MESH INDEPENDENCE AND CFD SIMULATION OF SCRAMJET COMBUSTOR

MRIDU SAI CHARAN A S

Student, PES University, Bangalore, Karnataka, India

ABSTRACT

This paper documents the study and analysis of hydrogen-air mixing and to a limited scope on combustion in scramjet combustors. The fuel is injected into the combustor by a set of injectors present on the strut wall. The combustor geometry used in this paper is similar to the DLR (German Aerospace Center) experimental setup. The combustor setup comprises of a wedgeshaped strut assembly as the flame holder. At its base, Hydrogen is injected by virtue of the series of injectors. The geometry also consists of a diverging section at the top wall but not at the bottom wall, making the combustor a one-sided diverging section beyond the strut. In this report, cases with only air flow, i.e., without hydrogen injection, cold flow, i.e., with hydrogen injection but without combustion and reacting flows which simulate combustion are investigated. As a means of validation, the numerical results from the study are compared with experimental research conducted by the DLR. Qualitative differences are brought between numerical and experimental schlieren images. In addition, a mesh independence study was also constituted to observe convergence pattern.

KEYWORDS: Computational Fluid Dynamics, SCRAMJET, Combustion & Supersonic

INTRODUCTION

The SCRAMJET is an air breathing propulsion system capable of achieving M>6. Wide-ranging research right from experimental to numerical are being conducted all over the world for enabling the use of SCRAMJET for hypersonic propulsion. The most promising SCRAMJET engine is the one in which combustion is achieved using hydrogen as fuel. SCRAMJET engines have been built and have been successfully demonstrated by the USA on ground and in flight through their X-43 aircraft. SCRAMJET engines with hydrogen as the fuel are an active field of research. The complexities involved in this engine are: mixing of fuel and air, ignition of combustion mixture and flame holding. Combustor is a critical component of the SCRAMJET engine where majority of the above mentioned complexities are primarily encountered. Supersonic combustion comprise turbulent mixing – eddy currents, shock interaction and reflection, and heat release into the flow field. The flow field within the combustor is therefore very complex and is considered to be a significant challenge in design and development of the same with an optimized geometry which would ideally promote efficient combustion and heat transfer.

LITERATURE REVIEW

The experiments conducted by the German Aerospace Center (DLR) have shaped the way for experimental research in scramjet combustors. In this report, the findings of the DLR report titled - "Experimental Investigation of Combustion Process in a Supersonic Combustion Ramjet (SCRAMJET)" by Waidmann, et al.[1] is computationally simulated and the results between the two are compared and the computational techniques used for this comparison are validated.
COMBUSTOR GEOMETRY

The combustor geometry considered for analysis is similar to the experimental setup given in Waidmann, et al. [1] and used at the DLR. In the experiment, inlet air is preheated, expanded through a de Laval nozzle, i.e., an isolator and enters the combustor at a Mach number of 2. The combustor in leas a constant area cross section of 58 mm upto the end of the wedge like strut. This section is succeeded by a one-sided divergent section with the top wall diverging at an angle of 3 degrees with respect to the axis. It has a length of 242 mm. This section compensates for the expansion of the boundary layer. The dimensions of the combustor are 45 mm (width) and 300 mm (length) respectively. The height of the combustor at inlet is 50 mm and 68 mm at outlet. A wedge like strut is located in the central part of the combustion chamber at 35 mm downstream from the combustor inlet. The strut has a length of 32 mm and a wedge angle of 12 degree. A schematic of the combustor geometry is presented in Figure 1. The fuel is injected by the injectors located on the strut base parallel to the flow in the combustor. The initial conditions for the inlet air stream and the injected fuel are tabulated in Table 1.

