Studying the Effects of a Backward Facing Step in Enhancing Fuel Mixing in Scramjets

Grayson S. Hayes* and Surabhi Singh†
Embry-Riddle Aeronautical University, Daytona Beach, FL, 32114

As atmospheric flight pushes closer to the hypersonic regime, it is necessary to create propulsion methods capable of sustained thrust. For this, the solid-fuel scramjet (SFSJ) has become a promising candidate for delivering high specific impulse with relatively low complexity and cost. In order to induce proper fuel-air mixing, the backward facing step (BFS) is introduced. This study uses computational fluid dynamics (CFD) to identify the flow features immediately following the BFS and evaluate the flow conditions in the region. Additionally, the effect of the step height on the length of the detached zone following the BFS was explored.

I. Introduction

As the design flight speed of an aircraft increases, it is important to take into account the various propulsion methods which can be used. Depending on the flight regime, the most effective air-breathing propulsion method will change. For Mach numbers less than 3, the turbojet is often the propulsion method of choice due to its efficiency. Beyond this Mach number they are outperformed by the solid fuel ramjet (SFRJ) engine, where a supersonic flow is slowed down through a series of reflected shocks. After this, the now subsonic flow passes through an isolator which decouples the incoming flow from the flow in the combustion chamber [1]. Combustion, either from a solid fuel grain or injected liquid fuel, occurs in the combustion chamber and is then accelerated by a converging-diverging (CD) nozzle.

As the incoming flow reaches the hypersonic regime ($M_{in} > 5$), the total pressure loss across the train of reflected shocks too large for the engine to operate efficiently [1]. Due to these reasons, the supersonic-combustion ramjet i.e. scramjet (Figure 1) was developed. The scramjet operates with supersonic combustion instead of subsonic which allows for greater incoming Mach numbers. Additionally, since the scramjet flow is already supersonic, there is no need for a CD nozzle. Instead, only a diverging nozzle is needed. The main advantage for this is that the mass flow rate through the nozzle is not limited. As such, very large mass flow rates can be achieved [2].

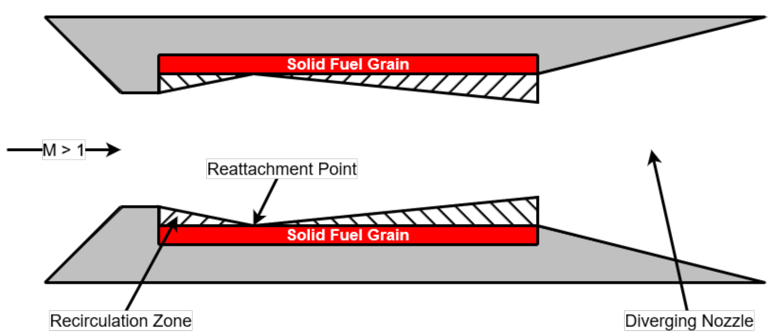

Fig. 1 Diagram of a SFSJ

As scramjets are being developed, there is a need to better study solid fuel scramjets (SFSJ) and their mixing strategies. The use of the backward facing step (BFS) in an SFSJ to induce mixing is well-documented as a method of producing adequate mixing of the fuel grain into the combustion region[1, 3–5], but little effort is placed into actually characterize the driving flow structures which aid this mixing. Immediately following the BFS, a recirculation zone is created which has relative low velocities and high temperatures compared to the core flow [3]. It is within this recirculation zone that a large stream-wise vortex exists which greatly contributes to the fuel mixing. It is the goal of this study to characterize the flow within this recirculation zone computationally and determine its effect on fuel mixing.

*B.S. Student, Department of Aerospace Engineering, and AIAA Student Member
†Assistant Professor, Department of Aerospace Engineering Aerospace Engineering, and AIAA Faculty Member

II. Simulations

For this study, CFD was used to characterize the flow structure following the BFS and identify the flow structures within the recirculation zone. The nozzle which was simulated for this study has a M_∞ of ~ 2. Additionally, no combustive processes have been simulated for the present study.

