Mathematical Modeling and Investigation of Mixing Characteristics in Combustion System of Jet Engine using CFD

[1]Saddam Hussain, [2]Chandra Pratap Shukla
[1]M.Tech Scholar, Vidhyapeeth Institute of Science and Technology, Bhopal, India
[2]Assistant Professor, Vidhyapeeth Institute of Science and Technology, Bhopal, India
cps23232@gmail.com

ABSTRACT
The combustor poses the most challenging issues of the scramjet engine's three essential parts. Turbulence mixing, shock interaction, and heat release in supersonic flow are all components of the intricate process of supersonic combustion. The flow environment inside the scramjet engine's combustor is very complex, making it challenging to design and create a supersonic combustor with an ideal geometry. Such a combustor will encourage a suitable combination of the fuel and air during the residence duration of the fuel-air mixture, allowing for the intended chemical reaction and, as a result, the production of heat. It takes a solid grasp of fuel injection procedures, in-depth familiarity with the laws regulating supersonic mixing and combustion, and an awareness of the variables influencing combustor losses to do this assignment.
Improved fuel-air mixing and flame stabilization are necessary for the development of scramjet engines with acceptable performance. In the current study, the impact of the fuel injector's geometrical design on fuel-air mixture and flame stabilization is examined statistically. A numerical tool is developed using the Reynolds Average Navier-Stokes equations. A chemical kinetics model is used to determine a finite rate of chemical reactions. The basic issue of supersonic mixing layers, modeling attempts for high-speed combustion, and practical computations of real scramjet combustors are among the subjects discussed. In this study, we examine two distinct fuel injector types based on flow field stability, shock wave form, and combustion rate. According to simulations, the Pylon injector's quick air-fuel mixing raises both the overall temperature and energy compared to the central wedge-shaped injector. In the case of a pylon injector, shock wave absorption capability and flow field stability are balanced according to stream line flow.
Keywords: Scramjet Engines; Computational Fluid Dynamics; Chemically- Reacting Flows; Reynolds-Averaged Navier-Stocks; Turbulent Flows; Supersonic Combustion, Central strut injector, Pylon Injector.

1. INTRODUCTION

The most practical air-breathing propellant design for hypersonic flight (Mach number more than 5) has been thought to be the Supersonic Combustion Ramjet (SCRAMJET) motor. The study of ignition in supersonic streams has recently advanced thanks to the creative work of scramjet motors. A thorough analysis of the scramjet invention using hydrogen fuel is being conducted globally, with a great deal of attention paid to various space launcher eras and global quick reaction observation missions. Regardless, the use of hydrogen fills and a large heat sink in the scramjet concept gives basically improved mission potential for upcoming military strategic rockets. Scramjet is a motor that breathes air. The payload's weight and rocket range are expected to be increased by the implementation of the rocket structure in light of the scramjet motor. The United States of America has discovered and demonstrated a supersonic burning ramjet motor for an air breathing drive system both on the ground and in flight. Fuel and air are successfully blended with a real focus on improving the scramjet combustor's performance. Cross-stream mixing between fuel and air is very problematic due to the air stream's tremendously active liveliness. From now on, a special liquid tool is needed to achieve full blending.

The scramjet is composed of three basic components:

A combustor, which produces heat by smoldering vaporous fuel with barometric oxygen, a focalizing bay, which packs and slows down approaching air, and a wandering spout, which accelerates the heated air to provide push. Very different from a typical simple motor, such as a turbojet or turbofan motor, a scramjet packs the air by using the attainable velocity of the flying machine moving through the air rather than by using pivoting, fan-like segments. Thus, a scramjet doesn't need any moving parts.

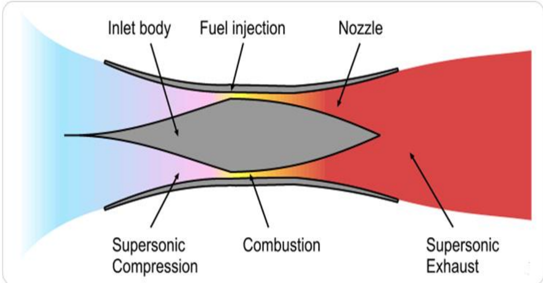

Fig.1.1- Basic Components of Scramjet

2. LITERATURE REVIEW

1.K.M.Pandey, *Member IACSIT* and A.P. Singh - An overview of high-speed combustion chamber flows combined with different fuel blends and types to conduct experimental and computational study for a variety of complex flow fields in different ways. In an effort to achieve the standards for flame retention and combustion complete with sufficient flow field stability, numerous researchers have tried to develop a design that provides efficient mixing and combustion. The main emphasis is on the local heat discharge intensity, which affects the gas-dynamic flow regime along with the duct shape, flame initiation and stabilization processes, injection methods, and fuel-oxidizer mixing quality. According to the survey, some areas require more attention, such as the combustion chamber's overall pressure loss, the contour that produces a flow at Mach 2 for a range of considerably greater Mach values for supersonic combustion, and the assessment of the combustion chamber's flow characteristics and their effects using a CFD tool and a variety of fuels**.**

