World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:10, No:12, 2016

Wall Heat Flux Mapping in Liquid Rocket Combustion Chamber with Different Jet Impingement Angles

O. S. Pradeep, S. Vigneshwaran, K. Praveen Kumar, K. Jeyendran, V. R. Sanal Kumar

Abstract**—**The influence of injector attitude on wall heat flux plays an important role in predicting the start-up transient and also determining the combustion chamber wall durability of liquid rockets. In this paper comprehensive numerical studies have been carried out on an idealized liquid rocket combustion chamber to examine the transient wall heat flux during its start-up transient at different injector attitude. Numerical simulations have been carried out with the help of a validated 2d axisymmetric, double precision, pressure-based, transient, species transport, SST k-omega model with laminar finite rate model for governing turbulent-chemistry interaction for four cases with different jet intersection angles, viz., 0°, 30°, 45°, and 60°. We concluded that the jets intersection angle is having a bearing on the time and location of the maximum wall-heat flux zone of the liquid rocket combustion chamber during the start-up transient. We also concluded that the wall heat flux mapping in liquid rocket combustion chamber during the start-up transient is a meaningful objective for the chamber wall material selection and the lucrative design optimization of the combustion chamber for improving the payload capability of the rocket.

Keywords**—**Combustion chamber, injector, liquid rocket, rocket engine wall heat flux.

I. Introduction

THE injector design influences overall engine performance, throttle capabilities, chamber wall durability and combustion stability of liquid rockets. Note that one of the most important subsystems of modern liquid propellant rocket engines is the injector head. The influence of injector attitude on wall heat flux involved in the liquid rocket combustion chamber plays an important role in determining the chamber wall durability and further deciding the type of insulating material required to protect the chamber from the corrosion and many studies are reported over the years. The wall heat flux varies with the angle of the injector. After a leap in the technology, scientists found out that most of the rocket failures are associated with the flaws in the combustion chamber due to over pressure during its start-up transient. Typically, rocket combustion chambers are normally operated at higher pressure, ranging from 1 to 20 MPa. These internal high pressures produces longitudinal as well as hoop stresses at the thrust chamber walls [13]. Additionally, due to the high temperatures created in the rocket engines the wall materials tend to significantly lower the working tensile strength. Furthermore, the most severe thermal gradients, that occurs mainly during the start, cause severe thermal strain and local yielding [13], [14]. This is very critical at high wall heat flux conditions. As a result, the hoop stresses are created and we conjectured that due to the high temperature involved inside the combustion chamber the tensile strength of the material gets lowered leading for a possible catastrophic failure.

Note that rocket combustion chambers are subjected to severe thermal loads, due to the thermal gradient developed during the combustion. This, in turn, requires the components of a thrust chamber assembly (injector head, chamber side walls, and nozzle walls) to be cooled. Hence the proper design of a rocket combustion chamber needs the cognizance of the heat fluxes within the chamber [1]-[3]. The piled up experience (experiments and numerical computations) with the conventional impinging and co-axial injectors is sufficient for combustion chamber design. Nevertheless, the existed knowledge on wall heat flux mapping is not adequate for a lucrative design of a liquid rocket engine. The earlier researchers tried to predict the performance of the rocket thrust chamber by implementing a two-step method. First the chemical procedure is carried out and then thermodynamic formulation was done. The presumed Pdf equilibrium model is in agreement with the cold flow and hot fire model [4]. To find the distribution of the atomized fuel it is necessary to provide the spray input such as injector angle, penetration size and droplet size for numerical model. But to overcome this difficulty, Eulerian coupling strategy is used for primary atomization of the liquid-propellant [5]. The kinetic model of hydrogen combustion chamber is carried out by several investigators and reported that numerical results are in excellent agreement with the experimental observations [6]. The hydrogen flame speed observations are done by using the various coefficients and parameters [7]. Thus, in order to reduce the cost and increase the reliability of the combustion chamber it is necessary to focus on primary design and cost reducing technologies.

