SCRAM JET COMBUSTOR WITH PYLON INJECTOR BY COMPUTATIONAL FLUID DYNAMICS ANALYSIS

Lalit Kumar Gaur[1], Dr.M.K.Gaur[2], Prof.C.S.Malviya[3]

[1]*M.Tech Scholar (Mechanical Engineering),*

[2] *Professor & Guide, M.I.T.S, Mechanical Engineering ,Gwalior (M.P.), India*

[3]*Professor & HOD, Mechanical Engineering, Gwalior (M.P.), (India)*

ABSTRACT

Computational fluid dynamics (CFD) is a computer-based simulation method for analysing fluid flow, heat transfer, and related phenomena such as chemical reactions. This project uses CFD for analysis of flow and heat transfer. Some examples of application areas are: aerodynamic lift and drag (i.e. airplanes or windmill wings), power plant combustion, chemical processes, heating/ventilation, and even biomedical engineering (simulating blood flow through arteries and veins). CFD analyses carried out in the various industries are used in R&D and manufacture of aircraft, combustion engines, as well as many other industrial products.

Scramjet engines are a type of jet engine, and rely on the combustion of fuel and an oxidizer to produce thrust. Similar to conventional jet engines, scramjet-powered aircraft carry the fuel on board, and obtain the oxidizer by the ingestion of atmospheric oxygen (as compared to rockets, which carry both fuel and an oxidizing agent). This requirement limits scramjets to suborbital atmospheric flight, where the oxygen content of the air is sufficient to maintain combustion.

The scramjet is composed of three basic components: a converging inlet, where incoming air is compressed; a combustor, where gaseous fuel is burned with atmospheric oxygen to produce heat; and a diverging nozzle, where the heated air is accelerated to produce thrust. Unlike a typical jet engine, such as a turbojet or turbofan engine, a scramjet does not use rotating, fan-like components to compress the air; rather, the achievable speed of the aircraft moving through the atmosphere causes the air to compress within the inlet. As such, no moving parts are needed in a scramjet. In comparison, typical turbojet engines require inlet fans, multiple stages of rotating compressor fans, and multiple rotating turbine stages, all of which add weight, complexity, and a greater number of failure points to the engine.

From this experimental work we can conclude that this type of injector may solve the recent problem of scramjet combustor in use and this analysis shows the solution regarding stabilized flow. From tangential velocity contours we can see the stability of flow which is the major problem with planer strut injector as which provide limitation in Mach no of engine but may give continuous flow and combustion through the flight. From pressure and temperature analysis we can decide that this pylon injector provide stability in variation in pressure

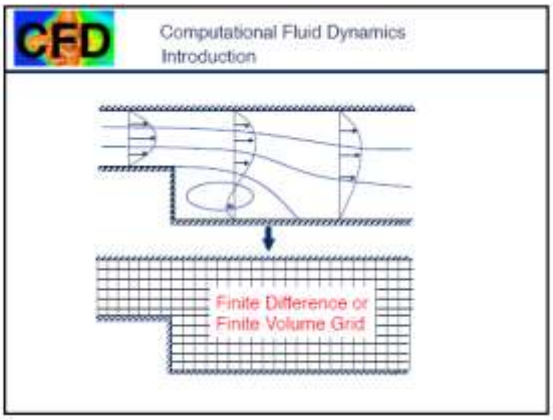
and temperature though the flow condition. This work may give solution of scramjet research vehicle in terms of correction in stability of combustion and Mach no of engine.

