International Journal of Thermofluid Science and Technology (2021)
Volume 8, Issue 4, Paper No. 080401
https://doi.org/10.36963/IJTST.2021080401
Research Article

Study of Sonic jet and Shock Boundary Layer Interaction using CFD

P. Nithish Reddy[1*], K. Venkatasubbaiah[2]

[1] *Sreenidhi Institute of Science and Technology -Yamnampet ,Gatkesar 501301,Hyderabad,India*
[2] *Indian Institute of Science and Technology Hyderabad ,Sangareddy-502 285,India*

ATICLE INFO

Received: 12 June 2021;
Received in revised form:
12 Sept. 2021;
Accepted: 22 Sept. 2021;
Published online:
10 Oct. 2021

Keywords:
Supersonic flows
Recirculation regions
Mach disc
Sonic Injection
Shock-BL Interaction

ABSTRACT

Numerical investigations are carried out to capture the near-field interaction between the sonic jet and supersonic cross-flow. Such interactions are most commonly seen in various high-speed propulsion applications. The results obtained using different turbulence models are validated with the experimental results. The parametric investigations include the study of recirculation regions, boundary layer separation, and shock-induced wall pressure distribution for a wide range of operating conditions. As the free stream Mach number increases, the length of the separation regions varies and similarly the strength of shock structures and the peak pressure. Injection pressure ratio and jet injection angle significantly impact the formation of local ignition zones and, therefore, combustion.

1. Introduction

The transverse jet injection in supersonic cross-flow is an important phenomenon to study as it has several applications like fuel injection in scram-jet combustors and other space vehicles. The transverse jet injection is widely applied in scramjet applications as it gives a high degree of fuel-air mixing in supersonic flows [1-5]. The interaction of the jet with the supersonic cross-flow results in complex shock wave boundary layer interaction (SWBLI). Shock waves generate an adverse pressure gradient, boundary layer separation ahead of the injector, and recirculation regions in the wake of the injection slot [6,7]. The recirculation regions provide a constant source to sustain the flame, and it improves combustion efficiency. Fric and Roshka [8] studied vortical structures involved in transverse jet injection, especially low-speed flows. These results revealed four coherent structures: one vortex wrapping around the jet, second in the downstream wake, third near field jet-shear layer vortex, and fourth far-field counter-rotating vortex pair. After realizing the importance of these stream-wise counter-rotating vortices, many researchers concentrated on different injection strategies focusing on the generation of these swirling structures [9,10]. Riggens et al. [11] studied fuel injection angle on scramjet combustion at high flight Mach numbers. Results showed that thrust force is enhanced by injecting the fuel at 30 degrees for Mach 13.5 and Mach 17. They concluded that the fuel jet momentum added to the net thrust by angled injection in high-speed flows.

[*] *Corresponding e-mail*: dr.nithish.reddy@gmail.com (P. Nithish Reddy)

Nomenclature

CFD	Computational Fluid Dynamics	*RNG*	Renormalization Group
DES	Detached Eddy Simulation	*SST*	Shear Stress Transport
DNS	Direct Numerical Simulation	*BL*	Boundary Layer
M	Mach Number		
N_2	Nitrogen	*Subscripts*	
N- S	Navier Stokes	w	Value at Wall
P.R	Pressure Ratio	sp	Seperation Length
SW	Shock Wave	∞	Free Streem Value

The later part of the research was concentrated on the influence of different fuel types on the formation of these structures and jet penetration. The conclusions were the molecular weight of fuel shown a big variation in injection velocities which influences the growth rate of the jet shear layer and the mixing properties. Several studies on jet penetration revealed that the molecular weight of fuel has not much effect on jet penetration [12,13]. However, efficient fuel-air mixing, jet penetration does not directly initiate the combustion process; ignition and flame holding also play a major role [14,15]. The upstream recirculation region and recirculation regions behind the bow shock are potential auto-ignition regions[16]. Kyung et al. [17] carried out numerical investigations on combustion enhancement when a cavity was used for the hydrogen fuel injection through a transverse slot nozzle into a supersonic hot air stream. Huang et al. [18] reported the influences of the turbulence model and the slot width on transverse slot injection. Khali and Yao [19] carried out a numerical study on perpendicular injection of the sonic jet into supersonic cross-flow using detached-eddy simulation (DES).