Simulations performed for this study used ANSYS Fluent with its density-based solver and the SST k-ω model [6]. The geometry for these simulations includes the internal contour of the nozzle as well as the BFS shown in Figure 2. As a note, an external domain (a 45-degree cone extending five inches) for the simulations was added to show that the setup would exit to atmosphere, but this is not pictured in Figure 2. Two step-heights were simulated during this study; a quarter-inch step and an eighth-inch step. These heights were chosen to replicate the flow-field when there is an eighth-inch fuel grain at the beginning and end of its life, with the eighth-inch representing the beginning of a burn and the quarter-inch representing the end (when there is no fuel grain left). Using this geometry at each step-height, an unstructured mesh was created in Fidelity Pointwise. The mesh cell count for these simulations was 255 thousand cells, and a hybrid structured and unstructured grid was used. An image of the mesh at the step can be found in Figure 3. The mesh for these simulations had a y^+ of ~ 0.01 for most of the domain, but there was a spike at the corner of the step at ~ 60.

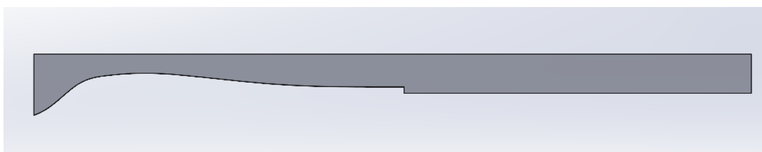

Fig. 2 Image of the simulated geometry

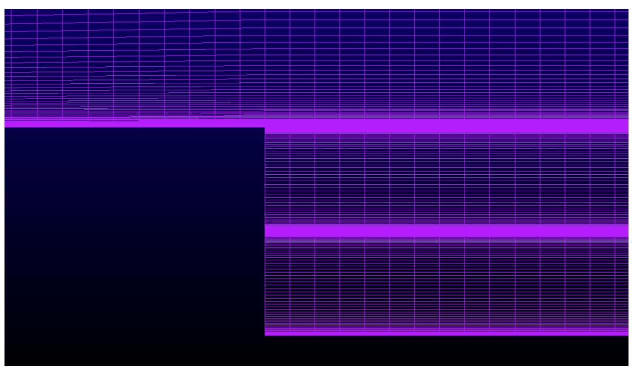

Fig. 3 Image of mesh used at the BFS

The following simulations were performed with the ultimate goal of seeing how the recirculation zone immediately after the BFS is affected by the step height and to identify the main flow structures driving mixture in SFSJs. The boundary conditions presented in Table 1 were used for all simulations discussed in this study.

Boundary Condition	Value
p_{inlet}	80 psia
p_{outlet}	14.7 psia

Table 1 CFD Boundary Conditions

III. Results

Simulations performed on the quarter-inch step (Figure 4 and Figure 5) show the same flow structures seen in literature [7]. The boundary layer thickens as the flow approaches the ramp since information is traveling upstream about the presence of a sudden expansion, but the supersonic portion of the flow can not get this information. This inability to move the information upstream results in the barrel shock which forms at the step. The recirculation zone forms immediately after the step in the zone of separation. In this zone, the flow is moving at a much lower speed than the core flow of the combustion chamber, and a vortical structure forms as seen in Figure 6.