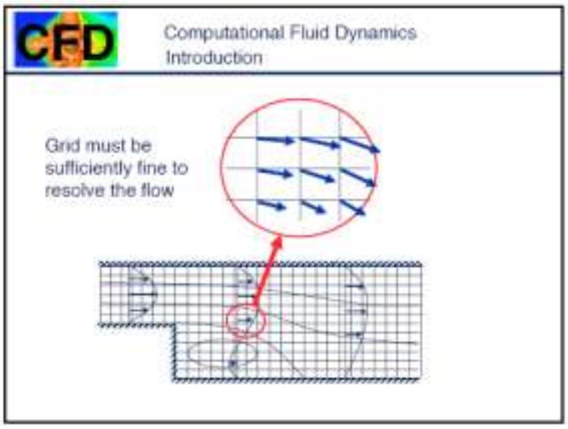
I. COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics (CFD) is a computer-based simulation method for analysing fluid flow, heat transfer, and related phenomena such as chemical reactions. This project uses CFD for analysis of flow and heat transfer. Some examples of application areas are: aerodynamic lift and drag (i.e. airplanes or windmill wings), power plant combustion, chemical processes, heating/ventilation, and even biomedical engineering (simulating blood flow through arteries and veins). CFD analyses carried out in the various industries are used in R&D and manufacture of aircraft, combustion engines, as well as many other industrial products.

It can be advantageous to use CFD over traditional experimental-based analyses, since experiments have a cost directly proportional to the number of configurations desired for testing, unlike with CFD, where large amounts of results can be produced at practically no added expense. In this way, parametric studies to optimise equipment are very inexpensive with CFD when compared to experiments.

This section briefly describes the general concepts and theory related to using CFD to analyse fluid flow and heat transfer, as relevant to this project. It begins with a review of the tools needed for carrying out the CFD analyses and the processes required, followed by a summary of the governing equations and turbulence models and finally a discussion of the discretisation schemes and solution algorithms is presented.





II. CFD COMPUTATIONAL TOOLS

This section describes the CFD tools required for carrying out a simulation and the process one follows in order to solve a problem using CFD. The hardware required and the three main elements of processing CFD simulations: the pre-processor, processor, and post-processor are described.

There is a variety of commercial CFD software available such as Fluent, Ansys CFX, ACE, as well as a wide range of suitable hardware and associated costs, depending on the complexity of the mesh and size of the calculations. Commercial CFD packages can cost up to about $20000 (US Dollars) per year for licenses, maintenance, and support. Complicated transient cases with fine meshes will require more powerful computer processors and RAM than simpler cases with rough meshes. A typical engineering workstation (i.e. 32 GB processing RAM with quad processors) at a cost of approximately $3000-$5000 (US Dollars), or a combination of several processors running in parallel, is probably the minimum investment needed to get started.

III. SCRAMJET ENGINES - DESIGN PRINCIPLES

Scramjet engines are a type of jet engine, and rely on the combustion of fuel and an oxidizer to produce thrust. Similar to conventional jet engines, scramjet-powered aircraft carry the fuel on board, and obtain the oxidizer by the ingestion of atmospheric oxygen (as compared to rockets, which carry both fuel and an oxidizing agent). This requirement limits scramjets to suborbital atmospheric flight, where the oxygen content of the air is sufficient to maintain combustion.

The scramjet is composed of three basic components: a converging inlet, where incoming air is compressed; a combustor, where gaseous fuel is burned with atmospheric oxygen to produce heat; and a diverging nozzle, where the heated air is accelerated to produce thrust. Unlike a typical jet engine, such as a turbojet or turbofan engine, a scramjet does not use rotating, fan-like components to compress the air; rather, the achievable speed of the aircraft moving through the atmosphere causes the air to compress within the inlet. As such, no moving parts are needed in a scramjet. In comparison, typical turbojet engines require inlet fans, multiple stages of rotating compressor fans, and multiple rotating turbine stages, all of which add weight, complexity, and a greater number of failure points to the engine.

