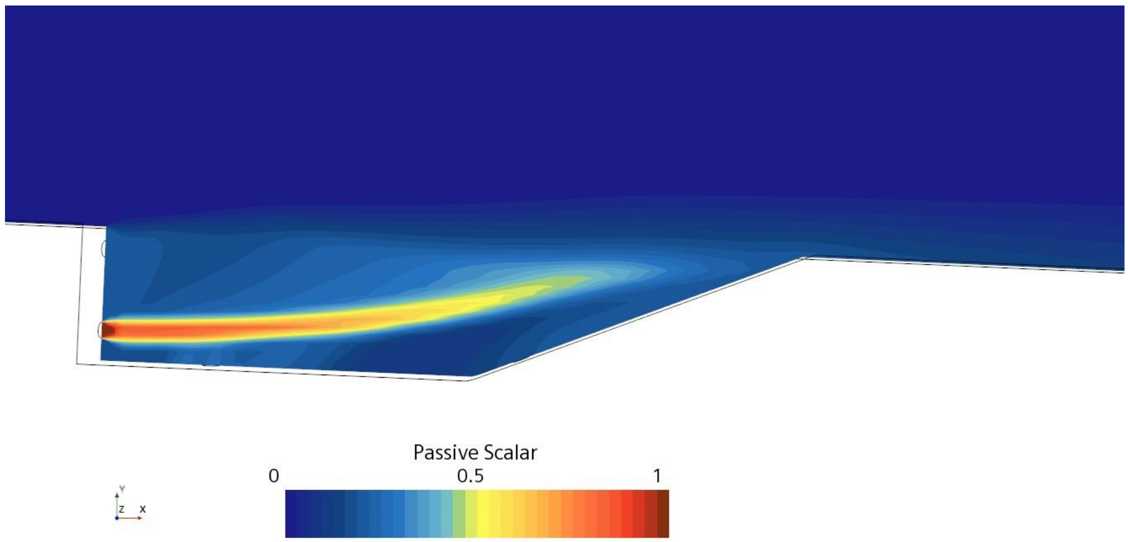

Figure 11: Passive Scalar Nozzle 3

Nozzle 3 shows a representation of the fuel being inserted from the bottom left side of the cavity. At first, this seemed like a better placement of the fuel jet in comparison to Nozzle 1. However, as the simulation finished you could see that there was actually very little recirculation in the cavity. Instead, there was a lot of unmixed fuel that would be combusted leading to a situation where the combustion would be considered fuel rich. A fuel rich combustion is not ideal in supersonic cavities.

Nozzle 4

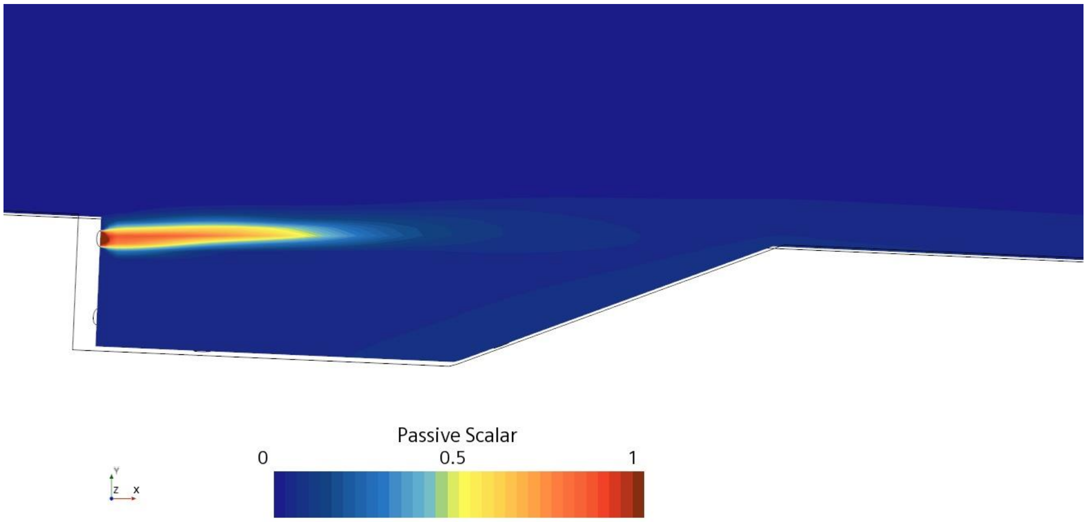

Figure 12: Passive Scalar Nozzle 4

Nozzle 4 shows no circulation. The way the fuel is injected straight from the left side of the mixing chamber gives it no ability to circulate with the air.