Yousuf and Farrukh [20] conducted an experimental study on the effect of a micro-jet array into a supersonic cross-flow at Mach 1.5; from the results, they observed that the counter-rotating vortices merge as the spacing between the jets is reduced. Nithiyaraj and Govardhan [21]conducted a particle image velocimetry study on the jet mean and instantaneous flow fields. They successfully visualized the formation of Mach disc and shear layer between jet and cross-flow. Ruofan et al. [22], for the first time, simulated the jet injection into supersonic cross-flow using Partially-averaged Navier-Stokes (PANS) method, which is a bridging method between Reynolds-averaged Navier-Stokes (RANS) and direct numerical simulation (DNS). Soni and Ashoke [23] numerically analyzed hydrogen and ethylene injection into the supersonic flow from the back of the strut. They reported the effect of Strut geometry and jet diameter on the mixing of air and fuel. Anazadehsayed et al. [24] studied the effect of micro air jet on the mixing characteristics of the hydrogen jet into the supersonic cross-flow. From various parametric studies using RANS model, they observed that when an air jet is present downstream of the hydrogen jet, the mixing of fuel in the downstream enhances significantly (up to 300%). Various methods have been proposed to enhance the mixing process between fuel and air [25]. Several studies revealed a detailed understanding of the near field means flow structure in a single jet-supersonic cross-flow. [26,27]. Pudsey and Boyce [28] numerically studied the effects of sonic gaseous hydrogen injection through multiple transverse injectors subjected to a supersonic cross-flow. Results showed improvements of up to 5% in the overall mixing efficiency due to increased mixant interface area and intermediate stirring through wake vortices between each injector. Franklin and Suresh [29] used a large eddy simulation to analyze the interaction between a sonic air jet and a supersonic air cross-flow. Dickman [30] numerically studied a transverse injection into cross-flow by the finite volume method. They reported that the shock/boundary-layer interaction alters the flow greatly near the sonic jet.

Literature indicates that the recirculation region downstream and upstream of the jet in the transverse injection system has major importance. In these regions, the velocity is low, and flame sustainability is high. The formation of such recirculation zones depends on both the jet and free stream conditions. Here, numerical investigations are carried out on the effects of injection angle, free stream Mach number, and injection pressure on flow field characteristics. Three different turbulence models, such as renormalization group RNG k-ϵ model, Reynold stress model, and shear stress transport SST k-ω model, are considered to validate the current results with that of experimental.

2. Physical model

For validating numerical methods, current results are compared with experimental results available in the literature. Aso and Okuyama [3] experimentally studied the mixing phenomena in supersonic flows with slot injection. The experimental model consisting of a flat plate with a slot, as shown in Fig. 1.

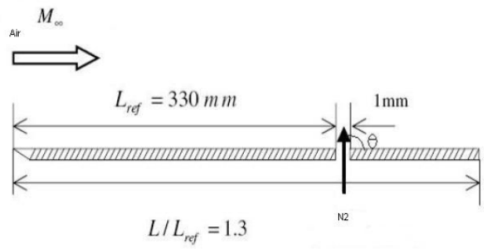

Fig. 1 Schematic diagram of transverse jet injection

The distance from the plate leading edge to the slot is L_{ref}= 330 mm, slot width is 1 mm. The total length of the plate is 429 mm. The operating conditions chosen are free stream Mach number =3.75, the total pressure = 1.2 MPa, and the total temperature =299 K. Slot nozzle designed to have a convergent sonic throat at the exit. Nitrogen gas is injected at a Mach number of 1 and pressure ratio (P.R) of 10.29. The pressure ratio is defined as the ratio of total injection pressure to free-stream static pressure.

3. Numerical methodology

Two dimensional Reynolds averaged Navier-Stokes equations (RANS) are solved by the Finite volume method. Turbulence is modeled with three different turbulence models: RNG k-ϵ model, Reynold stress model, and shear stress transport SST k-ω model[31]. Air is considered an ideal gas with variable properties. Sutherland law is used to calculate the viscosity, and piecewise polynomial is used to calculate temperature-dependent specific heat. The boundary conditions at the inflow are specified as free stream operating conditions, and the flow variables at the outflow are extrapolated from the interior. No-slip boundary conditions are imposed at the solid walls for the velocity field. The adiabatic boundary condition is used at the solid walls for the temperature field. Implicit formulations with second-order upwind schemes are applied to solve the equations. A fine grid is used near the injector and walls. To capture the jet and shock/boundary-layer interactions wall $y+ <5$ is realized.

