Three-dimensional Computational Analysis of Transverse Injection in a Supersonic Combustor

P Karthikeyan¹, G Sri Santhosh kumar², R Yuvaraj², and A Jeeva²

¹Assistant Professor, Department of Aeronautical Engineering, PAAVAI Engineering College, Namakkal
²UG scholar, Department of Aeronautical Engineering, PAAVAI Engineering College, Namakkal
karthikpaero@gmail.com¹

Abstract. The scramjet engine plays a promising role for a successful future hypersonic flight system. The performance of scramjet system depends on many factors like mixing rate of fuel and air, proper flame stabilization mechanism and suitable injection system. The flame stabilization is provided by means of introducing a Cavity along the wall of the combustor. The cavity effects on the performance of supersonic combustion will be studied here with the help of three-dimensional combustor CFD model. The reliability of the numerical results obtained by using the computational approach depends on the proper validation of the CFD code with their corresponding experimental results. The corresponding experimental results from the literature will be identified and used here for the validation of the numerical code. The final CFD results after its validation with the experimental results will be presented in terms of x-y plots and contours for the discussion.

1. Introduction
This chapter will provide a basic introduction to scramjet technology by presenting the definition, historical timeline, possible applications, and the current status of scramjet engines.

1.1 Definition of a Scramjet Engine
A SCRAMJET engine shown in figure 1 and 2 (which stands for Supersonic Combustion RAMJET) is a major development in air breathing propulsion system where combustion occurs in supersonic mode. The supersonic combustion ramjet or scramjet engine provides high thrust and low weight for hypersonic flight vehicles. Unlike a conventional turbojet engine or ramjet engine, a scramjet engine has no complex moving parts. It consists of a diffusor at inlet, nozzle at the exit and a combustor having a fuel injector, and flame holder. The combustion chamber consists of different types of fuel injection system (transverse, parallel or angled injection scheme), flame stabilization mechanism and fuel ignition for the sustained combustion
When a scramjet engine mounted on a high-speed aircraft, while moving forward, large quantities of air flow are drawn inside the engine through the inlet. The incoming air gets decelerated when passing through the isolator region, and the dynamic pressure gets converted into a higher static pressure [2]. This high pressure, also called as ram effect facilitates in adding the heat to the incoming air inside the combustion chamber, without the need for having the separate compressor system. In the combustion chamber, fuel is properly mixed with the air at a molecular level for proper ignition. Flame holders in the combustor help in stabilizing the process of supersonic combustion. While burning occurs in subsonic mode in the ramjet engine (M<1), it occurs completely in supersonic mode (M>1) in the scramjet engine. After the combustion chamber, the hot exhaust gas passes through a hypersonic nozzle, where the flow is accelerated to a high Mach number.

2. Methodology
This computational analysis is done with respect to the following methodologies. The CFD code used for the present analysis, need to be properly validated either with the results obtained by conducting one’s own experiments or by referring the experimental results available in the open literature. The second case is followed here as conducting one’s own experiments is highly budgeted and huge resources are required. Hence it is decided to carry out the numerical study of scramjet engine as shown in figure 3 and 4.
3. CFD Software Code

CFD stands for Computational Fluid Dynamics. Figure 5 shows it involves obtaining the solutions to the problems involving fluid mechanics, heat transfer and mass transfer using the computer.

3.1 Basic Mathematical Equations used in CFD Code

There are four natural laws that allow the determination of the unknowns of the flow problem:

- Continuity Equation (Conservation of Mass)
- Newton's 2nd Law (Conservation of Momentum)
- First Law of Thermodynamics (Conservation of Energy)
- Conservation of a species (Applicable for Combustion study)

The equations are the expression of the physical laws which must hold everywhere in the flow that is at every point P in the flow. The equations are applied in each and every grid or node points, to obtain solution by iterations.

4. CFD Pre-Processor

4.1 GAMBIT -Modelling

GAMBIT is a geometry and mesh generation software, generally used with FLUENT. GAMBIT's combination of CAD interoperability, geometry clean-up, decomposition and meshing tools results in one of the easiest, fastest and most direct pre-processing paths from CAD to quality CFD mesh.

4.2 Grid Studies

Grid is the same as Mesh and the function of both is to divide the flow domain (or volume) into smaller regions where the mathematical equations are applied. This process is called discretization. The size and shape are constrained to reflect flow physics of interest in that location as well as the applicability of the numerical technique.
It uses the same equations expressed in a finite volume formulation. If we were to actually solve for every point in the flow domain then we probably will never solve the problem in our lifetime. Rather than applying the equations at point P, the equations are applied to a small volume enclosing the Point P.

This is a finite volume Formulation. These volumes are the grids or meshes. Fluent is a finite volume solver. The mathematical equations are written over this volume and converted to suitable algebraic equation as shown in figure 5 and 6.

