Defence Science Journal, Vol. 74, No. 2, March 2024, pp. 288-292, DOI : 10.14429/dsj.74.18510

Flow Analyses of Integrated Liquid Fuel Ramjet Propulsion System

M. Srinivasa Rao* and Abhishek Richhariya

DRDO-Defence Research and Development Laboratory, Hyderabad – 500 058, India
**E-mail: msrinivasarao.drdl@gov.in*

ABSTRACT

A CFD study is performed to check the integrated flow behaviour of the Liquid Fuel Ramjet Propulsion system by including air intakes, combustor and nozzle. Resolving both supersonic and subsonic flow scales in the same domain makes the simulations complex. Addition of combustion with stiff chemistry makes the simulations more difficult. Analyses are carried out using commercially available CFD software. Liquid fuel is injected as discrete phase and the flow turbulence is modelled using Realizable k-ε turbulence model. Jet-A + air combustion has been simulated using combined finite rate/eddy dissipation model. Finite rate chemistry was modelled using three step chemistry which was obtained from the published literature. Flow structures such as oblique shocks, normal shocks and combustion are captured in the analyses. Normal shock with respect to the pressure obtained because of the pressure raise due to combustion is observed. Also, obtained CFD results are compared with that the experimental values that are conducted in connect pipe mode, for the same operating point, and a good agreement between the two are observed.

Keywords: Liquid fuel; Ramjet combustor; V-gutter; Reacting flow simulation; Flame holder

NOMENCLATURE

A	Pre exponential factor
Ta	Activation temperature
Y_i	Molar concentration of species i
n	Exponential constant
k	Turbulent kinetic energy
ε	Eddy dissipation rate
C^*	Characteristic velocity
α	Reactants
β	Products

1. INTRODUCTION

CFD analyses of the complete Ramjet propulsion system will give seamless understanding of the interaction between the combustion process and the intakes. In principle Ramjet propulsion system consists of air intakes, combustor and nozzle along with other components of propulsion system such as film cooling liner, fuel expulsion system. Ramjet propulsion system works on principle of Brayton cycle where, the supersonic atmospheric air enters into the air intakes by a series of oblique shocks before the presence of normal shock and there after the flow gets mixed with fuel; combusts and expands in nozzle to provide necessary thrust to the supersonic vehicle.

Intakes of the supersonic vehicle is detrimental in performance of the ramjet engine. Intakes also plays an important role in vehicle dynamics and over efficiency of the system.

Received : 12 September 2022, Revised : 17 December 2022
Accepted : 03 January 2023, Online published : 26 March 2024

Higher pressure recovery, flow uniformity and compatibility with combustor are the design criteria for the supersonic intakes[1]. Aerodynamics of the flow inside air intakes is quite complex which includes a series of oblique shocks and the separation of supersonic flow and subsonic flow by a normal shock. Intakes also shall crater for the removal of adverse boundary layer by bleeding to obtain a stable subcritical operation.

Atomisation of fuel into finer spray, coupling of liquid and gas phases, providing sufficient energy for ignition and flame stabilisation of the complexities that are to be addressed during the design of liquid fuel ramjet combustor. Insufficient mixing of fuel and air causes unstable combustion and the combustor with appropriate fuel injection system is designed to obtain a stable flame and high combustion efficiency[2].

With the aid of high end parallel computing CFD became a tool for designers of aerospace applications[3].

CFD was extensively used for establishing flow structures and characteristics of intakes[4]. Also, reacting flow simulations for a liquid fuel ramjet combustor are carried out and are compared with experimental results.[5] Flow simulations are performed for entire missile configuration including external and internal flows and the performance predictions were carried out[6-8]. Even though the simulations are carried out for entire missile configuration, no literature evidence of the CFD simulation for entire propulsion system, modeling combustion reactions, seems to be available.