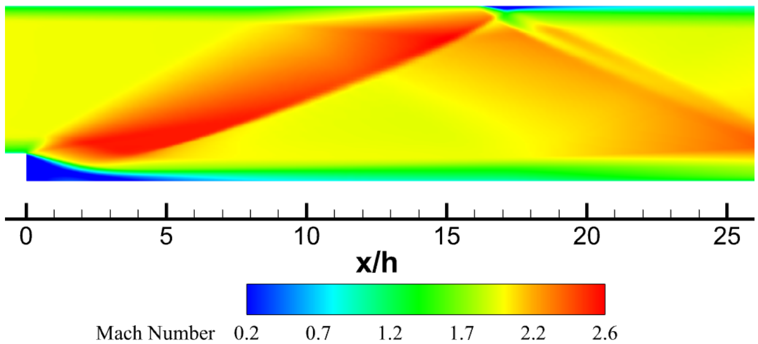

Fig. 4 Mach field following BFS

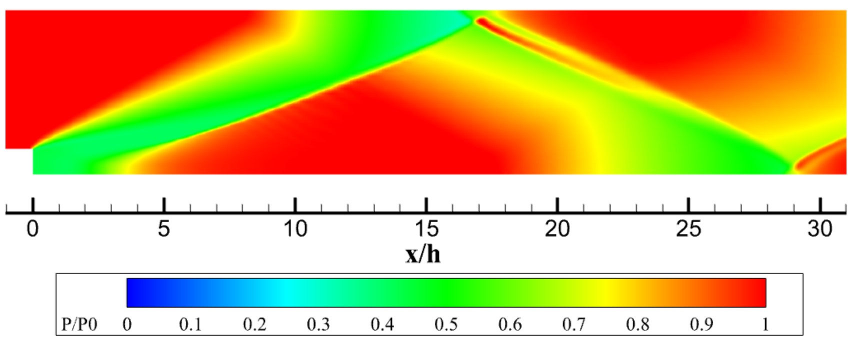

Fig. 5 Pressure field normalized by atmospheric pressure

In the flow-field following the BFS, the barrel shock propogates downstream until it impinges on the top wall, resulting in Shock-Induced Flow Separation (SIFS). This shock reflects, and this cycle repeats through the test section.

Using the wall data gathered in Fluent, the extent of the recirculation/separation region can be estimated by plotting the wall shear stress along the bottom wall of the nozzle. By using the wall shear stress, the extent of the separation region can be found by looking at the regions in which the wall shear stress is less than zero [8]. Wall shear stress distributions for the whole bottom wall and the separation region can be found in Figure 7 and Figure 8, respectively, where the x-coordinate is normalized by the step height and $x/h = 0$ is the point where the step occurs. As seen in Figure 7, there are actually several separation zones present on the bottom wall. This is because of the reflected shocks impinging on the wall and causing the flow to separate multiple times.

Looking at the eighth-inch step, Figure 9 shows the same general flow structure as the quarter-inch step. The main difference is that the core flow does not expand to as high of a Mach number when compared to Figure 4. This is expected, since the step-height is smaller, the flow does not have as much room to expand.

Figure 10 shows the zoomed-in view of the recirculation zone immediately behind the BFS. The flow structure is similar to that of the quarter-inch step height is revealed. The length of the recirculation zone in this case is smaller than that of the quarter-inch step, which is a known relationship [3]. There is a linear relationship between the step-height and the length of the recirculation zone in SFRJs [9]. In SFRJs, it is advantageous to use a larger step height because the increased size of the recirculation zone leads to an enhanced flame stability. At the same time, due to the regression of the fuel grain during operation, a larger fuel grain can be used. This would allow for the engine to have a smaller step