The Mascotte model has been developed to understand the chemical process involved inside the combustion chamber of rocket engine [8]. The combustion at high pressure is an

O. S. Pradeep, K. Praveen Kumar are Undergraduate Students at the Department of Mechanical Engineering, Kumaraguru College of Technology (Affiliated to Anna University Chennai) Coimbatore – 641 049, Tamil Nadu, India (phone: +91-9566789323, +91-9003958399; e-mail: spradeepkct@gmail.com, praveen4u.15@gmail.com).

S. Vigneshwaran is an Undergraduate Student at the Department of Aeronautical Engineering, Kumaraguru College of Technology (Affiliated to Anna University Chennai) Coimbatore – 641 049, Tamil Nadu, India (phone: +91-8903735717; e-mail: vigneshsankar11@ gmail.com).

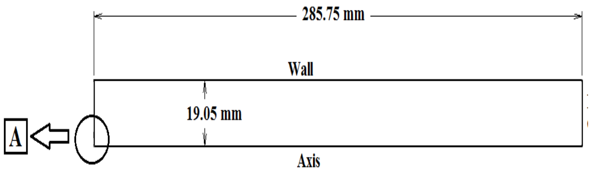
K. Jeyendran is an Undergraduate Student at the Department of Mechanical Engineering, United Institute of Technology (Affiliated to Anna University Chennai) Coimbatore – 641 020, Tamil Nadu, India (phone: +91-9585731710; e-mail: jeyendran2013uit@gmail.com)

V. R. Sanal Kumar is Professor and Aerospace Scientist, currently with Aeronautical Engineering Department, Kumaraguru College of Technology, Coimbatore – 641 049, affiliated to Anna University, Tamil Nadu, India (phone: +91-8754200501/9388679565; e-mail: vr_sanalkumar@yahoo.co.in).

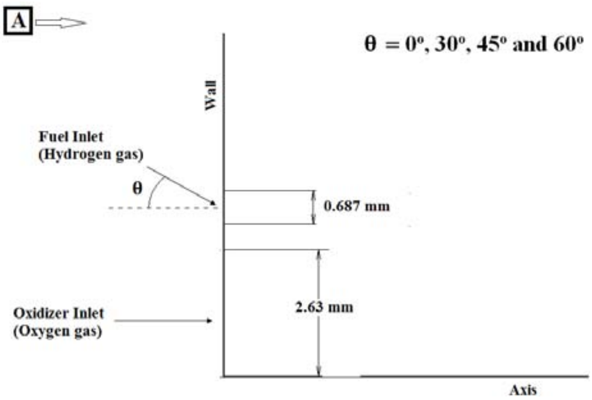
important issue for propulsion and it is done by flamelet-progress-variable (FPV) turbulent combustion model combined with Reynolds Averaged Navier-stokes equation Solver (RANS). It showed a way for understanding the effects of real and importance of kinetic scheme [9]. The data provided from Mascotte experiment is unreasonable and an attempt is made to numerically simulate the data provided by IWRCM. It is found that both homogeneous and heterogeneous mixtures match the data [10]. Modelling the trans- and supercritical mixture in the combustion chamber involves difficulty and thus numerical models can be done only with the better understanding of coupling of fluid dynamics, chemical kinetics and acoustics [11]. The regenerative cooling of methane can also, combined with the LO2, enhance the performance of combustion [12].

Literature review further reveals that the liquid rocket combustion chamber designers tried various types of injectors along with different attitudes. In the shower type injector, having zero jet intersection angle (θ = 0), the combustion may somewhat delayed, compared to other jet impinging types (0 < θ < 180), due to the parallel injection of the oxidizer and fuel to the combustion chamber. Note that, by altering the jet intersection angle of fuel and oxidizer, one can ensure the better combustion characteristics through proper mixing within the allowable residence time of the fuel and oxidizer.

In this paper, we varied the injector angle of fuel and oxidizer for mapping the heat flux with respect to the corresponding injector attitude for lucratively designing the combustion chamber by supplementing suitable materials in appropriate regions for withstanding the corresponding heat flux during the entire mission.