### 5. Geometry Set-up

|                      |   |
|----------------------|---|
| **Table 1. Combustor geometry** |   |
| Injection Surface Diameter (d) | 1.93 mm |
| Combustor Chamber Height (h)    | 21.29 mm |
| Combustor Section Width (W)     | 30.48 mm |
| Combustor Section Length (l)    | 40.28d  |
| Cavity Depth (H)                | 3.1845 mm |
6. Flow Condition

Figure 8 shows the numerical calculation is applied to a three-dimensional combustor with fuel injection at upstream of the cavity for the conditions (Based on test conditions in McDaniel [1] experiment and computational conditions is presented in below table. The size of the injector is 1.93 mm in diameter. The length of the combustor is 40.28d is required here, in order to clearly visualize the vortex formation in the near field and far field of the combustor.

|                      |            |
|----------------------|------------|
| Length before the 1st injection hole | 4.9d       |
| Length before the 2nd injection hole | 11.5d      |

| Boundary conditions for computational domain |
|---------------------------------------------|
| Free stream Mach no. of air (M)             | 2.0        |
| Static pressure (P)                         | 274000 Pa  |
| Static Temperature (T)                      | 300 K      |
| Mach no. at injector exit (Minj)            | 1.25       |
| Static pressure at injector exit (Pinj)      | 101323 Pa  |
| Temperature at injector exit (Tinj)          | 300 K      |
| Size of injector exit (d)                   | 1.93 mm    |

7. Results And Discussion

Figure 9 shows the computational results obtained from a 3D backward step combustor is validated with the experimental results of McDaniel et al (1991) was carried out in this chapter. The profiles of u velocity, v velocity, temperature and pressure distribution along the axial length of the combustion chamber are presented for both computational and experimental results. The contours obtained for different flow variables at different x/d coordinates are also presented in the comparison [3-4].

![Figure 7. Computational model of backward step combustor (Meshed Geometry)](image-url)
Above Figure highlights the meshed geometry of 3D backward step combustor created by CFD software. The mesh is densely packed for the region near to wall as well near to the fuel injector as shown in figure 8.

![Figure 8](image)

**Figure 8.** Experimental set up by McDaniel (1991)

Figure 9 and 10 shows The experimental model analysed by McDaniel in 1991 is presented in Figure above which is same as the geometry of numerical model considered for the current investigation.

![Figure 9](image)

**Figure 9.** Centreline Pressure -experiment

![Figure 10](image)

**Figure 10.** Pressure contour along the XY plane passing center line- computational results

Figure 11 and 12 shows The pressure contour along the XY plane passing through center line is compared to both experimental results and computational results and the results are well matched with each other.

![Figure 11](image)

![Figure 12](image)
Figure 1. Centreline Temperature - experiment

Figure 2. Temperature contour along the XY plane passing center line- computational results

Figure 3. Validation of computational results (Left side) with experimental results (Right side) – Mass fraction contour along YZ plane passing through following X-coordinates, x/d= -3.0, x/d=0.0, x/d=3.1 & x/d=6.6

The mass fraction contour along the XY plane passing through centreline is compared to both experimental results and computational results. Both the contours are similar to each other, especially the penetration height of fuel in both the figures found to be more similar. The mass fraction contour along YZ plane passing through different x-coordinates, x/d= -3.0, x/d=0.0, x/d=3.1 & x/d=6.6 are validated for both experimental results and computational results.

8. Conclusion
3D Geometrical model was created using CFD as same shape as the experimental setup. Same simulation parameters/Boundary conditions are applied for the comparison of CFD and experimental results. The pressure, temperature and mass fractions contours are well matching with the corresponding experimental results. This proved our main objective of CFD validation with experimental results and this CFD tools can be further used to study the mixing efficiency of fuel and air of scramjet engine.

References
[1] McDaniel J Fletcher D Hartfield R and Hollo S, 1991, Staged Transverse injection into Mach 2 flow behind a Rearward facing step AIAA Paper 91-5071.
[2] K.M. Pandey and A.P. Singh, 2012, Numerical Simulation of Combustion chamber without cavity at Mach 3.12 pp134-141.
[3] N. N. Fedorova, M. A. Goldfield and Yu. V. Zakharova, 2019, Experimental and numerical analysis of hydrogen jet autoignition in backward-facing-step-stabilized model scramjet combustor pp 01-06
[4] Ronald S. Fry, 2004, A Century of Ramjet Propulsion Technology Evolution pp 27-57
[5] Buggeln, R., Shamroth, S., Lampson, A., & Crowell, P. (1994). Three-dimensional (3-D) Navier-Stokes analysis of the mixing and power extraction in a supersonic chemical oxygen
iodine laser (COIL) with transverse I2 injection. 25th Plasmadynamics and Lasers Conference. doi:10.2514/6.1994-2435

[6] Rouzbar, R., & Eyi, S. (2015). Three Dimensional Flow Analysis of a Cavity-Based Scramjet Combustor. 22nd AIAA Computational Fluid Dynamics Conference. doi:10.2514/6.2015-3212

[7] S, D., & H, A. (2019). AODV Route Discovery and Route Maintenance in MANETs. 2019 5th International Conference on Advanced Computing & Communication Systems (ICACCS). doi:10.1109/icacccs.2019.8728456

[8] H. Anandakumar and K. Umamaheswari, An Efficient Optimized Handover in Cognitive Radio Networks using Cooperative Spectrum Sensing, Intelligent Automation & Soft Computing, pp. 1–8, Sep. 2017. doi:10.1080/10798587.2017.1364931

[9] Alhussan, K., & Garris, C. (2005). Computational Analysis of Flow Inside a Diffuser of Three-Dimensional Supersonic Non-Steady Ejectors. Fluids Engineering. doi:10.1115/imece2005-80843

[10] Malo-Molina, F. J., Gaitonde, D. V., Ebrahimi, H. B., & Ruffin, S. M. (2010). Three-Dimensional Analysis of a Supersonic Combustor Coupled to Innovative Inward-Turning Inlets. AIAA Journal, 48(3), 572–582. doi:10.2514/1.43646