Nozzle 5

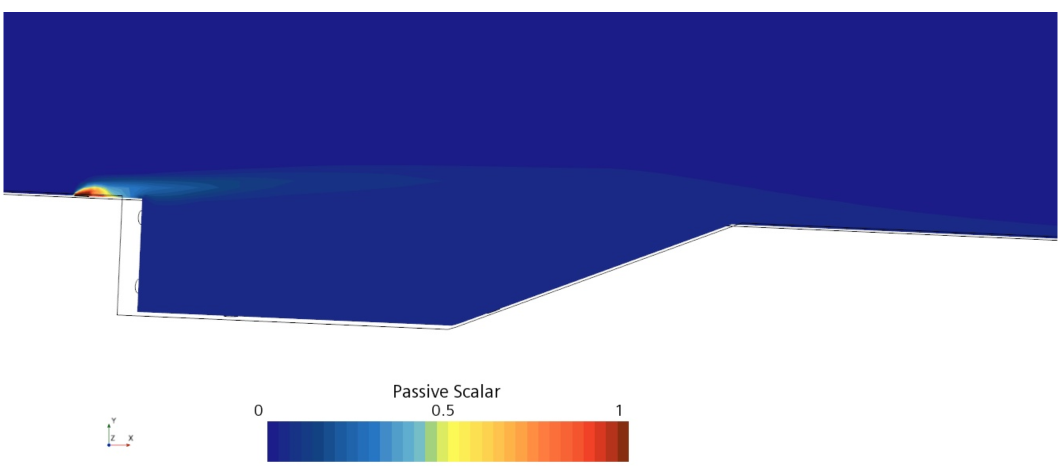

Figure 13: Passive Scalar Nozzle 5

The last nozzle that was created demonstrates why the fuel must be injected into the lower chamber and not directly into the main flow of air. The air is moving so fast in the main chamber, the fuel has no time to mix and eventually combust. A fuel inlet at this position would lead to a large waste in resources.

Conclusions

Based on the results, the current fuel jet placement represented by Nozzle 1 is the best option. The placement of the nozzle allows for recirculation to occur in the cavity. The more the fuel/air mixture recirculates, the better the combustion. In comparison, Nozzle 2 is placed on the other side of the cavity. The fuel/air mixture is not placed to achieve good recirculation in the chamber, which would yield sub-par combustion. Nozzle 4 is much the same way. Even though it is placed where more recirculation would be possible, it is too close to the air flowing through the main cavity. The air that is moving at Mach 2 would cause the fuel being injected to have little time to combust. Nozzle 3 appeared to be placed better at first, but over time the simulation showed that the upper area of the cavity has no circulation of the air/fuel mixture. This unused space would cause fuel rich combustion, which is not ideal. Fuel being released through Nozzle 5 would not combust due to the speed of the air in the main cavity. These observations prove that the current nozzle placement is ideal for supersonic cavities.

Further Research

If this research was to be continued, showing the actual combustion of the fuel/air mixture in the cavity would be a very good next step. The reason this wasn't completed for this research was due to the complexity of simulations involving combustion and time constraints. Visualizing the combustion to further prove the results would be a good extension to this research. Also, comparing results to other research by changing the inlets to be distorted would be a good way to prove already published data, or choosing to physically model the inlet and change the length and diameter to find the optimized length and size ratio would be a way to easily continue the research and add to the flameholder discussion.

References

[1] Baccarella, D., Liu, Q., McGann, B. J., and Lee, T. (2019). "Combustion induced choking and unstart initiation in a circular constant-area supersonic flow." *AIAA Journal*, 57(12), 5365–5376.

