“Mathematical Modelling and Numerical Investigation of Scram Jet Engine with Multistage Wall Injector Using Fluent”

VIJAY KUMAR TIWDE¹
M. Tech Scholar Mechanical Engg. Dept. VIST BHOPAL
VARDAN SINGH NAYAK²
Asst. Professor Mechanical Engg. Dept. VIST BHOPAL

ABSTRACT

Fuel injection technology in Scramjet engines is an area that is still evolving today. The fuel used by a scramjet is usually liquid or gas. Efficient combustion requires the mixing of fuel and air in approximately stoichiometric ratios. The main problem of scramjet fuel injection is that the airflow is quite fast, meaning that there is a minimum time for fuel to mix with air and ignite to produce thrust (essentially milliseconds). Main fuel used for combustion is hydrogen. Hydrocarbons present more challenges than hydrogen due to greater ignition delays and the need for more advanced mixing techniques. Increasing the mixture, and thus reducing the length of the comb, is an important aspect in designing a scramjet engine. Several techniques are used today for fuel injection in a Scramjet engine.

In Present study we Investigate Scramjet engine with multi stage wall injector, various mass flow rate and micro gravity effect by the CFD Simulation using ANSYS as a CAE tool. Simulation results show an excellent stable flow phenomenon of the complete system which is the desirable stream function for the results accuracy. Present study shows that in MSWI (Multistage wall injector), ALPHA angle which is in between side wall and injecting surface play a major role in combustion. In present study we have used 30°, 45° and 60° degree for the wall injector and simulation results shows that as we increase the angle combustion efficiency is increased and emission is decreased for the system.

KEY WORDS: Fuel injection, Mach number, Scramjet, Thrust, SST model, Flame holder, MSWI, and Microgravity etc.

1. INTRODUCTION

1.1 Scramjet Engines

The Supersonic Combustion Ramjet (SCRAMJET) motor has been seen as the most encouraging air-breathing impetus framework for the hypersonic flight (Mach number above 5). Lately, innovative work of scramjet motor has advanced the investigation of ignition in supersonic stream. Broad examination is being completed over the world for understanding the scramjet innovation with hydrogen fuel with huge consideration concentrated on different eras of space launchers and worldwide quick response observation missions. In any case, application for the scramjet idea utilizing high warmth sink and hydrogen fills offers essentially upgraded mission potential for future military strategic rockets. Scramjet being an air-breathing motor. The execution of the rocket framework in light of the scramjet drive is imagined to improve the payload weight and rocket range. Supersonic burning ramjet motor for an air breathing drive framework has been identified and exhibited by USA on ground and in flight. Real center towards enhancing the scramjet combustor execution is delivered to the successful blending of fuel and air. Because of highly active vitality of the air stream, cross stream blending in the middle of fuel and air is extremely troublesome. Henceforth unique liquid instrument is required to accomplish complete blending.

In the course of recent years, impressive exertion has been coordinated with the improvement of a utilitarian air breathing motor. The most suitable motor to be taken into account is the Supersonic Combustion Ramjet motor (or Scramjet motor). Extensive scale tasks to incorporate the Scramjet motor outline have incorporated the British Aerospace HOTOL (Horizontal Take-Off and Landing) venture and the NASA NASP (National Aerospace Project) venture [19]. Fuel and air must blended at sub-atomic level in the adjacent field of fuel infusion. To affect such components in supersonic inflows, the fuel injectors are expected to be basically formed causing less stream misfortunes. At that point, fuel infusion likewise are expected to be done reasonably to use
the stream field produced by fuel injectors minus all potential limitations degree. Current examinations are focused around the impact of fuel injection plan on a model scramjet combustor execution. The system requires the arrangement of physical impediments in the combustor to give stream insightful vortices that improve the blending of fuel and air. Such methodologies are utilization of in reverse confronting step and inclines [24]. In reverse confronting step produces distribution zone that contains hot gasses in it and serves as a constant ignition source. On the other hand, the burden of in reverse confronting step is of moderately high stagnation weight misfortune. Inclines at supersonic stream create hub vortices which help in detailed scale blending of fuel with air. Collaboration of stuns created by slopes with the fuel stream produces boro-center torque at the air and fluid fuel interface, upgrading small scale blending. Holes utilize a coordinated methodology as fuel injector and fire holder. This was first outlined and utilized by CIAM (Central Institute of Aviation Motors) in Moscow in a joint Russian/French double mode scramjet flight-test [25]. Distribution zones present in pits build the habituation time of the flammable blend and thus are better contender for fire holding.