In the present work, CFD simulations are performed for the entire propulsion system including air intakes, combustor and nozzle. The simulations are carried out with an objective

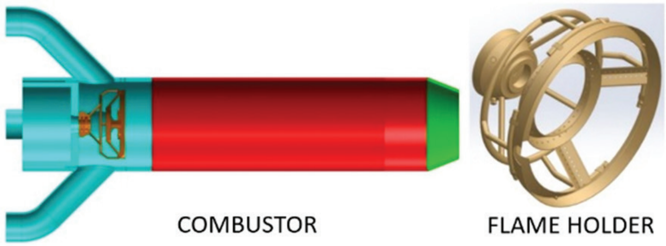


Figure 1. Model of the combustor used for validation.

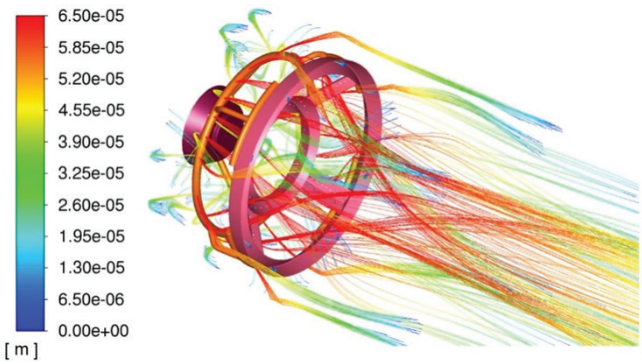


Figure 2. Particle tracks showing fuel droplets.

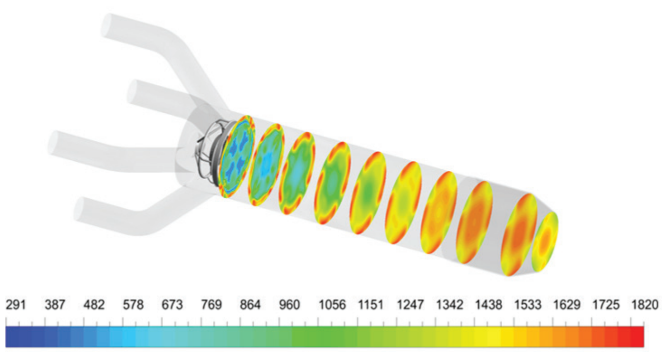


Figure 3. Temperature distribution in combustor.

Table 1. Comparison of CFD results and experimental values–validation studies

		CFD	Expt
Chamber pressure	bar	2.83	3.0 ± 0.10
Gas temperature	K	1750	1733
Total pressure at inlet	bar	3.08	3.38 ± 0.10
C^* efficiency	-	91.06	91.69 ± 3

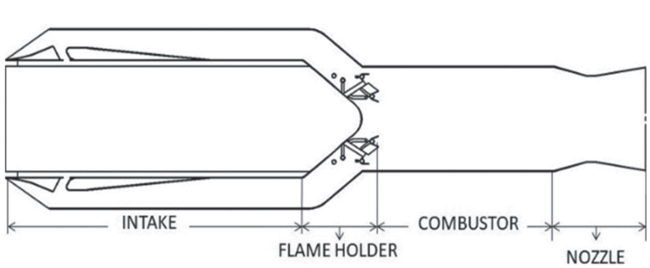


Figure 4. Sketch of complete LFRJ propulsion system (not to scale).

of obtaining flow behavior in air intakes coupled with chemical reactions in combustor. This work gives us the insight of the effect of the flow in the intakes because of the heat release occurring in the combustor. The interaction between the intakes and the combustion can be better understood. Simulation of intakes along with combustion is a difficult task because of complexity in the flow phenomenon and stiff chemistry. Details of the connect pipe testing whose results are used to compare with the results obtained using CFD analyses is not in the scope of the present work and this work is mainly concentrated on the methodology to carryout CFD simulations for the entire LFRJ propulsion system.

2. VALIDATION STUDIES

Validation for the reacting flow is performed using the testing data of a liquid fueled ramjet combustor for which extensive in house test data is available. Considered LFRJ combustor is having four circular air inlets which dumps air flow into the combustor. V-gutter type flame holders are used for stabilising the flame. Model of the combustor highlighting the flame holder for the present combustor is shown in Fig. 1.