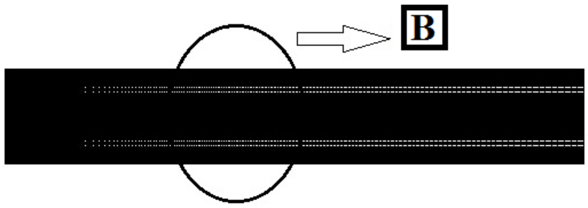


(a) An Idealized Liquid Rocket Combustion Chamber

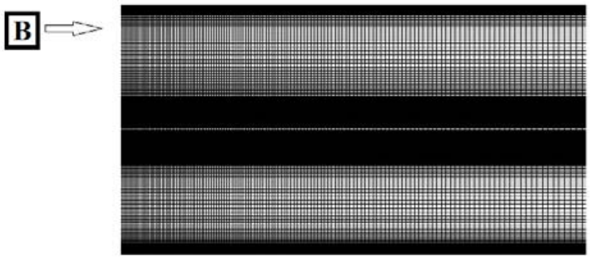


(b) Enlarged view of the near injector region marked as A in (a)

Fig. 1 An Idealized Physical Model of a Liquid Rocket Combustion Chamber



(a) Grid System in the Computational Domain



(b) Enlarged view of the Grid System in the Middle Region

Fig. 2 Structured Grid System in the Computational Domain

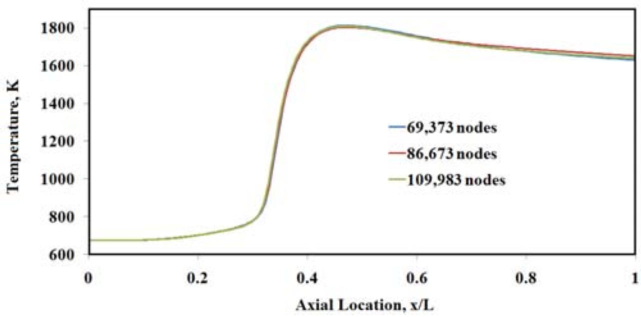


Fig. 3 Comparison of axial temperature profiles for different mesh sizes

II. Overview of the Numerical Methodology

In this paper, numerical studies are carried out with the help of a validated transient, axisymmetric and two-dimensional SST k-omega turbulence model with Laminar Finite Rate model. The selected turbulence and combustion model are based on the literature input. In the numerical model, a control volume based technique is used to convert the governing equations to algebraic equations. The viscosity is determined from Sutherland formula. In all the cases, compressible subsonic mass inflow condition is prescribed. Fig. 1 shows the physical models of the Liquid Rocket Combustion Chamber considered in the numerical analysis. Fig. 2 shows the structured grid system in the computational domain of the corresponding selected physical model shown in Fig. 1. A typical structured grid system in the computational domain is selected after a detailed grid refinement exercises. The grids are clustered near the walls and the propellants mixing region along the axis of the combustion chamber using suitable stretching functions. Initial wall temperature, inlet static pressure and temperature are specified. At the solid walls a no slip boundary condition is imposed. Fig. 3 shows the grid refinement exercises representing the axial static temperature comparison for three different mesh sizes. It is evident from

Fig. 3 that the variation of axial temperature, for the mesh size 86,673 and 109,983, is negligible. Hence the medium mesh size of 86,673 nodes is selected for the further analysis.

In the parametric analytical studies, the combustion chamber wall heat flux has been examined by varying the attitude of the fuel injector by four different angles viz., 0°, 30°, 45° and 60°. For computational convenience, an idealized injector head is selected. The different orientations of the flow are achieved by resolving the mass flow component in x and y directions. This resolved component is given as inputs for further numerical analysis.

III. Results and Discussion

In this paper, the heat flux produced due to the combustion in a liquid rocket combustion chamber is measured numerically for different injector attitude with the same boundary conditions. Each case differs in the orientation of the fuel injector that is the injection angle for fuel is varied from 0 degree (co-axial) to 60 degree through 30° and 45^0. The propellants selected for the analysis are Gaseous hydrogen as fuel with Gaseous oxygen as oxidizer. Since distance between the fuel and oxidizer holes is very close (see Fig. 1), the jet interaction happened for coaxial jets in the upstream region of the combustion chamber and the combustion took place spontaneously. The heat transfer to the combustion chamber wall due to the combustion is measured in the form of heat flux. A fixed temperature of 300 K is applied to the wall and the heat flux (q) to the wall from a fluid cell is calculated from the convective heat transfer coefficient h_f, local fluid temperature (T_f) and wall temperature (T_w) based on:

$$q = h_f \, (T_f - T_w) \tag{1}$$

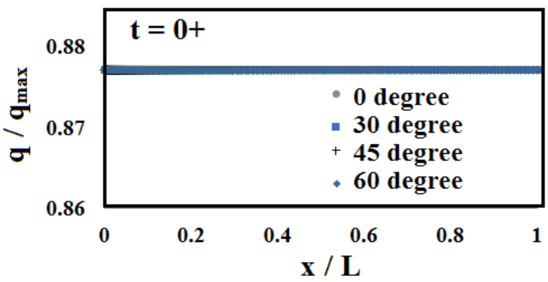
The fluid-side heat transfer coefficient, h_f is computed based on the local flow-field conditions.

Table I corresponds to the boundary conditions selected for the numerical analysis. Combustion between gaseous hydrogen and gaseous oxygen produce water vapor. This reaction is a single step reaction taking place at finite rate. For the selected liquid rocket combustion chamber, analysis was performed by varying injector attitude for fuel and corresponding heat fluxes are measured at the wall. The non-dimensionalized heat fluxes at different times ($t = 0^+$ - 10 ms) for four different cases are plotted in Figs. 4 (a)-(e). The high heat flux value at t = 0+ ms is due to the initialization of the computational domain before beginning the computation.

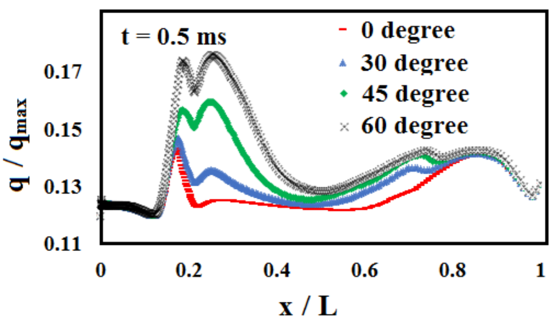
Among the four cases (θ = 0°, 30°, 45° and 60°) the maximum heat flux measured is 3.29697 MW/m^2, which is measured in the case of 60° orientation of fuel injector at 10 ms. Similarly, the chamber wall location is non-dimensionalized against the full chamber length of 0.28575 meter. It can be inferred from Figs. 4 (a)-(e) that the variation of fuel injector angle causes significant variation in the heat flux at the chamber wall. It can be deduced from Fig. 4 (e) that there is an oscillation in the peak heat flux region due to variation in injector attitude. We observed that, at the beginning of the start-up transient, when the time advances the maximum wall-heat flux zone of the liquid rocket engine shifted towards the nozzle end for the cases with higher jet intersection angles and further it shifted back towards the injector head. Nevertheless, as seen in Figs. 5 (a)-(e), there were no specific trends shown for the peak temperature zone for the axial combustion gas possibly due to the non-linear relation between the jet intersection angle and the mixing length within the given envelop. Figs. 5 (a)-(e) represent the variation of axial static temperature for different cases at different times with same boundary conditions. We discerned that there is a shift in the peak temperature region at each time and also this shifting is unsteady, unlike the heat flux at the chamber wall. Fig. 5 (f) provides the enlarged view of the temperature profile at 10 ms, which is derived from Fig. 5 (e). Note that lowest axial temperature is corresponds to a case with jet intersection angle of 30° and highest temperature is corresponding to coaxial injection. This unsteadiness may be due to the turbulent combustion nature of the propellants being injected with different jet intersection angle and the corresponding flow properties prior to mixing and combustion with finite rate reaction, which are corroborated in Figs. 6 (a)-(d) and 7 (a)-(d) through pressure and velocity contours for four different cases at two different time intervals.