2. K.M.Pandey and T.Sivasakthivel Since the scramjet engine is one of the most potential future propulsion technologies, many academics have an interest in it. The flow field of the hydrogen-fueled scramjet combustor with a planer strut flame holder has been numerically simulated under two distinct operating conditions using a finite-rate/eddy-dissipation reaction model, the standard k-ε turbulence model, and the two-dimensional paired implicit NS equations: engine ignition and cold flow. The outcomes show that the numerical approach employed in this study is suitable for simulating the scramjet combustor's flow field. When engine ignition is present, static pressure is dispersed throughout the upper and lower walls considerably more evenly than when cold flow is present. The scramjet combustor's top and bottom walls have three distinct pressure increases. The eddy generated in the strut, which acts as a flame holder in the combustor, can prolong the mixture's residence time in the supersonic flow.

3-DEEPU, Mukundan N.1*, GOKHALE, Sadanand. S.2, and JAYARAJ, Simon3- The point implicit finite volume method has been used to numerically model supersonic hydrogen combustion in air. In this way, all chemical species-related terms are addressed implicitly, while every other term are treated explicitly. The foundation of the solver is the Unstructured Finite Volume Method (UFVM), which incorporates a three-stage Runge-Kutta method for time integration and a RNG-based κ-ε two equation model for solving unsteady, compressible, turbulent Navier-Stokes equations. An eight-step mechanism of reaction is used to represent the chemical reaction of hydrogen with air.

3. OBJECTIVE OF THE STUDY:

This study shows how liquid fuel evaporates and burns through the use of the dispersed phase modeling capability to compute paired gas flow and liquid spray physics. The burning of the evaporated fuel is predicted using the equilibrium chemistry model of mixture fraction/probability density function (PDF). Through the solution of a transport equation for a single conserved scalar, the mixture-fraction/PDF modeling approach enables you to model non-premixed turbulent combustion. Using the premise of equilibrium chemistry, the concentrations of several chemical species, such as radicals and intermediate species, may be inferred from the expected mixture fraction. A chemical database provides access to the species' property information. A PDF is used to model the interaction between turbulence and chemistry.

Comparing the various fuel injector types in a scramjet combustor in a supersonic model is our primary goal in this work. The fuel used in this investigation is hydrogen, and two different types of fuel injectors—central strut wedge and pylon injector—are employed. FLUENT14.5 will do the simulation. Based on simulative outcomes, such as temperature, pressure, velocity variation, turbulent kinetic energy, total energy, etc., comparisons will be made. The CFD model (flow pattern) will additionally offer information regarding the model's stability, including whether it is stable or not.

4. METHODOLOGY

Basic Steps to perform CFD Analysis:

1. Preprocessing:

- **CAD Modeling:** CAD Model Creation: Utilize CAD modeling tools to create the geometry of the item or assembly that you wish to do FEA on. A 2D or 3D CAD model is possible.
- **Meshing:** In CFD, meshing is an essential function. The CAD geometry is divided into several tiny elements and nodes during this process. Mesh refers to the appropriate spatial organization of nodes and elements. The mesh size and orientations affect the analysis's duration and accuracy. The speed of the CFD analysis decreases but the accuracy increases as the mesh size (number of elements) increases.
- **Type of Solver:** Select the pressure-based or density-based solver for the problem. Select the appropriate physical model, such as laminar, turbulent, energy, multiphase, etc., for the task**.**
- **Material Property**: Select the fluid's material attribute.
- **Boundary Condition:** Specify the problem's desired boundary conditions, such as temperature, heat flux, mass flow rate, and velocity.

2. Solution:

- **Solution Method:** To solve the problem, select the solution method (first order, second order, etc.)
- **Solution Initialization:** Initialized the solution to obtain the problem's original solution.
- **Run Solution:** Run the solution with a specified number of iterations to ensure convergence**.**

3. Post processing.

- **Post Processing:** For seeing and evaluating the result. The outcome is available in a number of ways, including graph, value, animation, etc.

CFD Analysis of hydrogen combustion using Ansys Fluent

1. **Preprocessing:**

CAD Model: Generation of 2d axisymmetric geometry in Fluent.