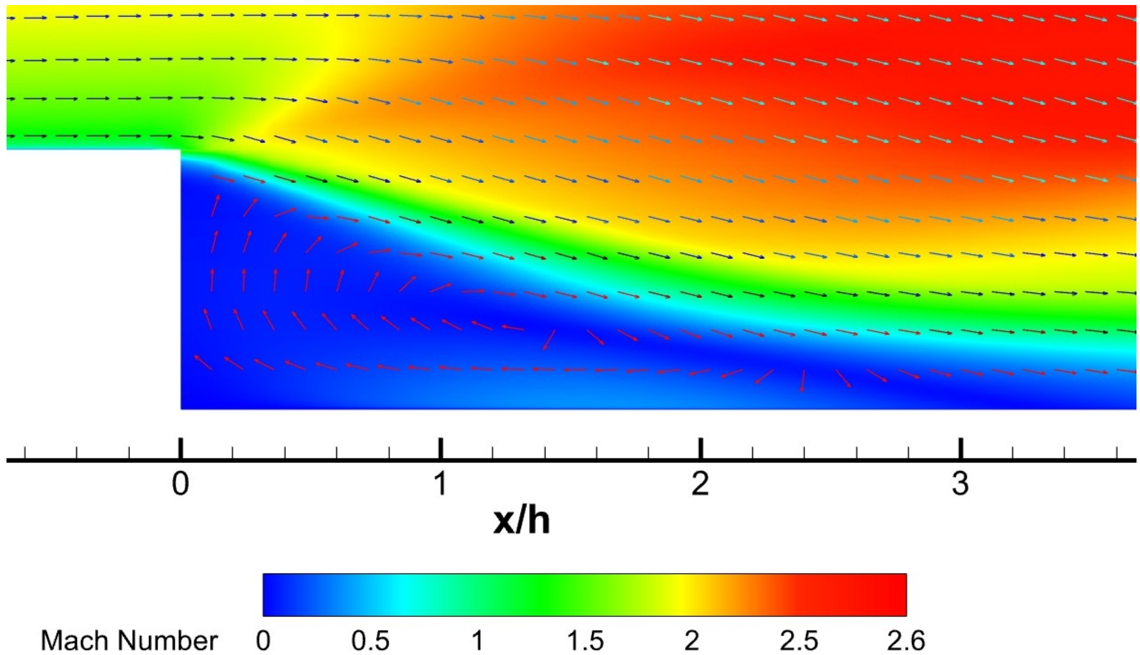

Fig. 6 Mach field following the BFS with velocity vectors showing the main flow structure

height at the beginning of operation (and, thus a weaker flameholding region) which evolved to become a larger step height over the course of operation, which would make the engine more stable later into operation.

When looking at the wall shear stress distributions for both configurations, the relationship between the separation region length and the step height becomes clearer. When comparing Figure 8 to Figure 12, the separation region shrinks from around 3.5 step-heights downstream to around 2 step-heights downstream, respectively.

A hint as to why the recirculation zone is so important for the production of fuel mixing can be found when looking at a plot for the magnitude of vorticity immediately following the BFS. Figure 13 reveals a much higher magnitude of vorticity within the separation region compared to the core flow with this characteristic also being replicated in the eighth-inch step configuration. It is likely that this higher vorticity is one of the leading contributors to the mixing induced by the recirculation zone.

In the SFSJ, the step height poses a conundrum during the design process since the internal flow regime differs from that of the SFRJ. It is important to consider that, since the core flow expands more as the step height increases, the reactants will spend less time within the combustion chamber. This will lead to a less efficient engine overall. On the contrary, the step needs to be large enough to induce proper mixing of the fuel and produce a flameholding region in which combustion is steady. To mitigate this issue, fuel grain geometries which limit the core flow velocity such as that presented in [3] have been applied to aid combustion stability in SFSJ configurations.

For the flow-field shown, it would be expected that fuel would spend a longer time in the recirculation zone when compared to the rest of the flow-field. As such, a more stable combustion process would occur and a flameholding region would form.