TABLE I
BOUNDARY CONDITIONS FOR THE LIQUID ROCKET ENGINE

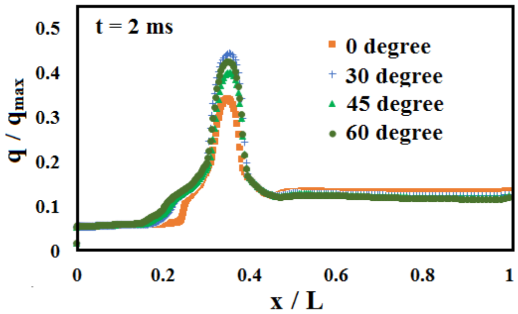
Property (Inlet)	Hydrogen	Oxygen
Mass flow rate	0.0331 kg/s	0.0904 kg/s
Static Pressure	6 MPa	5.85 MPa
Total Temperature	811 K	700 K
Hydraulic Diameter	1.374 mm	5.26 mm
Turbulent Intensity	5 %	5 %
H_2O Mass fraction	0.598	0.055



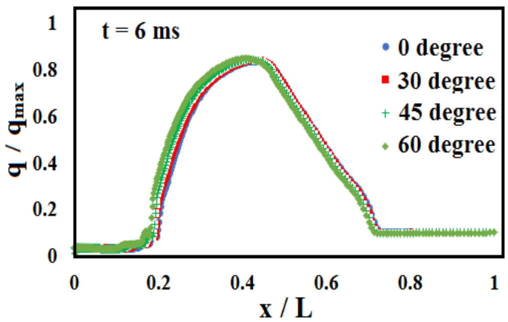
(a) Time 0^+ ms



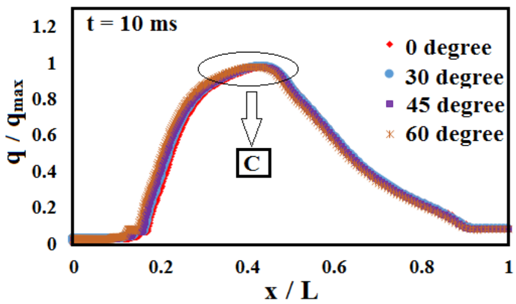
(b) Time 0.5 ms



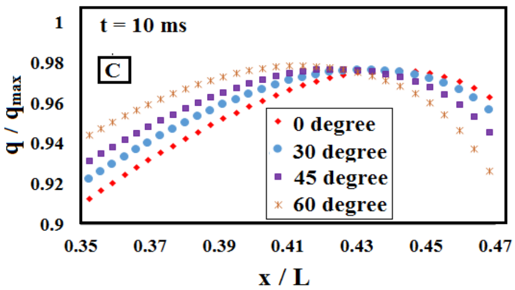
(c) Time 2 ms



(d) Time 6 ms

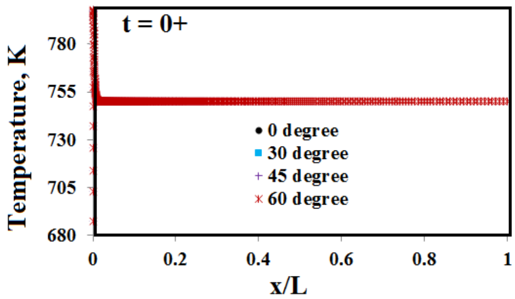


(e) Time 10 ms

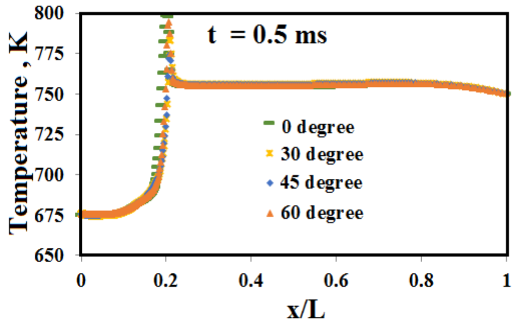


(f) Enlarged view of (e) at time 10 ms.

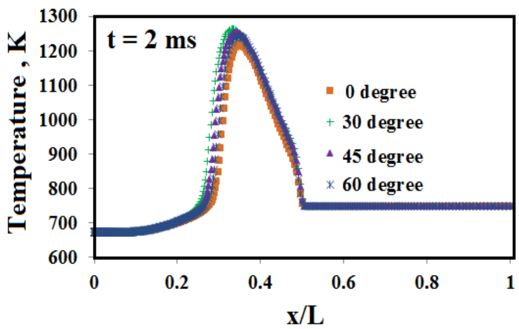
Fig. 4 Wall Heat Flux along the axial direction at different times