Due to the nature of their design, scramjet operation is limited to near-hypersonic velocities. As they lack mechanical compressors, scramjets require the high kinetic energy of a hypersonic flow to compress the incoming air to operational conditions. Thus, a scramjet-powered vehicle must be accelerated to the required velocity (usually about Mach 4) by some other means of propulsion, such as turbojet, railgun, or rocket engines.[22] In the flight of the experimental scramjet-powered Boeing X-51A, the test craft was lifted to flight altitude by a Boeing B-52 Stratofortress before being released and accelerated by a detachable rocket to near Mach 4.5. In May 2013, another flight achieved an increased speed of Mach 5.1.

While scramjets are conceptually simple, actual implementation is limited by extreme technical challenges. Hypersonic flight within the atmosphere generates immense drag, and temperatures found on the aircraft and within the engine can be much greater than that of the surrounding air. Maintaining combustion in the supersonic flow presents additional challenges, as the fuel must be injected, mixed, ignited, and burned within milliseconds. While scramjet technology has been under development since the 1950s, only very recently have scramjets successfully achieved powered flight.

IV. LITERATURE REVIEW

Shigeru Aso et.al [2005] worked on the topic of “Fundamental study of supersonic combustion in pure air flow with use of shock tunnel”, and their findings are – The increase of injection pressure generated strong bow shock, resulting in the pressure loses. The shock generator is an effective method to accelerate the combustion. The increase of the injection total pressure raises the penetration of fuel; thus, the reaction zone expands to the center of flow field.

K.M.Pandey and Siva Sakthivel. T[2010] worked on the topic of "Recent Advances in Scramjet Fuel Injection - A Review", and their findings are – Fuel injection techniques into scramjet engines are a field that is still developing today. The fuel that is used by scramjets is usually either a liquid or a gas. The fuel and air need to be mixed to approximately stoichiometric proportions for efficient combustion to take place. The main problem of scramjet fuel injection is that the airflow is quite fast, meaning that there is minimal time for the fuel to mix with the air and ignite to produce thrust (essentially milliseconds).Hydrogen is the main fuel used for combustion. Hydrocarbons present more of a challenge compared to hydrogen due to the longer ignition delay and the requirement for more advanced mixing techniques.

Enhancing the mixing, and thus reducing the combustor length, is an important aspect in designing scramjet engines. There are number of techniques used today for fuel injection into scramjet engines.

Kyung Moo Kim et.al [2004] worked on the topic of "Numerical study on supersonic combustion with cavity-based fuel injection", and their findings are – When the wall angle of cavity increases, the combustion efficiency is improved, but total pressure loss increased. When the offset ratio of upper to downstream depth of the cavity increases, the combustion efficiency as well as the total pressure loss decreases.

K. Kumaran and V. Babu [2009] worked on the topic of "Investigation of the effect of chemistry models on the numerical predictions of the supersonic combustion of hydrogen", and their findings are – Multi step chemistry predicts higher and wider spread heat release than what is predicted by single step chemistry. The single step chemistry model is capable of predicting the overall performance parameters with considerably less computational cost. A better tradeoff between thrust augmentation and combustion efficiency can be achieved through staged combustion.

G. Yu, J.G. Li, J.R. Zhao, et al. [2005] worked on the topic of "An experimental study of kerosene combustion in a supersonic model combustor using effervescent atomization", and their findings are – The smaller kerosene droplet having higher combustion efficiency. A local high temperature radical pool in the cavity is crucial in promoting the initiation and the subsequent flame holding of the kerosene combustion in a supersonic combustor.

M Deepu [2007] worked on the topic of "Recent Advances in Experimental and Numerical Analysis of Scramjet Combustor Flow Fields", and his findings are – Increase in jet to free stream momentum flux ratio will result in the increase of jet penetration to free stream for all kinds of jets. Injector orientation plays an important role in the strength of the bow shock, with the shocks created by oblique injector being substantially weak compared to transverse injector.

S. Zakrzewski and Milton [10] worked on the topic of "Supersonic liquid fuel jets injected into quiescent air", and their findings are – Supersonic liquid jets M = 1.8 develops from a flat front to a rounded bow within some 10 mm M = 5.2, the bow shape is more pointed and shows signs of an oscillation from more to less pointed.