2. LITERATURE
1.K.M.Pandey, Member IACSIT and A.P. Singh - Survey of experimental and numerical studies have been done for different complex flow fields with respect to different aspect by mixing of different types of fuel and mixture with high speed flows in the combustion chamber. In this field many researchers worked to develop a configuration giving efficient mixing and combustion, also meeting the requirements of flame holding and completion of combustion with sufficient stabilization in the flow field. The main attention is paid to the local intensity of heat release, which determines, together with the duct geometry, techniques for flame initiation and stabilization, injection techniques and quality of mixing the fuel with oxidizer, the gas-dynamic flow regime. From the survey it concluded that some area in which more attention to be paid like total pressure loss in combustion chamber and design a contour to make the flow at Mach number 2 for various higher Mach numbers for supersonic combustion and analyze the flow properties in combustion chamber and its effect by using the variety of fuel with the help of CFD tool. 2. K.M.Pandey and T.Sivasakthivel - As one of the most promising propulsive systems in the future, the scramjet engine has drawn the attention of many researchers. The two-dimensional coupled implicit NS equations, the standard k-ε turbulence model and the finite-rate/eddy-dissipation reaction model have been applied to numerically simulate the flow field of the hydrogen fueled scramjet combustor with a planer strut flame holder under two different working conditions, namely, cold flow and engine ignition. The obtained results show that the numerical method used in this paper is suitable to simulate the flow field of the scramjet combustor. The static pressure distribution along the top and bottom walls for the case under the condition of engine ignition is much higher than that for the case under the condition of cold flow. There are three clear pressure rises on the top and bottom walls of the scramjet combustor. The eddy generated in the strut acts as a flame holder in the combustor, and it can prolong the residence time of the mixture in the supersonic flow.3-DEEPU, Mukandan N.1*, GOKHALE, Sadanand. S.2, and JAYARAJ, Simon3- Numerical simulation of supersonic combustion of hydrogen in air has been done using point implicit finite volume method. This method treats all chemical species terms implicitly and all other terms explicitly. Solver is based on the solution of unsteady, compressible, turbulent Navier-Stokes equations, using Unstructured Finite Volume Method (UFVM) incorporating RNG based k-ε two equation model and time integration using three stage Runge-Kutta method. Reaction of hydrogen with air is modeled using an eight-step reaction mechanism. The preconditioning has found to be effective in overcoming the stiffness in chemically reacting flows. The method is validated against standard experiments for CFD code validation. The predicted values of temperature and species production were in good agreement with experimental results. The code is used to simulate the combustion of hydrogen injected to the wake region formed by a wedge shaped strut 4-K.M.Pandey and T.Sivasakthivel - In this study, k-ε model has been used to examine supersonic flow in a model scramjet combustor. The configuration used is similar to the DLR (German Aerospace Center) scramjet model and it is consists of a one-sided divergent channel with a wedge-shaped flame holder at the base of which hydrogen is injected. Here, we investigate supersonic cold flow with hydrogen injection. For the purpose of validation, the k-s results are compared with experimental data for temperature at the bottom wall. In addition, qualitative comparisons are also made between predicted and measured shadowgraph images. The k-ε computations are capable of predicting cold flow simulations well and good 5-L.Hariramakrishnan1, K.Nehru2, T.Sangeetha3-A computational analysis of a complete scramjet demonstrator model has been initiated. The computational results will take place under real flight conditions at a hypersonic speed. Prior to these tests, a numerical analysis of the performance of the demonstrator geometry is conducted. In this current paper, the results of the performance analysis for the newly designed 3-D inlet employing a single outer compression ramp as well as side wall compression are to be disessed. It is shown that the intake is able to generate flow conditions required for stable supersonic combustion using a central strut injector.
3. OBJECTIVE OF THE STUDY
Mixing, ignition and flame holding in combustor, ground test facilities and numerical simulation of Scramjet engine are the critical challenges in the development of scramjet engine. Among the critical components of the scramjet engine, the combustor presents the most formidable problems. The complex phenomenon of supersonic combustion involves turbulent mixing, shock interaction and heat release in supersonic flow. The flow field within the combustor of scramjet engine is very complex and poses a considerable challenge in design and development of a supersonic combustor with an optimized geometry. Such combustor shall promote sufficient mixing of the fuel and air so that the desired chemical reaction and thus heat release can occur within the residence time of the fuel-air mixture. In order to accomplish this task, it requires a clear understanding of fuel injection processes and thorough knowledge of the processes governing supersonic mixing and combustion as well as the factors, which affects the losses within the combustor. The designer shall keep in mind the following goals namely
1. Good and rapid fuel air mixing
2. Minimization of total pressure loss
3. High combustion efficiency.