The flame holder is made of two circular V-gutter and 4 radial V-gutters. Fuel is injected through plain orifice injectors. Reacting flow simulations are performed for the combustor and the results obtained from the CFD analyses are found to be having good match with that of the values obtained from connect pipe testing[5].

Particle tracks of the discrete phase fuel particles colored by the droplet diameter are plotted in Fig. 2. It is observed that most of the fuel particles are concentrated near to the core region. Contours of temperature along the length of the domain are shown in Fig. 3.

Results obtained from the CFD analyses are compared with that of the measure values in the test bed and are tabulated in Table 1.

The values taken from the CFD results are values at the points corresponding to the measured locations. C^* efficiency estimated from CFD and experiments is compared in Table 1, and a good match is observed between experiments and CFD. The pressure values are normalised with that of the static pressure at the inlet location.

3. LFRJ PROPULSION SYSTEM

With the confidence gained from the validation studies, flow analyses over the entire LFRJ propulsion system is attempted. As mentioned, air intakes, combustor and nozzles of LFRJ propulsion system is considered for the analyses.

LFRJ propulsion system consists of four numbers of rectangular air intakes for obtaining required amount of air mass flow rate. Air intakes are designed with multiple ramp system before normal shock for a supercritical margin more than 10 % at design condition. Compressed air enters into the combustor with multi stage fuel injections. Flame stabilization is achieved by using v-gutters. Sketch of the complete LFRJ propulsion system is shown in Fig. 4.

4. PARAMETERS OF INTREST

From the analyses of the complete LFRJ propulsion system, the following performance parameters are achieved. The following stations are identified for defining the performance parameters of the propulsion system.

Station 1 is before intake

Station 2 is at the end of diffuser of intake

Station 3 is at the end of combustion zone

Station 4 is at the end of nozzle

The following performance parameters are defined for the propulsion system

- Pressure recovery of air intake

$$PR = \frac{P_{02}}{P_{01}} X\ 100$$

- Mass Capture Ratio

Mass Capture Ratio (MCR) is defined as ratio of actual mass captured by the intake to the ideal mass capture

- C* efficiency

$$\eta_{C^*} = \frac{C^* obtained}{C^* Ideal} X\ 100$$

where, $C^* = \frac{p_c A_t}{\dot{m}}$

5. METHODOLOGY

Numerical simulations using commercially available CFD software have been described in validation studies. Geometry of the propulsion system includes four numbers of air intakes.

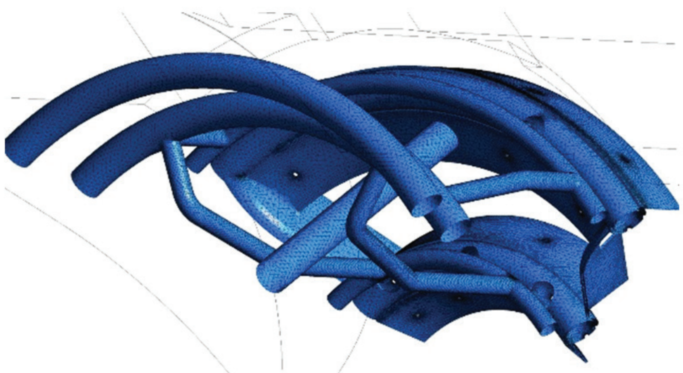


Figure 5. Surface mesh over the flame holder.

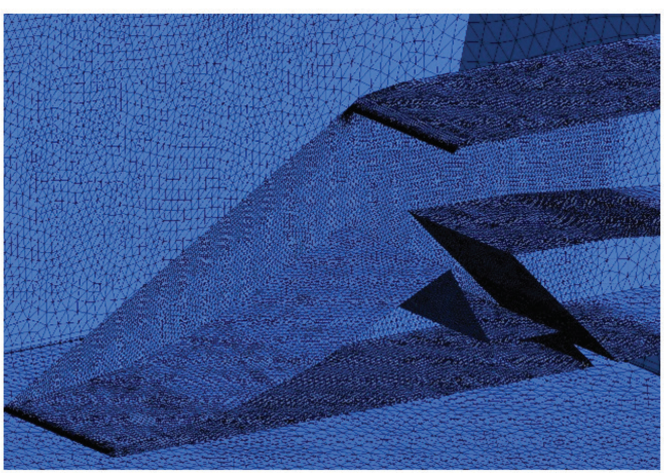


Figure 6. Surface mesh over 2D air intake.