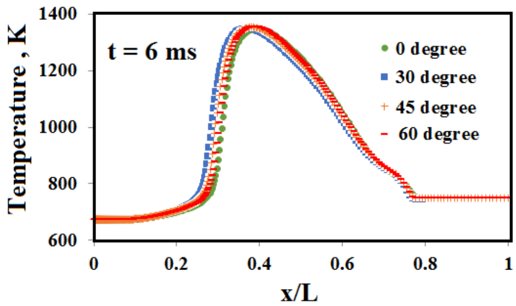
(a) Time 0^+ ms



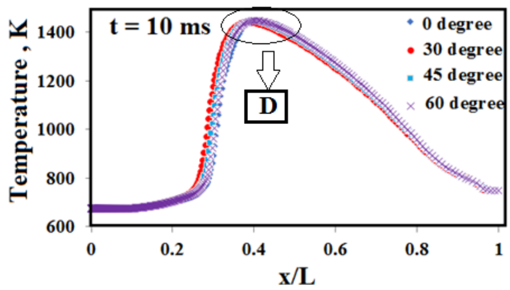
(b) Time 0.5 ms



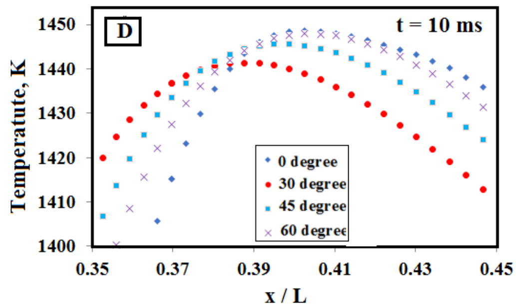
(c) Time 2 ms



(d) Time 6 ms



(e) Time 10 ms



(f) Enlarged view of (e) at Time 10 ms

Fig. 5 Axial Temperature comparison for different cases at different times

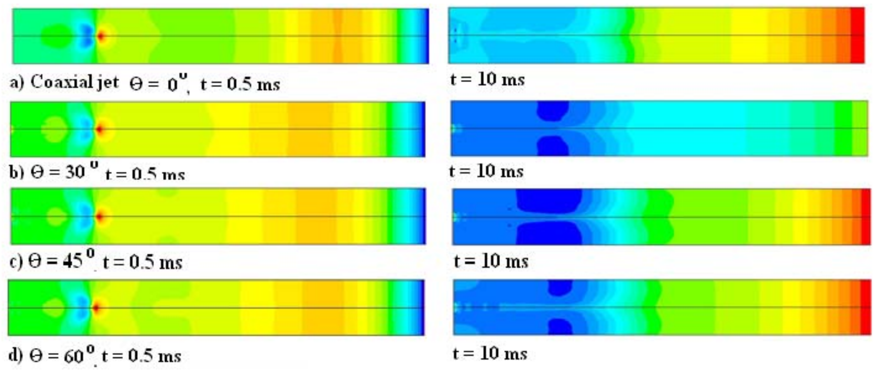


Fig. 6 Comparison of pressure contours for different cases at two different times

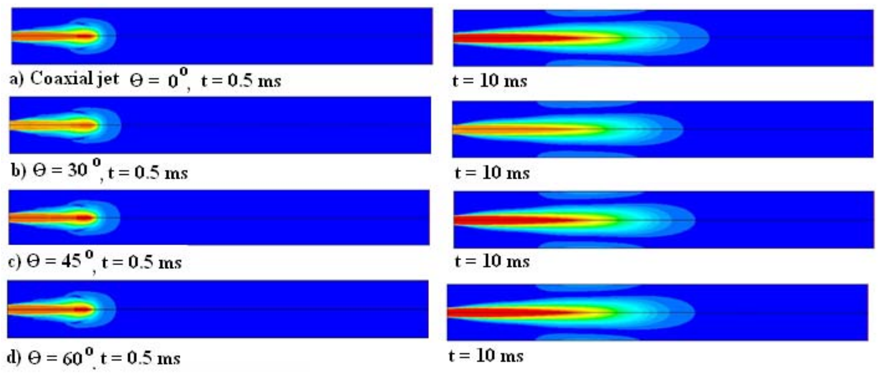


Fig. 7 Comparison of velocity contours for different cases at two different times