OBJECTIVE
In current study our main objective is to improve the combustion efficiency of Scramjet combustor by rapid mixing and minimize the total pressure loss with geometrical and operational conditions. To achieve our goal following steps will be taken to investigate the system-

4. METHODOLOGY
1 Basic Steps to Perform CFD Analysis
4.1.1 Pre-processing: CAD Modeling: Creation of CAD Model by using CAD modeling tools for creating the geometry of the part/assembly of which you want to perform FEA. CAD model may be 2D or 3d.
4.1.2 Meshing: Meshing is a critical operation in CFD. In this operation, the CAD geometry is discretized into large numbers of small element and nodes. The arrangement of nodes and element in space in a proper manner is called mesh. The analysis accuracy and duration depends on the mesh size and orientations. With the increase in mesh size (increasing no. of element), the CFD analysis speed decrease but the accuracy increases.
4.1.3 Type of Solver: Choose the solver for the problem from Pressure Based and density based solver.
4.1.4 Physical model: Choose the required physical model for the problem i.e. laminar, turbulent, energy, multi-phase, etc.
4.1.5 Material Property: Choose the Material property of flowing fluid.
4.1.6 Boundary Condition: Define the desired boundary condition for the problem i.e. temperature, velocity, mass flow rate, heat flux etc.
4.2 SOLUTION
   • Solution Method: Choose the Solution method to solve the problem i.e. First order, second order
   • Solution Initialization: Initialize the solution to get the initial solution for the problem.
   • Run Solution: Run the solution by giving no of iteration for solution to converge.
4.3 Post Processing: For viewing and interpretation of Result. The result can be viewed in various formats: graph, value, animation etc.
4.4 CFD METHOD APPLIED
The model was simulated and the required geometry configurations were pre-processed in ANSYS 19.2. This following section illustrates the method used in the CFD simulations in this particular study.

STEP I GEOMETRY OR MODEL FORMATION
The study focuses on the Analysis of Hydrogen Combustion Using ANSYS Fluent for the simulations. The generation of the model by using ANSYS shown below:

Figure 4.1 Schematic diagrams with dimensions
STEP 2

MESH FILE – To Be Meshed

Generated Mesh Model in the ANSYS shows in 4.3 and 4.4

Figure 4.2 CAD Model of 2d Geometry

Figure 4.3 Mesh Model of 2d Geometry

Figure 4.4 Close-up view of Mesh Model

Figure 4.4 Mesh Refinement Model of 2d axis symmetry Geometry

Figure 4.5 Mesh Refinement (close-up) Model of 2d axis symmetry Geometry
Mesh Type: grid meshing  
Element Edge Length = 2.95e-004 m  
No. of Nodes = 10051  
No. of Element = 50042  

Fluent setup: After mesh generation define the following setup in the Ansys fluent.  
Problem Type: 2D axisymmetric  
Type of Solver: Pressure-based solver.  
Physical model: Viscous: K, epsilon two equation turbulence model.  
Use P1, Finite rate/ Eddy dissipation model  
Material Property: Flowing fluid is air  
Density of air = 1.225 kg/m³  
Viscosity = 1.7894e-05  
Boundary Condition:  
Operating Condition: Pressure = 101325 Pa  

<table>
<thead>
<tr>
<th>Variables</th>
<th>Air</th>
<th>H₂</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ma</td>
<td>3.0</td>
<td>1.0</td>
</tr>
<tr>
<td>U (m/s)</td>
<td>750</td>
<td>1300</td>
</tr>
<tr>
<td>T (K)</td>
<td>340</td>
<td>250</td>
</tr>
<tr>
<td>P (Pa)</td>
<td>101325</td>
<td>101325</td>
</tr>
<tr>
<td>Density</td>
<td>1.002</td>
<td>0.097</td>
</tr>
<tr>
<td>YO₂</td>
<td>0.232</td>
<td>0</td>
</tr>
<tr>
<td>YH₂</td>
<td>0.736</td>
<td>0</td>
</tr>
<tr>
<td>YH₂O</td>
<td>0.032</td>
<td>0</td>
</tr>
<tr>
<td>YH₂</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>Mass flow rate (kg/s)</td>
<td>2.5</td>
<td>0.02</td>
</tr>
</tbody>
</table>

2. Solution:  
Solution Method:  
Pressure- velocity coupling – Scheme -SIMPLE  
Pressure – Standard  
Momentum – Second order  
Turbulent Kinetic Energy (k) Second order  
Turbulent Dissipation Rate (ε) Second order  
Solution Initialization: Initialized the solution to get the initial solution for the problem.  
Run Solution: Run the solution by giving 5000 no of iteration for solution to converge.  
Post Processing: For viewing and interpretation of Result. The result can be viewed in various formats: graph, value, animation etc.