CFD METHOD APPLIED

STEP I GEOMETRY OR MODEL FORMATION

ANSYS's assessment of hydrogen combustion. The primary focus of the study is fluent for simulations. The model was created with ANSYS and is shown below:

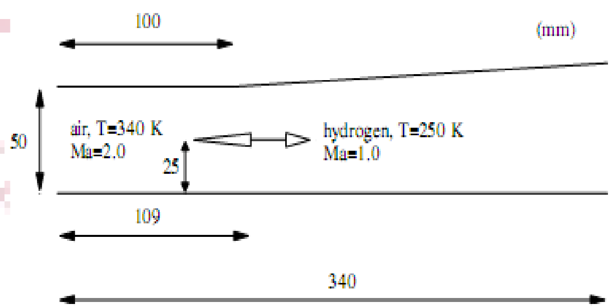

Figure-4.1: Reference model detail

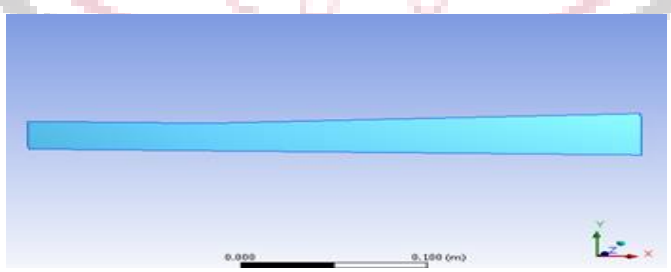

Figure 4.2 CAD Model of 2d Geometry

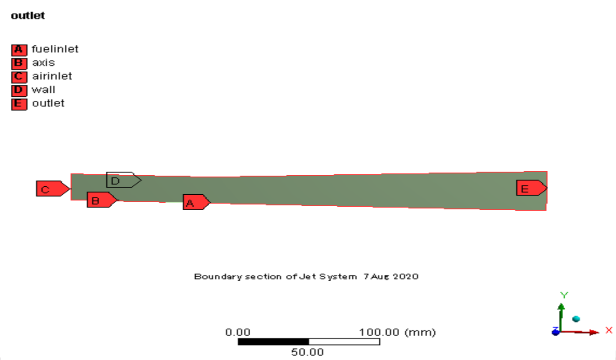

Figure 4.3 Boundary Section of Jet system for Simulation

STEP 2

MESH FILE – In ANSYS, the To Be Meshed Generated Mesh Model appears in 4.5, 4.6, and 4.7.

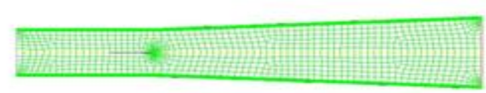

Figure 4.4 Mesh Refinement (close-up) Model of 2d axis symmetry Geometry

Mesh Type: Grid meshing
Element Edge Length =2.5E-004 m
No. of Nodes = 167898
No. of Element = 124678

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/m^3
Viscosity = 1.7894e-05
Boundary Condition:
Operating Condition: Pressure = 101325 Pa

Variables	**Air**	**H_2**
Ma	3.0	1.0
U (m/s)	750	1300
T (K)	340	250
P (Pa)	101325	101325
Density	1.002	0.097
Y_{O2}	0.232	0
Y_{N2}	0.736	0

Y_{H2O}	0.032	0
Y_{H2}	0	1
Mass flow rate (kg/s)	2.5	0.02

2. Solution:
Solution Method:
Pressure- velocity coupling – Scheme -SIMPLE
Pressure – Standard
Momentum – Second order
Turbulent Kinetic Energy (k) Second order
Turbulent Dissipation Rate (e) Second order
Solution Initialization: Initialized the solution to get the initial solution for the problem.
Run Solution: To get the answer to converge, run it through 5000 iterations.
Post Processing: For viewing and analyzing the outcome. The outcome is available in a number of ways, including graph, value, animation, etc.

5. Results & Discussion

Pressure, temperature, velocity variation, turbulent kinetic energy, stream line function, mass fraction of H2o, mass fraction of O2, mass fraction of H2, kinetic energy, and total temperature are the basis for the simulation of a scram jet engine with a central stud wedge shape and injector pylon geometry. The results are discussed below.

Central strut Wedge shape Injector & Pylon Structure:

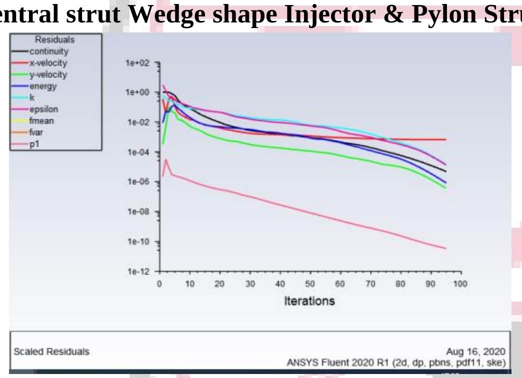

Figure 1 Residuals in Central strut Wedge shape Injector

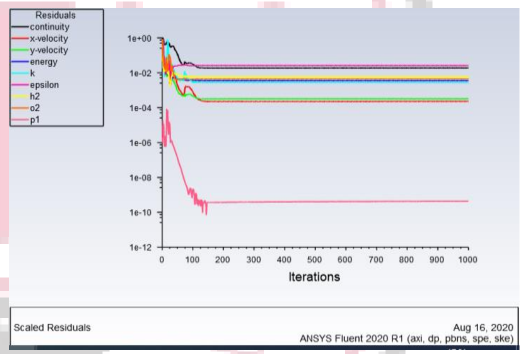

Figure 2 Residuals in Pylon Injector

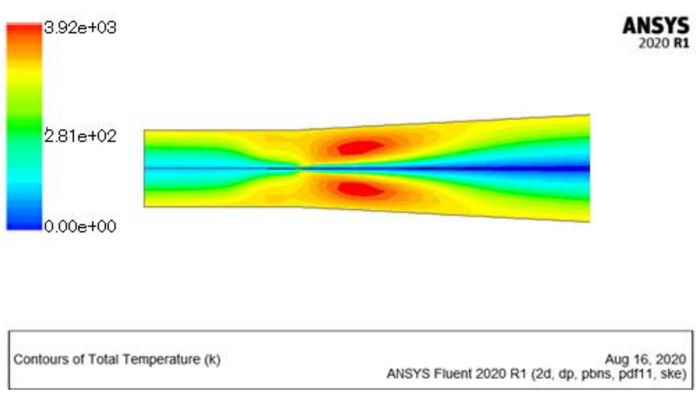

Figure 3 Contour of Total Temperature (K) in Pylon Structure

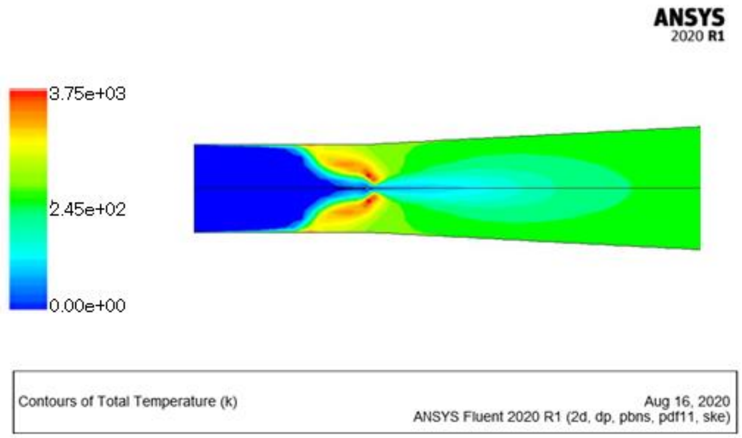

Figure 4 Contour of Total Temperature (K) in Central strut Wedge shape Injector

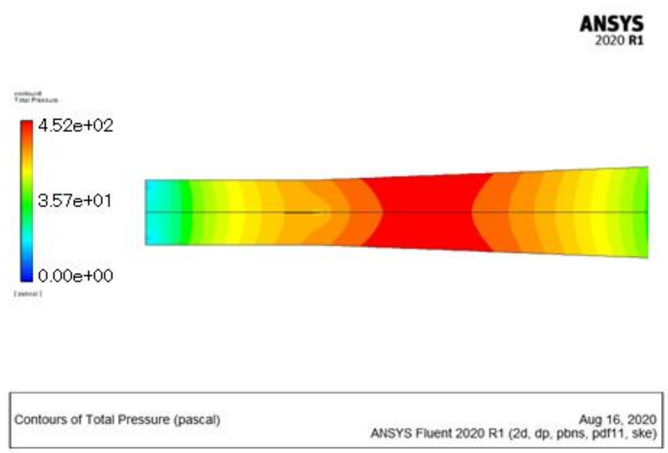

Figure 5 Contour of Total Pressure (Pa) in Central strut Wedge shape Injector

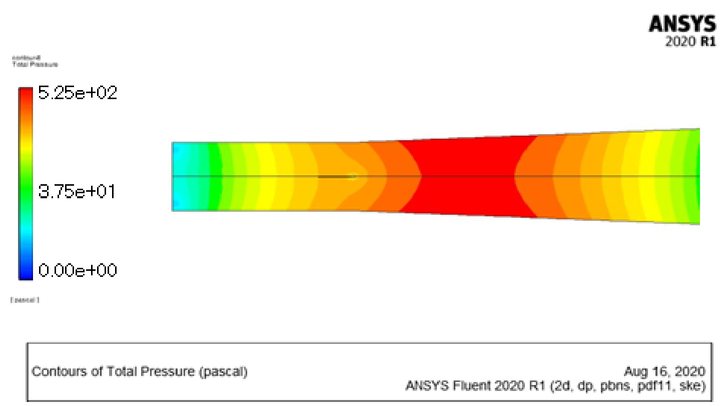

Figure 6 Contour of Total Pressure (Pa) in Pylon Structure

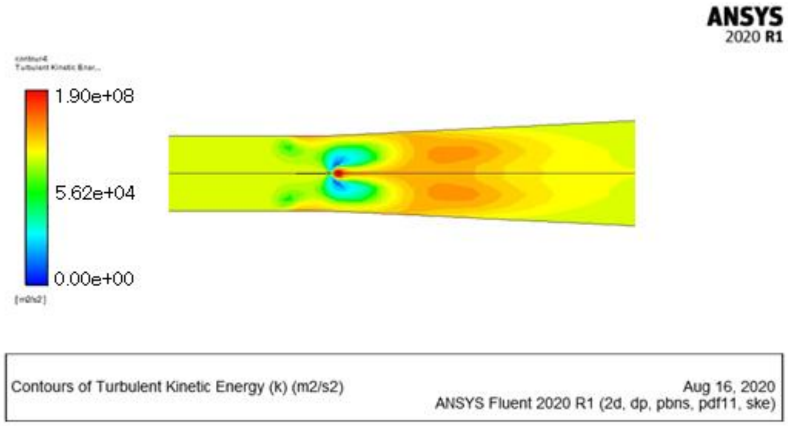

Figure 7 Contour of Turbulent Kinetic Energy in Central strut Wedge shape Injector