IV. Concluding Remarks

The wall heat flux mapping in an idealized liquid rocket combustion chamber has been carried out using a validated laminar finite rate model for governing turbulent-chemistry interaction with the conventional flow model. Compared to the classical coaxial injection the flow characteristics and wall heat flux show promising characteristics with variable jet intersection angles for meeting the future design objectives of liquid rocket engines. Therefore, future investigations should focus on the influence of injection conditions, injector attitude and injector geometries on the atomization and mixing. This study throws light for exploring all options for an accurate estimation of heat flux distribution along the liquid combustion chamber wall, with proper turbulent combustion model, for its lucrative design optimization.

Acknowledgment

The authors would like to thank the Management of Kumaraguru College of Technology, Coimbatore – 641 049, Tamil Nadu, India for their extensive support for completing this research work.

References

[1] Victor P.Zhukov, Dmitry I.Suslov, “Measurements and modelling of wall heat fluxes in rocket combustion chamber with porous injector head,” Aerospace Science and Technology 48 (2016) 67–74.

[2] J. Lux, D. Suslov, O.J. Haidn, On porous liquid propellant rocket engine injectors, Aerosp. Sci. Technol. 12 (2007) 469–477.

[3] Guiliano R, luigi Cutrone, Daniele Cardillio, “Numerical Investigation of Rocket Engine Combustion Flow Fields”, AIAA-2016-2146.

[4] A. Benarous, D. Karmed, R. Haoui, and A. Liazid “An Attempt to Predict The Performance Of A Rocket Combustion Chamber”, World Academy of Science, Engineering and Technology International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering Vol: 6, No: 11, 2012.

[5] A. Murrone, N. Fdida, C. Le Touze, L. Vingert, “Atomization of Cryogenic Rocket Engines Coaxial Injectors. Modeling Aspects and the Experimental Investigations”, Space propulsion 2014, May 2014, Cologne, Germany.

[6] Hagen Müller, Christoph A. Niedermeier, Jan Matheis, Michael Pfitzner and Stefan Hickel, “Large-eddy simulation of nitrogen injection at trans and supercritical conditions,” Physics of Fluids 28, 015102 (2016).

[7] Juan Li, Zhenwei Zhao, Andrei Kazakov, and Frederick L. Dryer, “An Updated Comprehensive Kinetic Model of Hydrogen Combustion”, *International Journal of Chemical Kinetics* (Vol. 36, p. 566-575 (2004)

[8] Mani Pourouchottamane, Francis Dupoirieux*, Lucien Vingert, Mohammed Habiballah, and Victor Burnleyt, “Numerical Analysis of the 10 Bar Mascotte Flow Field”, ONERA - B.P. 72 - 92322 Chatillon Cedex – France WAir Force Research Laboratory.

[9] A. Coclite, L. Cutrone, P. De Palma, G. Pascazio “Numerical investigation of high-pressure combustion in rocket engines using Flamelet/Progress-variable models” AIAA, 2 December 2014.

[10] Richard Farmer, Gary Cheng, Yen-Sen Chen, “CFD Simulation of Liquid Rocket Engine Injectors Part 2. Simulations of the RCM-2 Experiment”, Atomization, Combustion and Heat Transfer held in Lampoldshausen, Germany on 25-27 Mar 2001.

[11] A. Depoutre, S. Zurbach, D. Saucereau, J.P. Dumont, E Bodele, I. Gokalp, “Numerical calculation of MASCOTTE 60 bar case with THESEE”, Atomization, Combustion and Heat Transfer held in Lampoldshausen, Germany on 25-27 Mar 2001.

[12] Daniele Cardillo, Daniele Ricci, Pietro Roncioni, Francesco Battista, “Numerical Simulation of a LO2-CH4 Rocket Engine Demonstrator”, 65th International Astronautical Congress, Toronto, Canada, Sept. 2014.

[13] George P. Sutton & Oscar Biblarz *(2001).* Rocket Propulsion Elements (7th ed.). *Wiley Interscience. ISBN 0-471-32642-9.*

[14] Dexter K Huzel and David H. Huang (1971), NASA SP-125, Design of Liquid Propellant Rocket Engines.