![Combustor Schematic](image)

**Figure 1: Combustor Schematic.**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Air</th>
<th>Hydrogen</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach Number</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>Velocity (m/s)</td>
<td>730</td>
<td>1200</td>
</tr>
<tr>
<td>Static Temperature (K)</td>
<td>340</td>
<td>250</td>
</tr>
<tr>
<td>Static Pressure (bar)</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Stagnation Pressure (bar)</td>
<td>7.82</td>
<td>1.89</td>
</tr>
<tr>
<td>Density (kg/m³)</td>
<td>1.225</td>
<td>0.097</td>
</tr>
<tr>
<td>O₂ Mass Fraction</td>
<td>0.21</td>
<td>0</td>
</tr>
<tr>
<td>N₂ Mass Fraction</td>
<td>0.79</td>
<td>0</td>
</tr>
<tr>
<td>H₂ Mass Fraction</td>
<td>0</td>
<td>1</td>
</tr>
</tbody>
</table>

**Table 1: Inlet Conditions for Air and Hydrogen**

Assumptions

The analysis was conducted with two primary assumptions, that is, the flow has attained a steady state, and that the gases are compressible, obeying the ideal gas law.

COMPUTATIONAL METHODOLOGY

Ansys Workbench was used for meshing. Fluent was the primary CFD Solver and Ansys CFD-Post was used for post processing. Mesh quality was assessed through a mesh independence study as well as by maintaining a wall $Y+<1$. Three cases related to mesh size were chosen for the mesh independence study representing coarse, medium and fine mesh sizes. The validation of obtained results was conducted through graph comparisons as well as schlieren image comparisons of the original DLR research.
Meshing

Meshing was completed using Ansys Workbench. After progressive refinements, grids with 130,000, 600,000 and 2.2 million cells were chosen to simulate coarse, medium and fine cases for 2D analysis. Maximum Y+ at the bottom and top walls is less than 50. The mesh was refined near the walls to achieve Y+ in the range of 0-10 in the medium and fine cases.

Solver Setup

To simulate the flow, pressure based solver was used along with the coupled solution scheme. Turbulence was modelled after the RANS equations (4 equations) using the k-w SST (Shear Stress Transport) model. Combustion was modeled using the Eddy Dissipation (ED) model. This model correlated reaction rate with the time required for mixing of reactants at the molecular level. Ideal Gas assumption is once again invoked the evaluation of CP, and viscosity. The no-slip condition and a diabatic wall boundary condition are imposed and the equations for the flow are solved to the walls of the combustor. This model also assumes single step reaction chemistry to simplify calculations as well as reduce computation time.

RESULTS AND DISCUSSIONS

The results are validated qualitatively using experimental and numerical schlieren images. And quantitatively by drawing a contrast between the static pressure on the bottom wall, temperature along a y - profile and velocity on an x - profile along the combustion zone obtained from the analysis with the experimental data. The cases investigated are presented in detail in the following sections.

NO H2 Injection

In this section, a closed flow of air from the air inlet to the outlet is investigated. This investigation do not involve Hydrogen injection. Figure 2 depicts a contrast between the experimental schlieren and numerically computed schlieren. Due to the presence of the tip, oblique shocks are generated. These shocks propagate through the combustor till they encounter the walls of the combustor where, they are reflected back in to the flow field. These generated oblique shocks, reflected weaker shocks as well as the expansion fan generated at the bottom of the strut form a distinct pattern called shock train in the region downstream to the wedge. In addition, due to reflection of shocks, the boundary layers near the wall are affected, resulting in thickening of boundary walls and heightened wall temperatures. Since, the top wall is diverging, the shocks reflected off this surface possess a different shock angle relative to those reflected by the bottom wall. This leads to an unsymmetric shock train.

Figure 2: Experimental Schlieren (Left) vs Numerical Schlieren from Analysis (Right).

The validation of the results obtained have been presented in the form of graphs of Static Pressure at each node on the bottom wall, yprofile temperature and velocity along the combustion zone in Figures 3, 4 and 5.
Figure 3: Static Pressure along Bottom Wall.

Figure 4: Velocity along a line Parallel to x-axis Behind the Strut.