[2] Ben-Yakar, A., and Hanson, R. (1998). "Cavity Flameholders for ignition and flame stabilization in scramjets - review and experimental study." *34th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit*.

[3] Bornhoft, B. J., Peterson, D. M., Eymann, T. A., Hassan, E. A., Hagenmaier, M., and Baurle, R. A. (2020). "Reacting rans simulations of a dual-mode ramjet combustor: A code credibility study." *AIAA Scitech 2020 Forum*.

[4] Cisneros-Garibay, E., Pantano, C., and Freund, J. (2022). "Inert versus reacting turbulent flow in a supersonic cavity flameholder." *AIAA SCITECH 2022 Forum*.

[5] FREUND, J. O. N. A. T. H. A. N. B., LELE, S. A. N. J. I. V. A. K., and MOIN, P. A. R. V. I. Z. (2000). "Compressibility effects in a turbulent annular mixing layer. part 1. turbulence and growth rate." *Journal of Fluid Mechanics*, 421, 229–267.

[6] McGann, B., Lee, T., Ombrello, T., Carter, C. D., Hammack, S. D., and Do, H. (2019). "Inlet distortion effects on fuel distribution and ignition in scramjet cavity flameholder." *Journal of Propulsion and Power*, 35(3), 601–613.

[7] "Multiphysics Computational Fluid Dynamics (CFD) Simulation Software: Siemens Software." (n.d.). *Siemens Digital Industries Software*, <https://www.plm.automation.siemens.com/global/en/products/simcenter/STAR-CCM.html> (Apr. 24, 2022).

[8] PANTANO, C., and SARKAR, S. (2002). "A study of compressibility effects in the high-speed turbulent shear layer using direct simulation." *Journal of Fluid Mechanics*, 451, 329–371.

[9] Peltier, S., and Carter, C. D. (2015). "Response of a mach 3 cavity flameholder to a shock-induced distortion." *53rd AIAA Aerospace Sciences Meeting*.

[10] Peterson, D. M., & Hassan, E. A. (2016). Numerical predictions of mixing in a supersonic cavity flameholder. *54th AIAA Aerospace Sciences Meeting*. https://doi.org/10.2514/6.2016-1899

[11] Peterson, D. M., and Hassan, E. A. (2017). "Hybrid reynolds-averaged and large-eddy simulations of combustion in a supersonic cavity flameholder." *55th AIAA Aerospace Sciences Meeting*.

[12] Peterson, D. M., Hassan, E. A., and Bornhoft, B. J. (2022). "Improved delayed detached-eddy simulation of a round supersonic combustor with and without periodic boundary conditions." *AIAA SCITECH 2022 Forum*.

[13] Peterson, D. M., Hassan, E. A., Tuttle, S. G., Hagenmaier, M. A., and Carter, C. D. (2014). "Numerical investigation of a supersonic cavity flameholder." *52nd Aerospace Sciences Meeting*.

[14] Poinsot, T. J. (1992). "Boundary conditions for direct simulations of compressible viscous flows." *Journal of Computational Physics*, 99(2), 352.

[15] Potturi, A. S., and Edwards, J. R. (2014). "Large-eddy / reynolds-averaged navier-stokes simulation of cavity-stabilized ethylene combustion." *44th AIAA Fluid Dynamics Conference*.

[16] Sanchez, A. L., Huete, C., and Williams, F. (2016). "Diffusion-flame ignition by shock-wave impingement on a hydrogen-air supersonic mixing layer." *54th AIAA Aerospace Sciences Meeting*.

[17] Simcenter STAR-CCM+, Software Package, Ver. 15.04.008, Siemens, Plano, TX, 2021.

[18] Yuan, Y., Zhang, T., Yao, W., Fan, X., and Zhang, P. (2017). "Characterization of flame stabilization modes in an ethylene-fueled supersonic combustor using time-resolved CH* chemiluminescence." *Proceedings of the Combustion Institute*, 36(2), 2919–2925.