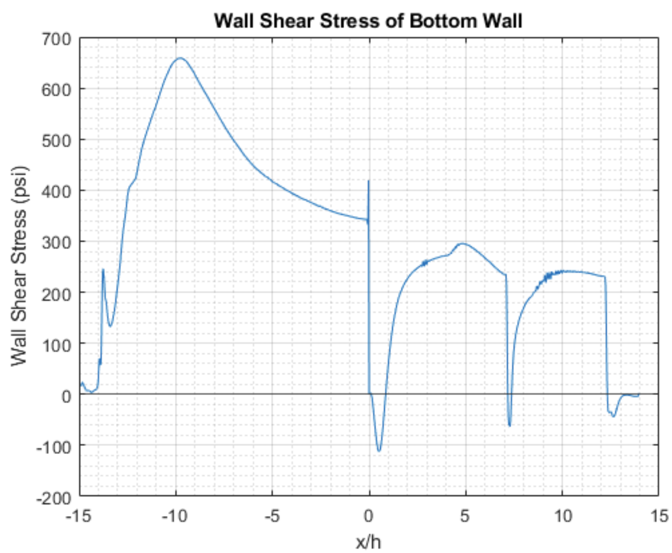

Fig. 7 Wall Shear Stress on the bottom wall for the quarter-inch step

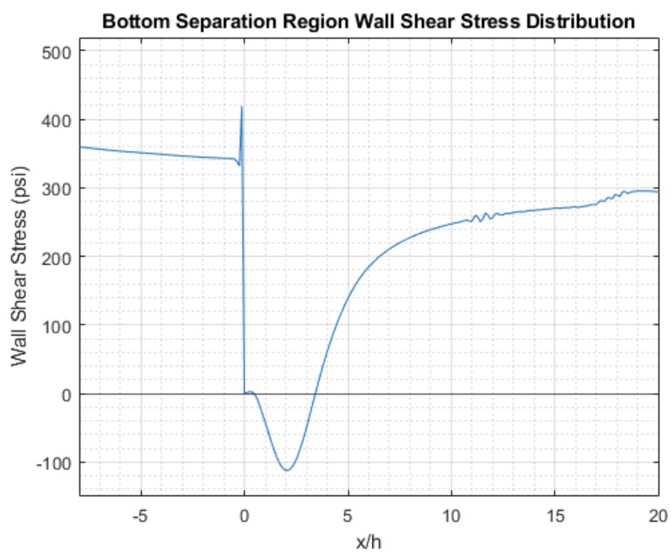

Fig. 8 Wall Shear Stress Distribution in the separation region for the quarter-inch step