5. RESULT AND DISCUSSION  
5.1 GENERAL  
The analysis of Scram jet engine with multi stage wall injector at different alpha angle, mass flow rate and gravity is done based on function of stream function, mass fraction of H₂O, mass fraction of O₂, mass fraction of H₂, Total energy, total temperature, total pressure, Mass fraction of CO2, NO2 and the result is discussed here below.
5.2.1 STEAM FUNCTION: Some of the stream function flow pattern which shows the stability of the system:

Fig-5.1 Contours of Stream Function

Fig-5.2 Contours of Stream Function (Surface function)

Fig-5.3 Contours of Stream Function close view

Fig-5.4 Contours of Stream Function close view
As shown in the figure, the stream function is minimum at the surfaces and its maximum at the surrounding along the axis. As shown, the stream function contours are maximum at the time of fuel injection.

5.2.2 VELOCITY MAGNITUDE (SYSTEM)

![Fig. 5.5 Contour of velocity magnitude @ case1](image1)

![Fig. 5.6 Contour of velocity magnitude @ case2](image2)

![Fig. 5.7 Contour of velocity magnitude @ case3](image3)
Fig. 5.8 Contour of velocity magnitude @ case4

Fig. 5.9 Contour of velocity magnitude @ case5

Fig. 5.10 Contour of velocity magnitude @ case6

Fig. 5.11 Contour of velocity magnitude @ case7
5.2.3 TOTAL ENERGY:

Fig. 5.12 Contour of velocity magnitude @ case8

Fig- 5.14 Contours of Total Energy @ case 1

Fig- 5.15 Contours of Total Energy @ case 2

Fig- 5.16 Contours of Total Energy @ case 3
From figure we can see the total energy changes are same in axial direction and which is near same after injection of \( h_2 \) from wall injector. We find maximum energy of \( 3.26e+09 \) j/kg at center after injection.

5.2.4 TOTAL TEMPERATURE:
Fig-5.22 Contours of Total Temperature @case1

Fig-5.23. Contours of Total Temperature @case2

Fig-5.24 Contours of Total Temperature @case3

Fig-5.25 Contours of Total Temperature @case4
The total temperature is increased at the time of injection and near the surface of injection but its temperature is maximum in near the region of injector after fuel injection done.
5.2.5 TOTAL PRESSURE:

Fig-5.30 Contours of Total Pressure@ case1

Fig-5.31 Contours of Total Pressure@ case2

Fig-5.32 Contours of Total Pressure@ case3

Fig-5.33 Contours of Total Pressure@ cas4
Fig-5.34 Contours of Total Pressure@ case5

Fig-5.35 Contours of Total Pressure@ case6

Fig-5.36 Contours of Total Pressure@ case7

Fig-5.37 Contours of Total Pressure@ case8
COx and NOx EMISSIONS

5.2.6 MASS FRACTION OF CO₂

Fig-5.38 Mass fraction of CO₂ @ case1

Fig-5.39 Mass fraction of CO₂ @ case2

Fig-5.41 Mass fraction of CO₂ @ case4

Fig-5.42 Mass fraction of CO₂ @ case5
5.2.7 MASS FRACTION OF NO₂

Fig-5.46 Mass fraction of NO₂ @ case1
Fig-5.47 Mass fraction of NO₂ @ case2

Fig-5.48 Mass fraction of NO₂ @ case3

Fig-5.49 Mass fraction of NO₂ @ case4

Fig-5.50 Mass fraction of NO₂ @ case5
6. CONCLUSION

CFD Simulation results show an excellent stable flow phenomenon of the complete system which is the desirable stream function for the results accuracy. From this study we can conclude that this type of multi-stage wall injector can solve the recent problem of scramjet combusters in use and the current analysis shows solutions about steady flow. In the tangent velocity figure we can see the stability of the flow which is the major problem with the Planer Strut injector, which provides a limit in the Mach no of the engine, but can give continuous flow and combustion through flight. This work can improve the stability of combustion and give a solution to the Scramjet research vehicle in terms of Mach no. Investigation of the complete system by the CFD results may conclude the following points:

1. Present study shows that in MSWI (Multi stage wall injector), ALPHA angle which is in between side wall and injecting surface is play a major role in combustion. In present study we have used 30, 45 and 60 degree for the wall injector and simulation results shows that as we increase the angle combustion efficiency is increased and emission is decreased for the system.

2. Mass flow rate of the hydrogen fuel is given a significant improvement in temperature and pressure and reduce the emissions rate which is a desirable for the SCRAMJET engine.

3. Velocity magnitude is continue increasing at a moderate rate as we increase the mass flow rate and angle (alpha) and reaching at its maximum range of 115m/s in case of 60 degree , maximum mass flow rate and simple gravity.

4. Finally we have also analyzed the complete combustion system for the various microgravity values like g/2, g/4 and g/6 and results shows that temperature, pressure, Total energy and emissions has a very negligible effect because the system is always in a supersonic mode.
7. REFERENCES