V. PRESENT WORK

Governing equation:This section is a summary of the governing equations used in CFD to mathematically solve for fluid flow and heat transfer, based on the principles of conservation of mass, momentum, and energy. Details of how they are actually used in the CFD computations are described in Appendix A1: CFD Computations.

VI. CONSERVATION EQUATIONS

The conservation laws of physics form the basis for fluid flow governing equations (previously listed as Equations 1-3 in Section 2.1: Governing Equations and Numerical Schemes). The laws are:

- Law of Conservation of Mass: Fluid mass is always conserved. (Equation 1)

$$\frac{\partial(\rho u_i)}{\partial x_i} = 0. \tag{3.1}$$

- Newton's 2nd Law: The sum of the forces on a fluid particle is equal to the rate of change of momentum. (Equation 2)

$$\frac{\partial}{\partial x_i}(\rho u_i u_j) = \frac{\partial}{\partial x_i}\left(\mu \frac{\partial u_j}{\partial x_i}\right) - \frac{\partial p}{\partial x_j}. \tag{3.2}$$

- First Law of Thermodynamics: The rate of head added to a system plus the rate of work done on a fluid particle equals the total rate of change in energy. (Equation 3)

$$\frac{\partial}{\partial x_i}(\rho u_i T) = \frac{\partial}{\partial x_i}\left(\frac{k}{C_p}\frac{\partial u_j}{\partial x_i}\right). \tag{3.3}$$

The fluid behaviour can be characterised in terms of the fluid properties velocity vector **u** (with components *u*, *v*, and *w*in the *x*, *y*, and *z* directions), pressure *p*, density ρ, viscosity μ, heat conductivity *k*, and temperature *T*. The changes in these fluid properties can occur over space and time. Using CFD, these changes are calculated for small elements of the fluid, following the conservation laws of physics listed above. The changes are due to fluid flowing across the boundaries of the fluid element and can also be due to sources within the element producing changes in fluid properties. This is called the Euler method (tracking changes in a stationary mass while particles travel through it) in contrast with the Lagrangian method (which follows the movement of a single particle as it flows through a series of elements).

VII. FINITE VOLUME METHOD AND DIFFERENCING SCHEMES

The mass, momentum, and scalar transport equations are integrated over all the fluid elements in a computational domain using CFD. The finite volume method is a particular finite differencing numerical technique, and is the most common method for calculating flow in CFD codes. This section describes the basic procedures involved in finite volume calculations. The finite volume method involves first creating a system of algebraic equations through the process of discretising the governing equations for mass, momentum, and scalar transport.

VIII. SOLUTION ALGORITHMS

The value of the scalar properties of interest (i.e. temperature) at a particular location in the computational domain depends on the flow's direction and velocity, which must also be solved for in the calculation process. To There are many algorithms available for this purpose, the most popular are the SIMPLE and PISO methods. This section describes the SIMPLE algorithm and compares it to the PISO algorithm. A short overview of the 'staggered grid' is also given. In order to calculate the entire flow field, the momentum equations in all three directions and the continuity equation must be solved, which include terms for each velocity component and the pressure gradient. There is therefore a coupling between pressure and the three velocity components. In cases where the pressure gradient is known, the usual discretisation techniques can be used to obtain discredited equations for velocity, since pressure can be calculated as any other scalar. However, there is no transport equation for pressure alone, and therefore pressure must be found using another method when the pressure gradient is not known. If the flow is compressible, the transport of density is found using the continuity equation and a scalar property such as temperature is found by the combination of the momentum equations and energy equation. Then the solutions for density and temperature are used for finding the pressure using the equation of state. When fluid is incompressible the density is not linked to pressure, a guess-and-check technique such as SIMPLE is required in order to solve the entire flow field.