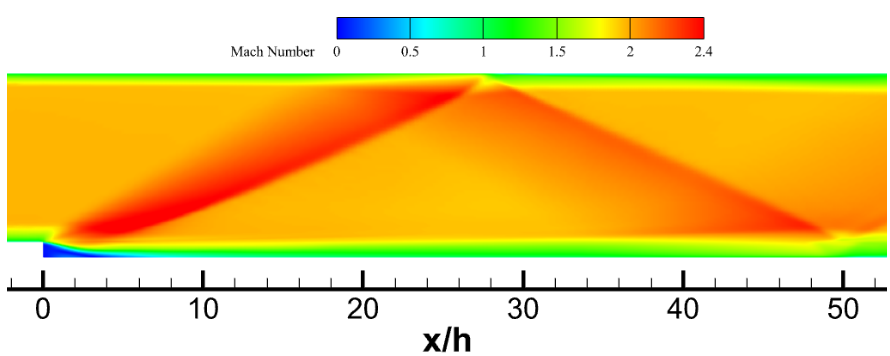

Fig. 9 Mach field following the BFS

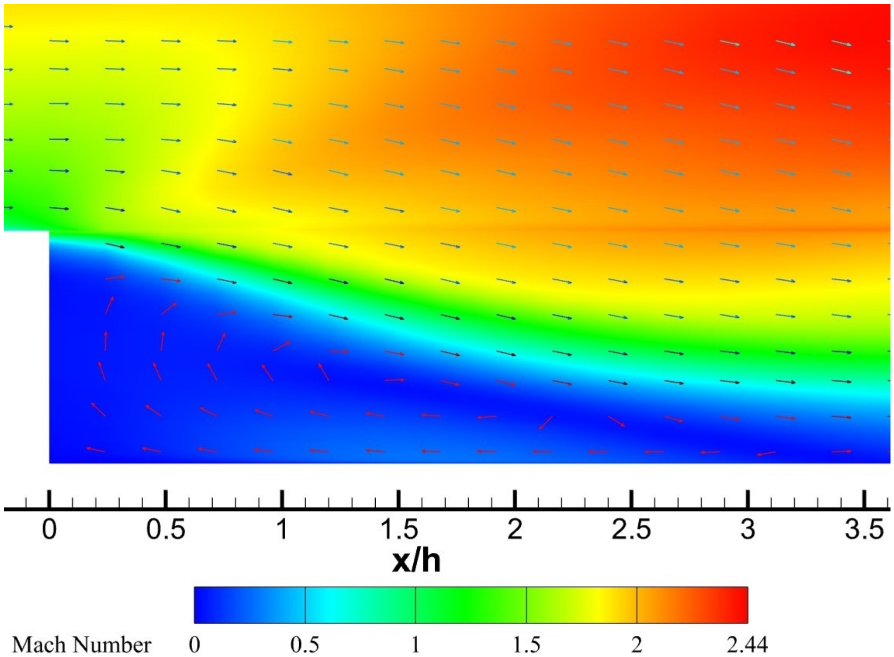

Fig. 10 Zoomed Mach field following the BFS with vectors to show the flow structure

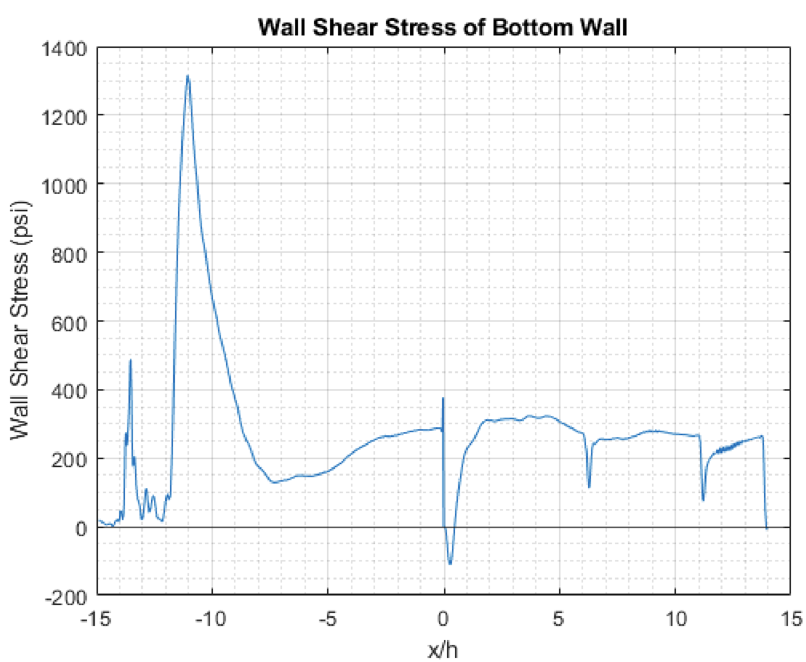

Fig. 11 Wall Shear Stress distribution along the bottom wall for the eighth-inch step