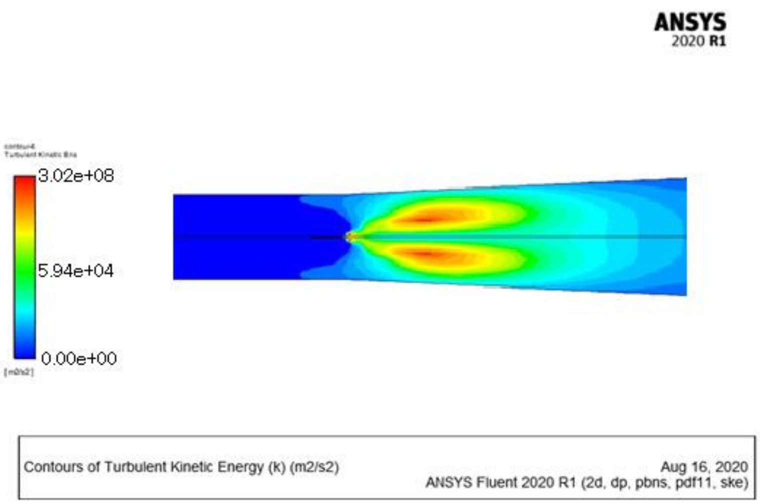

Figure 8 Contour of Turbulent Kinetic Energy in Pylon Structure

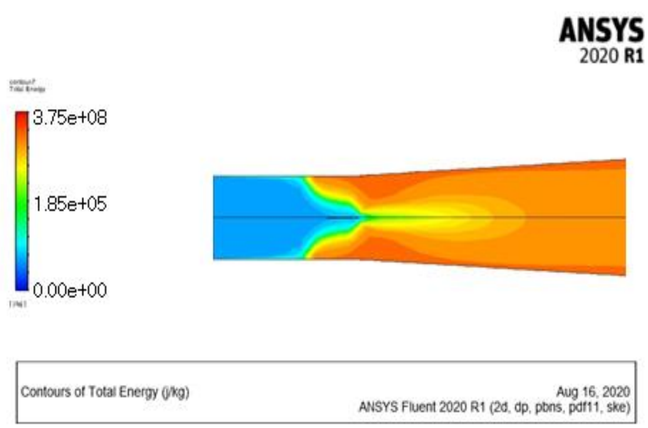

Figure 7 Contour of Total Energy in Central strut Wedge shape Injector

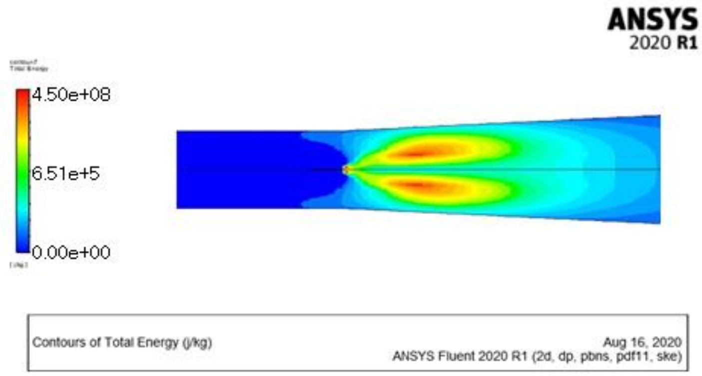

Figure 10 Contour of Total Energy in Pylon Structure

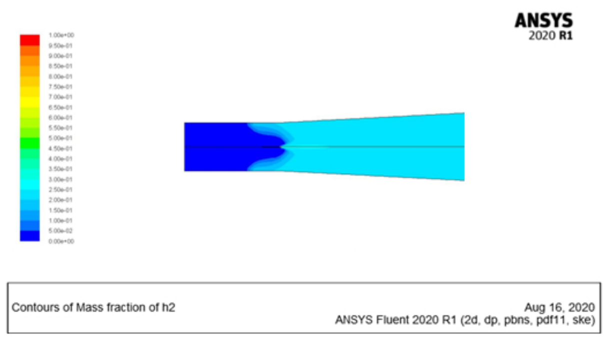

Figure 11 Contours of Mass fraction of H2

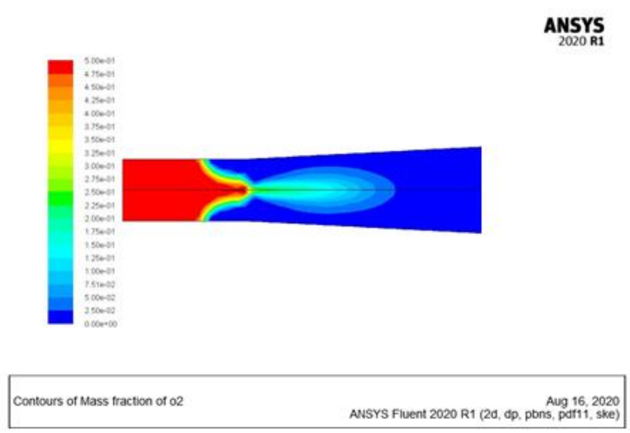

Figure 12 Contours of Mass fraction of O2

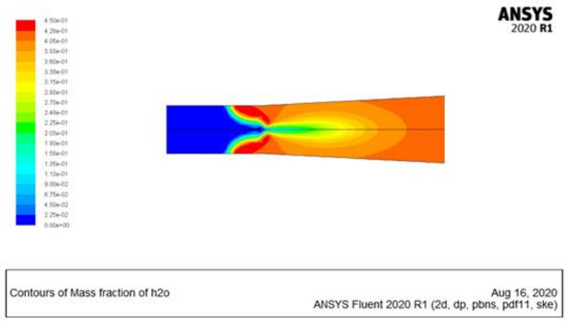

Figure 13 Contours of Mass fraction of H2O

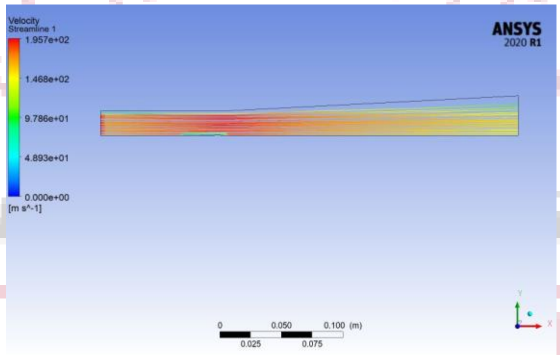

Figure 14 Velocity streamline of the final System

TABLE No . 1

Type of Fuel Injector	Total Temperature (K)	Total Pressure (Pascal)	Turbulent kinetic energy (k-m2/s2)	Total energy (j/kg)	Mass flow rate (Fuel) (kg/s)	Mass flow rate (Air) (kg/s)	Fuel inlet Temp. (K)	Air Inlet Temp. (K)
Central Strut wedge shape Injector	3.75e+03	4.52e+03	1.9e+08	3.50e+08	4.0	1.5	250	340
Pylon Injector	3.92e+03	5.18e+03	3.02e+08	4.75e+08	4.0	1.5	250	340

CONCUSION

Based on the following parameters, the current study simulates a scram jet engine with a central stud wedge shape and injector pylon geometry: pressure, temperature, velocity variation, turbulent kinetic energy, stream line function, mass fraction of H2o, mass fraction of O2, mass fraction of H2, kinetic energy, and total temperature. The results obtained demonstrate that the numerical approach empl

oyed in this work can adequately record the shock wave system and appropriately analyze the flow pattern of the scramjet combustor having centralized strut flame holder. We may infer from the simulation findings that switching from a central wedge

injector to a pylon injector increases the total temperature, pressure, and energy because the air-fuel mixture is improved, the shock wave absorbing capacity is raised, and the combustion rate is subsequently increased. In the case of a pylon injector, shock wave absorption capability and flow field stability are balanced according to stream line flow.