IX. SUMMARY OF SOLUTION ALGORITHMS

The SIMPLE algorithm is used for this project, in the solvers simple Foam and rhoSimple Foam. It is a known algorithm and used in many CFD codes. SIMPLER can be used to more efficiently use computer processing time (even though there are more calculations) since it uses the correct value for pressure. For certain types of flow, SIMPLEC and PISO can be just as efficient as SIMPLER. Which algorithm to use depends on the specific case being studied: the flow conditions, degree of dependence of momentum and scalars, and the specific numerical schemes used.

It is assumed that the air flow in the heat exchanger reaches a steady-state and does not contain fluctuations in time. Therefore steady-state calculations (rather than transient) are the focus. However, to investigate the possibility of this, one simulation was carried out and results looked as though they were steady-state. To calculate this situation, the solution algorithms can be calculated over time, in which case PISO is probably preferred, while SIMPLE can be converted to run transient calculations, and the continuity imbalance will contain an extra transient term to account for the time integrals of the variables, and an iteration process is used so that the solution converges for each time step (computationally expensive). In the case of this project, since steady-state behaviour is studied, the transient solution procedures are not described in detail.

X. BASIC STEPS TO PERFORM CFD ANALYSIS

10.1 Preprocessing

- **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.
- **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 increase.
- **Type of Solver:** Choose the solver for the problem from Pressure Based and density based solver. **Physical model:** Choose the required physical model for the problem i.e. laminar, turbulent, energy, multiphase, etc**.**

- **Material Property**: Choose the Material property of flowing fluid.
- **Boundary Condition:** Define the desired boundary condition for the problem i.e. velocity, mass flow rate, temperature, heat flux etc.

10.2 Solution

- **Solution Method :** Choose the Solution method to solve the problem i.e. First order, second order
- **Solution Initialization:** Initialized the solution to get the initial solution for the problem.
- **Run Solution:** Run the solution by giving no of iteration for solution to converge**.**

10.3 Post Processing

- **Post Processing:** For viewing and interpretation of Result. The result can be viewed in various formats: graph, value, animation etc.

CFD Analysis of hydrogen combustion using Ansys Fluent

10.4 Preprocessing

CAD Model: Generation of 2d ax symmetric geometry influent.

Fig-1.1 Ax Symmetric Geometry

- **Mesh:** Generate the mesh in the Ansys Mesh software.
  Mesh Type: grid meshing

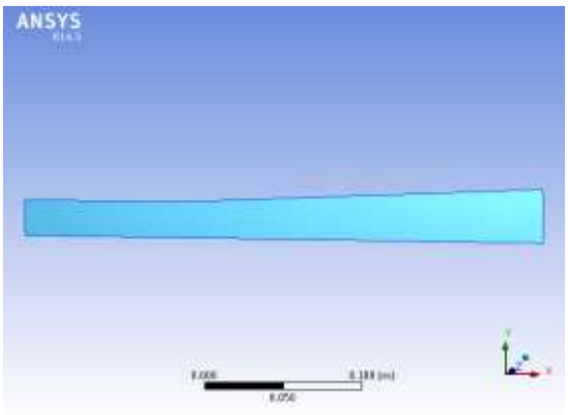


Fig-1.2. meshing of 2d axis symmetric geometry

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 mdel

Material Property: Flowing fluid is air

Density of air = 1.225 kg/m^3

Viscosity = 1.7894e-05

10.5 Boundary Condition

Operating Condition: Pressure = 101325 Pa

Variables	Air	H_2
Ma	2.0	1.0
u (m/s)	730	1200
T (K)	340	250
P (Pa)	101325	101325
ρ (kg/m^3)	1.002	0.097
Y_{O_2}	0.232	0
Y_{N_2}	0.736	0
Y_{H_2O}	0.032	0
Y_{H_2}	0	1
Mass flow rate(kg/s)	1.5	0.0015 to 0.004

10.6. 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

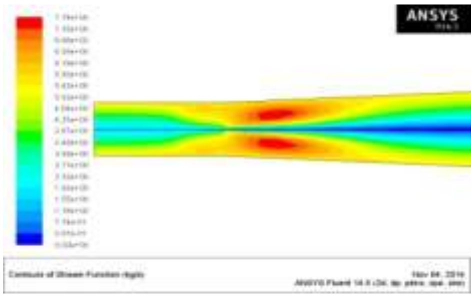
Solution Initialization: Initialized the solution to get the initial solution for the problem.