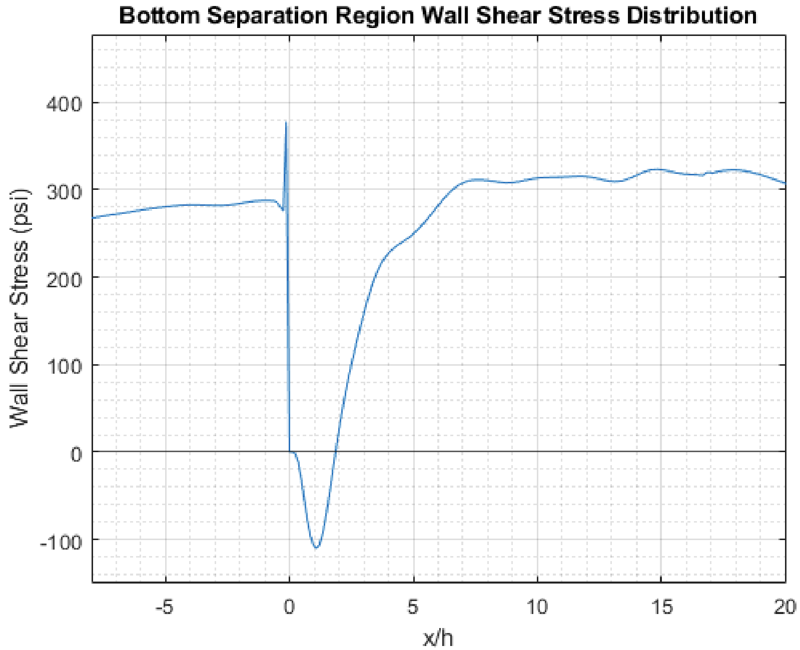

Fig. 12 Wall Shear Stress in the separation region for the eighth-inch step

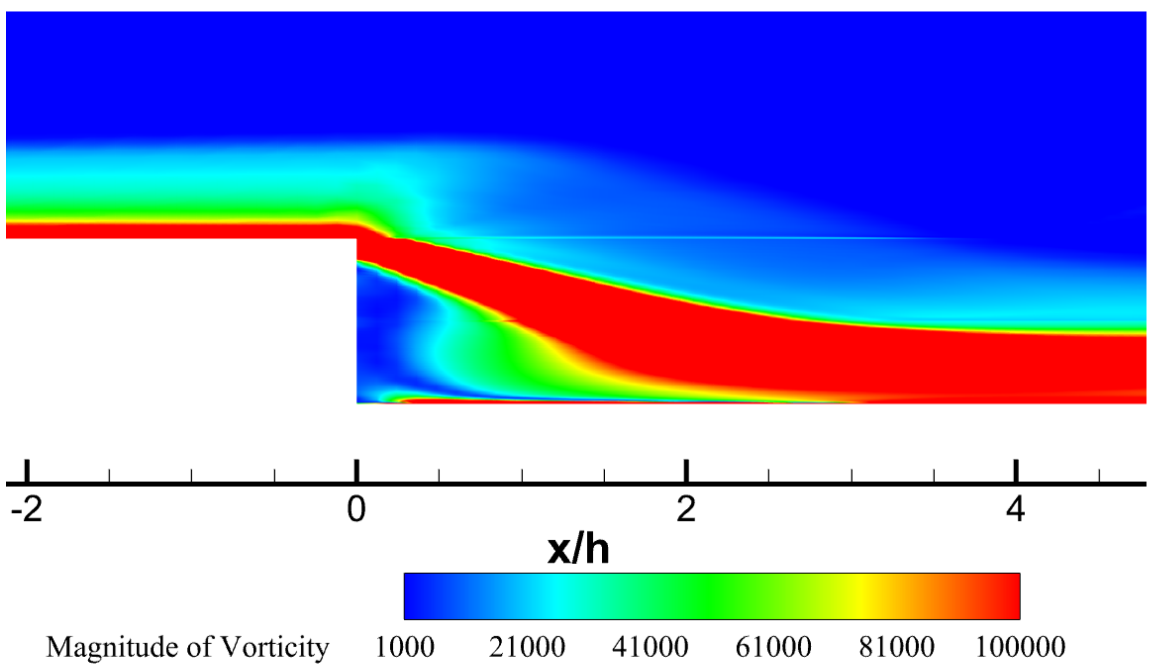

Fig. 13 Vorticity Contour for the quarter-inch step

IV. Conclusion

Overall, the present work has served as an investigation into the driving forces behind fuel-air mixture induced by the introduction of a backward facing step into a supersonic flow. This work has shown that immediately behind the BFS, a recirculation zone forms which contains a large central vortex. This region of high vorticity and low velocity keeps fuel particles in the combustion chamber for a longer duration of time. This allows for a more sustained combustion process within the region and to the creation of a flameholding region. Furthermore, the high vorticity helps in better fuel-air mixing. The creation of this flameholding region is vital to the sustained combustion and, therefore, operation of the SFSJ.