According to this study, the current scramjet combustor problem might be resolved by this type of injector, and the analysis shows the solution in connection to steady flow. The main issue with the planer strut injector is its stability, which limits the engine's Mach number while potentially allowing for constant flow and combustion during flight. According to this study, the current scramjet combustor problem might be resolved by this type of injector, and the analysis shows the solution in connection to steady flow. The scramjet research vehicle may benefit from this endeavor in terms of modifying the engine's mach number and combustion stability. This study shows that a Pylon injector increases the temperature by almost 13 times the inlet temperature and makes the scramjet combustor more stable additionally, it demonstrates how this kind of injector releases a significant amount of energy and speeds up the fuel-air mixture, both of which are critical for scramjet engines.

REFERENCES

[1] K.M.Pandey, Member IACSIT and A.P. Singh "Recent Advances in Experimental and Numerical Analysis of Combustor Flow Fields in Supersonic Flow Regime" (2010International Journal of Chemical Engineering and Applications, Vol. 1, No. 2, August 2010 ISSN: 2010-0221

[2] K.M.Pandey and T.Sivasakthivel "CFD Analysis of Mixing and Combustion of a Scramjet Combustor with a Planer Strut Injector" International Journal of Enviromental Science and Development, Vol. 2, No. 2, April 2011.

[3] DEEPU, Mukundan N.1, GOKHALE, Sadanand. S.2, and JAYARAJ, Simon "Numerical Simulation of Supersonic Combustion Using Unstructured Point Implicit Frinite Volume Method "International Journal of Computational Methods 04(02) · June 2007.

[4] K.M.Pandey and T.Sivasakthivel "CFD Analysis of a Hydrogen Fueled Mixture in Scramjet Combustor with Strut Injector by Using Fluent IACSIT International Journal of Engineering and Technology, Vol.3, No.2, April 2011.

[5] L.Hariramakrishnan1, K.Nehru2, T.Sangeetha3" Performance Analysis of Scramjet Inlet '' Imperial Journal of Interdisciplinary Research (IJIR) Vol-3, Issue-1, ISSN: 2454-1362, (2017).

[6] F. Xing1, M.M. Zhao2 and S Zhang3 "Simulations of a Cavity Based Two-Dimensional Scramjet Model 18th Australasian Fluid Mechanics Conference Launceston, Australia 3-7 December (2012).

[7] Prateek Srivastava and K.M. Pandey "COMPUTATIONAL ANALYSIS OF SUPERSONIC COMBUSTION USING CAVITY BASED FUEL INJECTION WITH SPECIES TRANSPORT MODEL AT MACH NUMBER International Journal of Science, Environment ISSN 2278-3687 (O) and Technology, Vol. 3, No 3, 2014, 923 – 930.

[8] M. Chapuis, C. Fureby, E. Fedina, N. Alin & J. Tegnér "LES MODELING OF COMBUSTION APPLICATIONS USING Open FOAM"(V European Conference on Computational Fluid Dynamics ECCOMAS CFD 2010 J. C. F. Pereira and A. Sequeira (Eds) Lisbon, Portugal, 14–17 June 2010.

[9] Lalit Kumar Gaur1, Dr.M.K.Gaur2, Prof.C.S.Malviya3 "SCRAM JET COMBUSTOR WITH PYLON INJECTORBY COMPUTATIONAL FLUID DYNAMICS ANALYSIS" IJEST Vol. VI ,202 ISSN- 2014-0202, (2015).

[10]M. Ferrier_ J. Garraud A. Ristori "Numerical simulations of supersonic combustion experiments for scramjet propulsion in the ONERA F4 hypersonic wind tunnel" Conference: 19th International Space Planes and Hypersonic Systems and Technologies Conference, At Georgia, Atlanta (2013).

[11] Santosh Kumar Dixit1, Rohit Kumar Choudhary2, Suresh Badholiya3 "Santosh Kumar Dixit1, Rohit Kumar Choudhary2, Suresh Badholiya3" Mathematical Modeling and Analysis of Different Type of Fuel Injector in Scramjet Engine Using CFD Simulation in Fluent "Volume 5 Issue II, February 2017 IC Value: 45.98 ISSN: 2321-9653 (2017).

[12] B.S., Aerospace Engineering, M.S., Aerospace Engineering, "Numerical Simulation of Enhanced Mixing in Scramjet Combustor Using Ramp, Tabs and Suction Collar" Seung-Jae Hwang , B.S., Aerospace Engineering, University of Kansas IJERT , VOL. VII 204-208. (2011).