Figure 5: Static Temperature along a line Parallel to y-axis Behind the Strut.
Cold Flow

This section deals with the results of cold flow, i.e., air inlet along with Hydrogen injection but without combustion. The injection of hydrogen adds an extra complexity to the flow. Mixing now needs to be considered and given prominence as the molecular mass of hydrogen is significantly less than that of air. Figure 6 shows the density contour. Similar to the previous case, here too, due to the presence of the tip, oblique shocks are generated which are consequently reflected of the top and bottom walls of the combustor. A significant change from the current case relative to the previous case is due to the fact that reflected shocks passing through the wake of the wedge are slowed down considerably in the wake due to mixing of Hydrogen and Oxygen. This mixing leads to the slowing down or deceleration of the flow leading to weakening of the shocks and expansion fans. In addition, they are also deflected by the streams of Hydrogen that are injected by the injectors into the flow. At a distance away from the base of the strut where the streams of air and hydrogen are completely mixed, the flow is accelerated back to supersonic velocities and a behaviour similar to the previous case can be observed. A numerical schlieren could not be plotted for this due to the hydrogen injection which heavily affected the density gradient.

![Figure 6: Density Contour for Cold Flow.](image)

Reaction Flow

This section deals with the results of combustion of the injected hydrogen in the combustor. Combustion by itself is an interesting and complex problem of CFD. In the case of SCRAMJETs, this complexity is compounded by the fact that combustion is supersonic. There is also a new requirement to deal with products of combustion, energy release from the combustion and mixing of reaction products with left over reactants. Figure 7 shows the density contour plot. Similar to the previous cases, here too, due to the presence of the tip, oblique shocks are generated which are consequently reflected of the top and bottom walls of the combustor before interacting with the wake of the strut, where active combustion occurs. In addition, the reflected shock waves are deflected by the mixing reactants, hydrogen jets and combustion products. A numerical schlieren could not be plotted for this case due to complexities involved with combustion. Combustion has been simulated as a single step reaction using the ED model. In the original DLR Experiments, combustion has been simulated for H2 injection mass flow rates of 1.5, 2.0, and 2.5 g/s. However, in this investigation, only the 1.5 g/scase is simulated. In addition, to simulating the overhead injection system used by the DLR, the area above and below the injection plane was patched to temperatures of 600K. This acted as an ignition system and initiated combustion. As a validation for combustion, the OH images from the DLR research could have been used but since the reaction was simulated as a single step reaction, the numerical OH image could not be plotted.
The validation of the results obtained have been presented in graphs of Static Pressure along the bottom wall, y profile temperature and velocity along the combustion zone is shown in Figures 8, 9 and 10.

**Figure 7: Density Contour for Reaction Flow.**

**Figure 8: Static Temperature along a Line Parallel to y-axis Behind the Strut.**

**Figure 9: Velocity along a Line Parallel to x-axis Behind the Strut.**
Figure 10: Static Pressure along Bottom Wall.

In addition to the graphs provided to support validation, the following contour plots are presented to understand the combustion process intuitively.

Figure 11: Rate of Reaction Contour.

Figure 12: Static Pressure Contour.

Figure 13: Mach Number Contour.
Mesh Independence

Mesh Independence study was performed for 3 cases which varied from one another on the basis of cell size. The three cases correspond to this are coarse, medium and fine meshes. The coarse mesh contains 130k cells, the medium mesh contains 660k cells and the fine mesh contains 2.2 million cells. For this study, wall Y* score is chosen as the primary indicator for mesh independence. Figures 15, 16 and 17 details the Y* on the top, bottom and strut walls in the form of a histogram.

Figure 14: H₂ Mass Fraction Contour.

Figure 15: Y* Distribution at the Top Wall.
CONCLUSIONS

A numerical simulation of the DLR research was conducted for reacting, non-reacting and no fuel-injection cases for a Mach 2 scramjet combustor. In the non-reacting flow scenario, it was noticed that the static temperatures of the air was less than the self-ignition temperature of the H2-air mixture. Therefore, two zones near the H2 injector were patched to the ignition temperature of the mixture to act as a flame holder and initiate combustion in the reacting flow case. The
Interactions of shock waves in the wake of the strut were examined and the effects of presence of H$_2$ or a reacting mixing layer in the wake on the shock wave interactions were also studied. Irrespective of the case under examination, quantitative comparisons by means of comparing data plots of static pressure at each node of the bottom wall, a temperature profile as well as x-velocity profile were made. In addition, a mesh independence study was also carried out to ensure robustness of solution scheme used.

REFERENCES