V. Future Work

In the future, work to produce a simulation which accurately reflects the behavior of the solid fuel grain to degrade under the forces imposed by the flow upon the fuel grain will be conducted. Such simulations will require the implementation of OpenFOAM, an open-source CFD solver, which allows for the use of a density-based solver with its multiphase models [10]. As such, this will provide a more accurate simulation than what Fluent can provide since Fluent requires the use of a pressure-based solver with its multiphase models. This is an issue since the pressure-based solver is not as accurate as a density-based solver when the flow-field is highly compressible.

In addition to more CFD analysis, experimental work involving the implementation of Schlieren Imaging will take place to verify the accuracy of the CFD simulations conducted and attempt to identify vortical structures in the flow. Schlieren imaging will use the density gradients present within the flowfield to create a visualization of the flow structures at the step. Vortex identification methods using Schlieren have proved to have promising results [11], and the same methods will be employed for this structure. Other flow visualization methods such as oil flow visualization and focused laser differential interferometry can also be employed to gather qualitative and quantitative data.

Following this work, it is planned to introduce a fuel grain under a Mach 2 cold flow which we can use to analyze the recession of the fuel grain under a supersonic flow and identify the main flow structures driving mixing. The evolution of these flow structures over time will be studied since the uneven recession of the fuel grain will create a different geometry, thereby changing the flow features over time. Similar studies have been performed [2, 3, 12], but none of these have been performed with a focus on the aerodynamic structures. Instead, they focus on the combustive processes which take place.

References

[1] Seleznev, R., Surzhikov, S., and Shang, J., "A review of the scramjet experimental database," *Progress in Aerospace Sciences*, Vol. 106, 2019, pp. 43–79.

[2] Cohen-Zur, A., and Natan, B., "Experimental Investigation of a Supersonic Combustion Solid Fuel Ramjet," *Journal of Propulsion and Power*, 1998.

[3] Ben-Yakar, A., Natan, B., and Gany, A., "Investigation of a Solid Fuel Scramjet Combustor," *Journal of Propulsion and Power*, Vol. 14, No. 4, 1998, pp. 447–455.

[4] Senior, W. C., Evans, J. V., Gejji, R. M., and Slabaugh, C. D., "Development and Investigation of an Optically Accessible Solid Fuel Ramjet Combustor," *AIAA SciTech*, 2021.

[5] Wang, L., Wu, Z., Chi, H., Liu, C., Tao, H., and Wang, Q., "Numerical and Experimental Study on the Solid-Fuel Scramjet Combustor," *Journal of Propulsion and Power*, Vol. 31, No. 2, 2015, pp. 685–693.

[6] Menter, F. R., "Improved two-equation k-omega turbulence models for aerodynamic flows," Tech. rep., NASA, 1992.

[7] Soni, R., Arya, N., and De, A., "Characterization of turbulent supersonic flow over a backward-facing step," *AIAA Journal*, Vol. 55, No. 5, 2017, pp. 1511–1529.

[8] Kim, H. D., Matsuo, K., and Setoguchi, T., "Investigation on onset of shock-induced separation," *Shock Waves*, Vol. 6, 1996, pp. 275–286.

[9] Netzer, A., and Gany, A., "Burning and flameholding characteristics of a miniature solid fuel ramjet combustor," *Journal of Propulsion and Power*, Vol. 7, No. 3, 1991, pp. 357–363.

[10] Harvey, E., "Combining a density-based compressiblesolver with a multiphase model," *CFD with OpenSource software*, 2020.

[11] Kindler, K., Goldhahn, E., Leopold, F., and Raffel, M., "Recent developments in background oriented schlieren methods for rotor blade tip vortex measurements," *Experiments in Fluids*, Vol. 43, 2007, pp. 233–240.

[12] Ben-Yakar, A., and Gany, A., "Experimental study of a solid fuel scramjet," *30th Joint Propulsion Conference and Exhibit*, 1994, p. 2815.