- Run Solution: Run the solution by giving 500 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.

XI. RESULT AND DISCUSSION

The analysis of scram jet engine with pylon injector is done based on function of stream function, mass fraction of H_2o,mass fraction of o_2,mass fraction of H_2,kinetic energy,total temperature ,total pressure, static pressure, static temperature and the result is discussed here below.

11.1 Steam Function



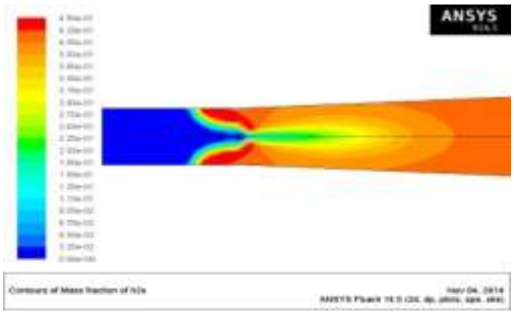


Fig-1.3. Contours Of Stream Function

Fig-1.4.Contours Of Mass Fraction Of H_2o

As shown in figure the steam function is minimum at the surfaces and its maximum at the surrounding between wall and injector. As shown the stream function contours are maximum at the time of fuel injection.

11.2 Mass fraction of H_2O

After h_2 gets injected and combustion takes place we find maximum mass fraction of h_2o at the surface near the injector and its near 4.5e-01pbns which shows maximum mass converted to h_2o with reaction to atmospheric air. The mass fraction near the axis after injection is near 2.25e-01pbns which is half the maximum amount of mass fraction in analysis.

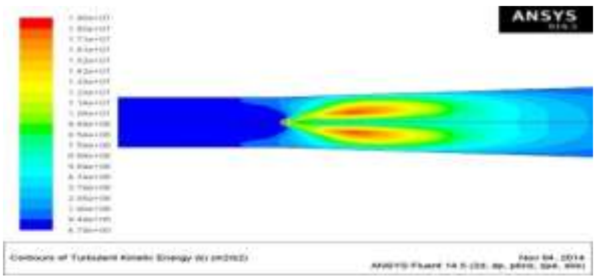


Fig-1.5.contours of kinetic energy

11.3 Kinetic Energy

As shown in figure the kinetic energy of air gets increase after injection and here the kinetic energy gets increase by oblique shock wave and maximum at injection start and after 25mm after injection as shown near the wall surface which is near 1.8e+07k .
Mass fraction of h2

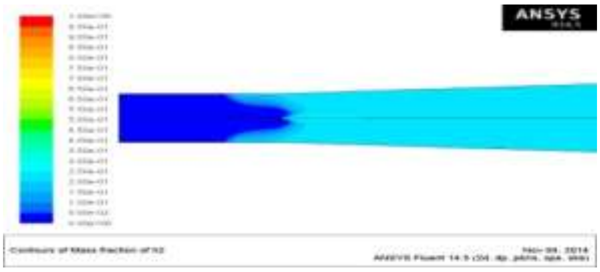


Fig-1.6.contours of mass fraction of H_2

As analysis shows the h2 mass fraction Increase near the fuel injector and after injection it's the same after injection through the surface from axis through the wall surfaces. Mass fraction is near the 2.5e-01 through flow.