[13] Dr K.M.Pandey1, Binita Nath2, "CFD Analysis of Supersonic Combustor using wedge shaped strut having circular inlet with standard k-є Non premixed turbulence model" International Journal of Advanced Trends in Computer Science and Engineering, Vol.2 , No.1, Pages : 205 – 214 (2013).Special Issue of ICACSE 2013 - Held on 7-8 January, 2013 in Lords Institute of Engineering and Technology, Hyderabad

[14]Vadivel S1, Melvin Jone A2, K Madhan Muthu Ganesh3 "Study the Flow Analysis of Hypersonic Vehicle Combustion" IRACST – Engineering Science and Technology: An International Journal (ESTIJ), ISSN: 2250-3498 (2016)

[15] K.M.Pandey, Professor, Department of Mechanical Engineering,N.I.T Silchar, " CFD Analysis of Supersonic combustion using Diamond shaped Strut Injector with standard K-Є Non-premixed Turbulence model, International Journal of Advanced Trends in Computer Science and Engineering uter Science and Engineering, 1 (1), March – April, 2012, 33.

[16] K.L.N. College of Engineering, Madurai, Tamil Nadu, India "Numerical Analysis Of Turbulence Mixing By Shockwave Interaction In Scramjet Combustor" International Journal of Innovative Research in Science, Engineering and Technology Volume 3, Special Issue 3, March 2014 2014 IEEE International Conference on Innovations in Engineering and Technology (ICIET'14) .
[17] Wei Huang, Shi-bin Luo, Mohamed Pourkashanian, Lin Ma, Derek B.Ingham, Jun Liu and Zhen-guo Wang "Numerical Simulations of a Typical Hydrogen Fueled Scramjet Combustor with a Cavity Flameholder" Proceedings of the World Congress on Engineering 2010 Vol II WCE 2010, June 30 - July 2, 2010, London, U.K .
[18] Nithin.N " CFD Analysis of Single Cavity with Varying Front Ramp Angles and L/D Ratio of a Hypersonic Scramjet Combustion Engine'' IJEDR Volume 3 | Issue 2 | May 2015 (ISSN - 2321-9939).
[19] G.Dinesh kumar1, D.Gowri Shankar2, D.Balaji3 " Computational evaluation of strut based scramjet engine combustion with different conventional fuel/oxidizer" IJERA : Volume 3 Issue 1, January-February 2013 .
[20] R. Mohamed Arif, S. Sangeetha " Effect of Ramp-Cavity Injector in Supersonic Combustion" International Journal of Scientific & Engineering Research, Volume 4, Issue 5, May-2013 1088 ISSN 2229-5518 .
[21] A.Surender paul1, G.Dinesh kumar2 "Numerical Analysis Of Turbulence Mixing By Shockwave Interaction In Scramjet Combustor" International Journal of Innovative Research in Science, Engineering and Technology Volume 3, Special Issue 3, Marc 2014 IEEE International Conference on Innovations in Engineering and Technology (ICIET'14) .
[22] AUGUSTO F. MOURA, MAURÍCIO A. P. ROSA "A NUMERICAL INVESTIGATION OF SCRAMJET ENGINE AIR INTAKES FOR THE 14-X HYPERSONIC VEHICLE" 22nd International Congress of Mechanical Engineering (COBEM 2013) November 3-7, 2013.
[23] Jeevan RaoÅ, BhargavÅ, Krupakar PasalaÅ and Srinivasa RaoÅ " Computational Study of Viscous Flow in a Scramjet" International Journal of Current Engineering and Technology E-ISSN 2277 – 4106, P-ISSN 2347 - 5161 , February 2014, Special Issue-2.
[24] Nithin .N " Numerical Analysis of Supersonic Scramjet Combustion Engine with Double Cavity Configuration at Mach 2 Fuel Injection" International Journal of Science and Research (IJSR) ISSN (Online): 2319-7064 (2013)
[25] Jonathan Philip West "NUMERICAL ANALYSIS OF ETHYLENE INJECTION IN THE INLET OF A MACH SIX SCRAMJET" Environmental science & technology, EST, V0l. VI,203-205, (2012)
[26] Nitish Acharya , Sajan Sharma, Lokesh Silwal, and Sudip Bhattrai "Design and Numerical Analysis of Mixed Compression Inlet of a Shcramjet Engine" International Journal of Advanced Trends in Computer Science and Engineering, Vol.2 ISSN 2278-3091 , No.1, Pages : 60-66
[27] Birgit U. Reinartz "PERFORMANCE ANALYSIS OF A 3D SCRAMJET INTAKE"(Proceedings of the World Congress on Engineering 2010 Vol II, ISBN: 978-988-18210-7-2 WCE 2010, June 30 - July 2, 2010, London, U.K..
[28] R. Srikrishnan, j. Kurian and v. Sriramulu "an experimental investigation of thermal mixing and combustion in supersonic flows",combustion and flame, vol 107, 1996, pp.464-474.
[29] M Deepu "Recent Advances in Experimental and Numerical Analysis of Scramjet Combustor Flow Fields", Vol. 88, May 2007.
[30] S. Zakrzewski and Milton "Supersonic liquid fuel jets injected into quiescent air", International Journal of Heat and Fluid Flow, vol 25, 2004, pp.833–840.