Table 2. Free stream conditions

Air static pressure	bar	0.55
Air static temperature	K	271

Taking into account of periodicity of the propulsion system, a 90˚ sector was modelled. At the exit of the combustor, the domain is extended by 5 times the radius of combustor in length and 3times the radius of the combustor so as to model the atmosphere condition exactly as per the test. A tetrahedral mesh of around 12 million cells is generated and the boundary layer is simulated with 5 numbers of prism layers with a wall Y+ > 50 so as to avoid laminar-turbulent transition zone.

Surface mesh over the flame holder is shown in Fig. 5. All the complex features of the flame holder can be observed here.

Surface mesh over the 2D air intake is shown in Fig. 6. Modelling of the ramps, bleed and cowl regions can be identified in Fig. 6. Mesh that is refined near to the ramps to resolve shock structures.

Governing Eqn. are solved with density based coupled solver. Reacting flow analyses were carried out for various ramjet operating conditions to check the flow interaction between intake and combustor. Among the given operating conditions, one of the condition is shown in Table 2. Flow is treated to be compressible since density varies as a function of temperature.Pressure far field condition with the given pressure and temperature as given in Table 2 along with Mach number is specified as inlet condition. Fuel is injected in droplet phase with an equivalence ratio of xx over the fuel injectors located on as an integral part of flame holder.

Combustion chemistry of jet A /air[9] is modelled using global three step mechanism for $C_{12}H_{23}$ with six species.

The three steps of reaction are as follows:

$$C_{12}H_{23} + 11.75\ O_2 \rightarrow 12\ CO + 11.5\ H_2O \quad (1)$$

$$CO + 0.5\ O_2 \rightarrow CO_2 +\ H_2O \quad (2)$$

$$O_2 +\ N_2\ \rightarrow\ 2NO \quad (3)$$

Combustion is simulated using combined finite rate and eddy dissipation model. Finite rate of reaction is modeled using Arrehenius rate of reaction given by:

$$r = AT^{n_T} e^{\frac{-T_a}{T}} Y_\alpha^{n_\alpha} Y_\beta^{n_\beta}$$

Constants of Arrhenius rate for each of the reaction[8]are tabulated in Table 3.

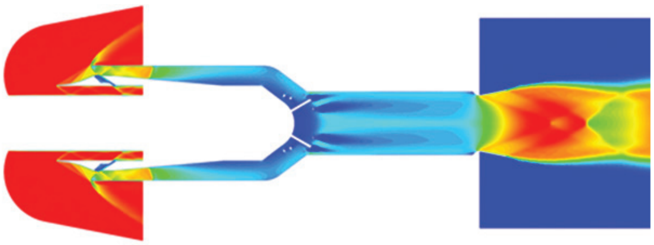
Table 3. Arrhenius rate constants of the reactions

A [m,kg, mol, s]	n_T	n_{C12H}	n_{O2}	n_{CO}	N_{N2}	T_a[K]
1.04 x10 ^{9}	0	1.0	0.5			10108
4.04 x10 ^{8}	0		0.5	1.0		6047
7.14 x10 ^{13}	0.5		0.5		1.0	38440

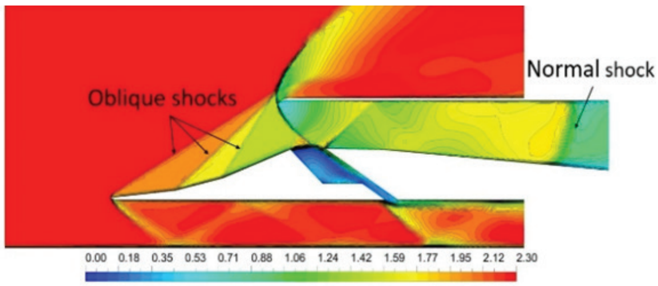
Droplet distribution in the fuel spray is considered to be varying according to Rosin-Rammler distribution. Fuel particles are injected as a hollow cone and the point properties of the fuel particles are calculated from cross flow correlations[10].