11.4 Total energy

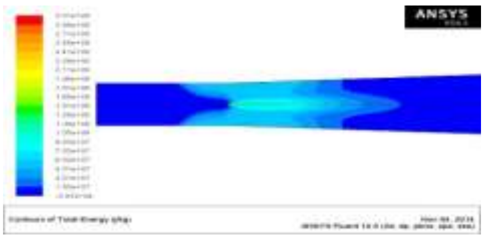


Fig- 1.7.Contours of total energy

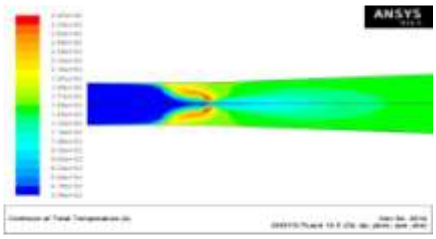


Fig-1.8. Contours of total temperature

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 pylon injector. We find maximum energy of 9.03e+07 j/kgat center after injection.

11.5 Total Temperature

The total temperature is increased at the time of injection and near the surface of injection but its temperature is in between 2740k near the region of injector after fuel injection done

11.6 Total Pressure

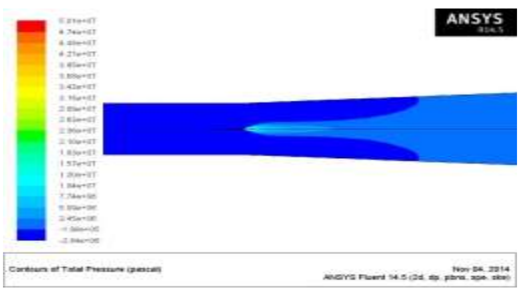


Fig-1.9. contours of total pressure

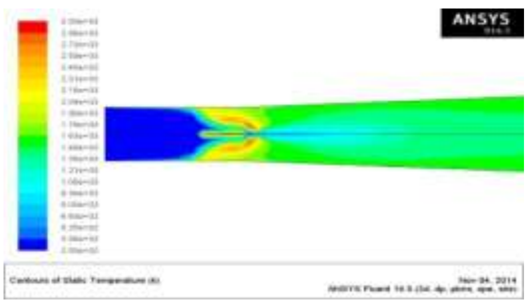


Fig-1.10. Contours of static

11.7 Temperature

The total pressure increases after fuel injection from pylon injector and total pressure is maximum at the area of injection which is near 1.57+07pa and the total pressure remains near same through the axis after ignition.

11.8 Static Temperature

Static temperature increases near the boundaries of injector and near the surface of fuel injection the velocity of fuel is converted to the temperature rise at surfaces and temperature is near same after injection for next 100mm in combustion.

11.9 Static Pressure

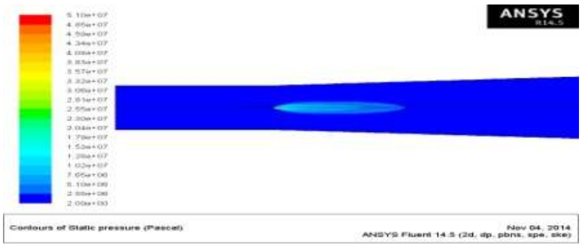


Fig-1.10.contour of static pressure

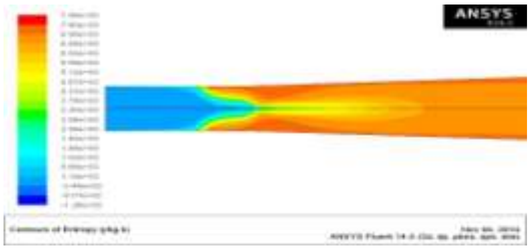


Fig-1.11-contours of entropy

As shown in figure 1.10 static temperature increases and its remains increasing through axis near 100mm and doesn't change after this region.

11.10 Entropy

The result of analysis shows that the entropy varies from -1260j/kgk to the 7860j/kgk through the flow and the maximum value of entropy is at wall boundaries after fuel injection and near the injector the entropy is 3300J/kgk after fuel injection entropy is at wall remains maximum as compared to injector axis.