6. RESULTS AND DISCUSSIONS

Each number contours obtained by the CFD analyses are shown in Fig. 7. All the flow features, that are captured in CFD analyses are seen in Fig. 7. Multiple oblique shocks that

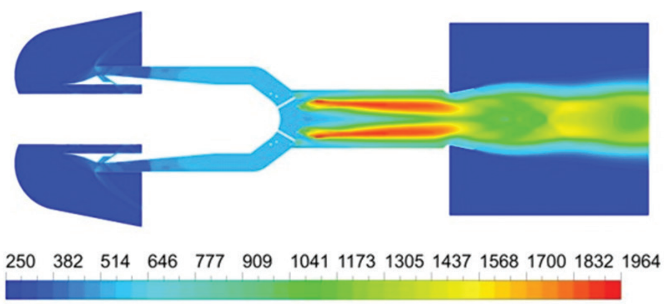


(a)

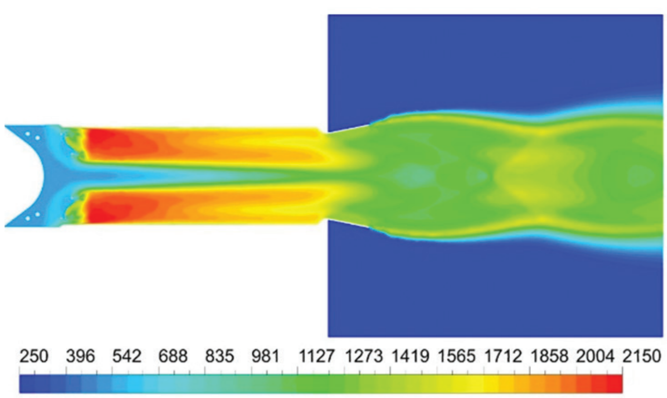


(b)

Figure 7. Mach number contours resolved at a plane in line with intakes; (a) Flow field in the propulsion system and (b) Shock structure of intake.



(a)



(b)

Figure 8. Temperature contours along the combustor; (a) Contours of temperature in a plane in line with air intakes and (b) Contours of temperature at a plane in between air intakes.

are formed at the ramps of intake and the normal shock in the diffuser is clearly observed and are highlighted in Fig. 7(b).

Shock diamonds because of the under expansion of the nozzle are observed at the exit of the nozzle and are shown in Fig. 7(a). Distribution of temperature along the domain is shown using temperature contours in Fig. 8. Flame holding near to the central v-gutter is observed in Fig. 8(a) and spreading of flame is observed in Fig. 8(b). From Fig. 8, it is observed that the radial spread of flame is less at a plane in line with intakes than at the regions in between intakes, this is because of the high velocities of the flow from the intake. It is observed that the radial v-gutter located in between air intakes is seen to be holding the flame and the flame spread in the radial direction is much better than the radial v-gutter located at a plane in line with air intakes.

Results obtained from CFD are compared with that of the experimental tests of connect pipe mode, and the values are shown in Table 4. The pressure values are normalized with that of inlet static pressure. From Table 4, the values that are obtained from the connect pipe tests are compared with the CFD values.

Table 4. Comparison of results

Parameter		CFD	Expt
PR	%	63.36	-
MCR	-	0.86	-
$P_{0.4}$ (normalised)	kPa	329.6	325.4
P_4 (normalised)	kPa	269.6	278.6
$T_{0.4}$	K	1585	1697
M_4	-	0.54	0.5

A close match in the chamber pressure is observed. Mach number obtained in the CFD simulations is also found to be in close match with the experimental values. Wind tunnel measurements are needed to compare pressure recovery and mass capture values, which are not addressed in this work. A difference of around 100 K in the total temperature is observed. This difference can be attributed to both combustion model in CFD and the error in the measurements.