11.11 Tangential Velocity

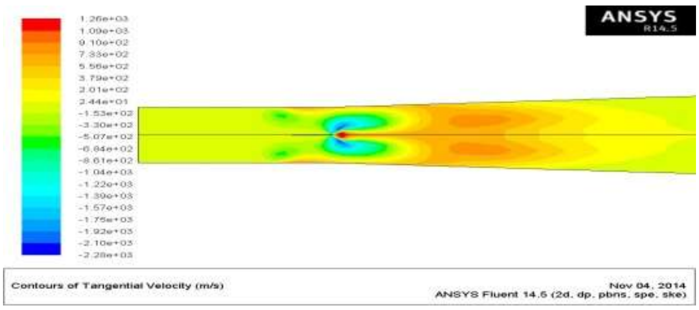


Fig-1.12-contours of tangential velocity

As analysis shows the tangential velocity is minimum at only injector boundaries near the boundary layer of flow .and the tangential velocity is near the 733m/s and the tangential velocity is maximum at distance 100mm and at the fuel injection. And tangential velocity is minimum because of oblique shockwaves on the boundary of injector at starting. This shows the average tangential velocity at both the side of injector axis which creates condition for better combustion.

XII. CONCLUSION

From this experimental work we can conclude that this type of injector may solve the recent problem of scramjet combustor in use and this analysis shows the solution regarding stabilized flow.

From tangential velocity contours we can see the stability of flow which is the major problem with planer strut injector as which provide limitation in Mach no of engine but may give continuous flow and combustion through the flight.

From pressure and temperature analysis we can decide that this pylon injector provide stability in variation in pressure and temperature though the flow condition.This work may give solution of scramjet research vehicle in terms of correction in stability of combustion and Mach no of engine.

REFERENCES

1] Shigeru Aso, ArifNur Hakim, Shingo Miyamoto, Kei Inoue and Yasuhiro Tani, "Fundamental study of supersonic combustion in pure air flow with use of shock tunnel", Department of Aeronautics and Astronautics, Kyushu University, Japan , ActaAstronautica, vol 57, 2005, pp.384 – 389.

[2] Kyung Moo Kim 1, SeungWookBaek and Cho Young Han, "Numerical study on supersonic combustion with cavity-based fuel injection", *International Journal of Heat and Mass Transfer*, vol 47, 2004, pp.271–286.

[3] yuanShengxue, "supersonic combustion", vol. 42, no. *2,* science in china (Series A), February 1999,

[4] Gruenig and f. Mayinger, "supersonic combustion of erosene/h2-mixtures in a model scramjet combustor", institute for thermodynamics, technical university Munich, and d-85747.

[5] K. Kumaran and V. Babu, "Investigation of the effect of chemistry models on the numerical predictions of the supersonic combustion of hydrogen", *Combustion and Flame*, vol 156, 2009, pp.826–841.

[6] T. Cain and C. Walton "review of experiments on ignition and flame holding in supersonic flow" Published by *the America Institute of Aeronautics and Astronautics,* RTO-TR-AVT-007-V2.

[7] Yu, J.G. Li, J.R. Zhao, L.J. Yue, X.Y. Chang and C.-J. Sung "An experimental study of kerosene combustion in a supersonic model combustor using effervescent atomization", Proceedings of the *Combustion Institute*, vol 30, 2005, pp. 2859–2866.

[8] M Deepu "Recent Advances in Experimental and Numerical Analysis of Scramjet Combustor Flow Fields", Vol. 88, May 2007.

[9] 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.

[10] K.M.Pandey and T.Sivasakthivel, "Recent Advances in Scramjet Fuel Injection - A Review," *International Journal of Chemical Engineeringand Applications* vol. 1, no. 4, pp. 294-301, 2010.