7. CONCLUSIONS

Reacting flow simulation were carried out using commercially available CFD software over a Liquid Fuel Ramjet combustor. CFD results obtained are found to be with good agreement with the experimental values. In order to obtain the flow parameters at various stations of liquid fuel ramjet combustor, CFD analyses is carried out for the entire propulsion system. All complex geometrical features are resolved and a mesh of 12 million cells is generated. To resolve compressibility of flow, density is treated to be varying as a function of temperature. Combustion chemistry of Jet A / air is modelled using global three step mechanism for $C_{12}H_{23}$ with six species. CFD analyses is carried out for one of the operating condition. Multiple oblique shocks that are formed at the ramps of intake and the normal shock in the diffuser is clearly observed in the Mach contours. Distribution of the temperature and the spread of flame is seen are observed through temperature contours. Chamber parameters obtained through the connect pipe tests are compared with that of CFD values and a reasonable match is observed between CFD and experiments. This work forms the basis for arriving at modification in flame holder and injection geometry to increase combustion efficiency of ramjet combustion chamber.

REFERENCES

1. Soltani, M.R.; Sepahi, Younsi, J. & Daliri, A. Performance investigation of a supersonic air intake in the presence of the boundary layer suction. *J. Aerospace Eng.*, 2015, **229**(8), 1495-1509.
doi: 10.1177/0954410014554815
2. Choong-won, Lee.; Su-yeon, Moon. & Chang-hyun, Sohn. Spray and combustion characteristics of a dump-type ramjet combustor. *KSME Int. J.*, 2003, **17**(12),2019-2026.
doi:10.1007/BF02982442
3. Srinivasa, Rao, M.; Sivaramakrishna, G.; Muthuveerappan, N.; Kishore, Kumar, S. & Venkataraman, Shankar. CFD analysis of an aero gas turbine combustor-validation and experiments. *In* Proceedings of International Conference and XX National Conference and on Internal Combustion Engines and Combustion (ICONICE), 2007.
4. Golovachov, Yu, P.; Kurakin, Yu, A. & Schmidt, A. A numerical investigation of non-equilibrium MGD flows in supersonic intakes. *AIAA*, 2001, 2883
5. Srinivasa, Rao, M.; Vivek, R.; Trivedi , Darshan, Kumar & Jena, Sunita, Devi. Performance predictions of liquid fuel ramjet combustor. *In* 30th International Symposium on Ballistics, 2017.
doi: 10.12783/ballistics2017/16874
6. Singh, Raja; Thangadurai, G.; Chandran, Subhash B, S.; Babu, V. & Sundararajan, T. Numerical analysis of integrated liquid ramjet engine. *Defense Sci. J.*, 2008, **58**(3), 327-337.
7. Bhandarkar, Anand; Basu, Souraseni; Manna, P. & Chakraborthy, Debasis, Aerodynamic characterization of ramjet missile through combined external-internal computational fluid dynamics simulation. *Defence Sci. J.*, 2016, **66**(6), 624-629
doi: 10.14429/dsj.66.9677
8. Chandra Murty, MSR. & Chakraborthy, Debasis. Coupled external and internal flow simulation of a liquid fueled ramjet vehicle. *Aerospace Sci. Technol.,* 2014, **36**(1-4), 20140.
doi: 10.106/j.ast.2014.03.011
9. Fedina, E.; Fureby, C.; Borzov, S.A.; Gusev, V.N. & Stepanova, T.V. Combustion LES of CESAR multi-burner annular combustor. *AIAA* , 2011,0785.
10. Lee, K.; Aalburg, C.; Diez, F.J.; Faeth, G.M. & Sallam, K.A. Primary breakup of turbulent round liquid jets in uniform cross flows. *AIAA*, 2017, **45**(8), 1907 -1916.

CONTRIBUTORS

Mr M. Srinivasa Rao obtained his MTech from National Institute of Technology (NIT) Bhopal. His research interests include: Computational fluid dynamics (CFD) and propulsion.
In the present study, he has made problem statement, carried out simulations and performed the analyses.

Mr Abhishek Richhariya obtained his MTech from IIT. His research interests include: Design and testing of ramjet propulsion systems.
In the present study, he has identified the technical assignment, defined the problem and was involved in performance analysis.