4. Validation and Grid independence

The accuracy of the present numerical model is tested by validating with the experimental results available in the literature. Details of the empirical model and operating conditions are given in section 2. In Figure 2, the numerically predicated wall pressure is plotted and compared with the experimental data by Aso and Okuyama [3]. Present results are reasonably accurate with the experimental results. Minor discrepancies maybe because of the limitations in pressure probes placement in supersonic flows as they may create secondary shocks in the experimental setup. Thus the experiments do not record sharp peaks just upstream of the injection slot. Present results obtained by the RNG k- ϵ turbulence model show better agreement with the experimental data than the other turbulence models, namely the Reynolds stress and SST k- ω turbulence models. At the same time, the length of the upstream separation region and the peak pressure upstream of the injection slot is underestimated by the Reynolds stress model and the k- ω turbulence model. The recirculation zone plays a significant role in chemically reacting flows. In this region, the velocity is low, and the flames can be sustained (Cegal et al. [6]). Thus it is important to capture the upstream recirculation zone. RNG k- ϵ model can predict the length of the upstream recirculation zone; thus, this model is used for the rest of the investigations.

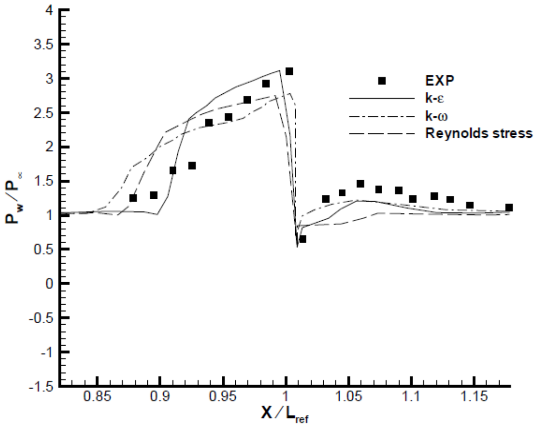

Fig. 2 Comparison of predicted wall pressure with experimental results.

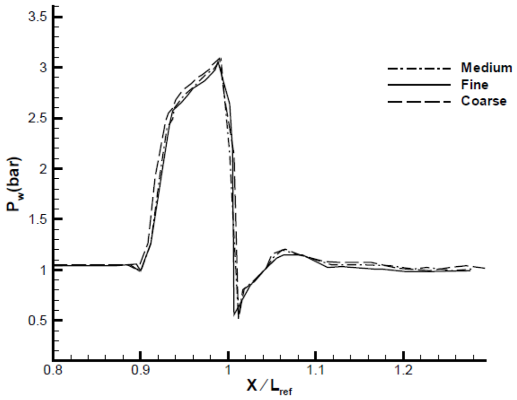

Fig. 3 Plate wall pressure distribution for different grid sizes

Numerical simulations are conducted with three different grid sizes: coarse mesh 8000 cells, medium mesh 15000 cells, fine mesh 25000 cells, and shown in Fig. 3. In Fig. 3, the wall pressure distribution is plotted, and results show that the length of the upstream separation region and other parameters are predicted well by all grid sizes. It is found that there is not much variation in results with different grid sizes; hence coarse mesh is used in the present investigation.

5. Results and Discussion

Transverse jet injection in supersonic cross-flow was studied numerically and reported the results for different injection angles and flow conditions. Air flows over a flat plate at a free stream temperature of 340 K and 1 bar pressure, throughout the investigations. The corresponding total pressure and total temperature are obtained from stagnation relations for each free stream Mach number. Nitrogen gas ($N2$) is injected at sonic speed(M=1) at a static temperature of 250 K.

The effects of injection angle, free stream Mach number, and injection pressure on separation, shock wave/boundary-layer interactions are reported here. The first results are obtained for free-stream Mach number 2, injection pressure ratio (P.R) 10 at an angle of 90 deg(see Fig.4). These results are considered as a reference case to investigate the effect of free-stream Mach number and injection pressure. Pressure and Mach contours of transverse jet interaction in supersonic cross-flow are shown in Fig. 4 (a & b). The injected gas act as an obstruction in supersonic cross-flow and produces the bow shock wave ahead of the injector. This shock wave has an adverse pressure gradient which causes the boundary layer separation regions ahead of the injector, as shown in Fig. 4 (a & b). Mach disc is formed surrounding the jet due to expansion of the jet, and separation regions are observed in the wake of the injection slot; this happened because the flow is turned parallel to the wall downstream of the jet. From Fig. 4, one can notice that the present results are in agreement with experimental observations such as bow shock, Mach disc, and separations regions due to interaction of jet in supersonic cross-flow. In Fig. 4(c), the wall pressure distribution is plotted along the plate length and also marked the separation length ahead of the jet. These separation region details are essential in transverse fuel jet injection systems because of their flame-held capability and auto-ignition in combustion situations.

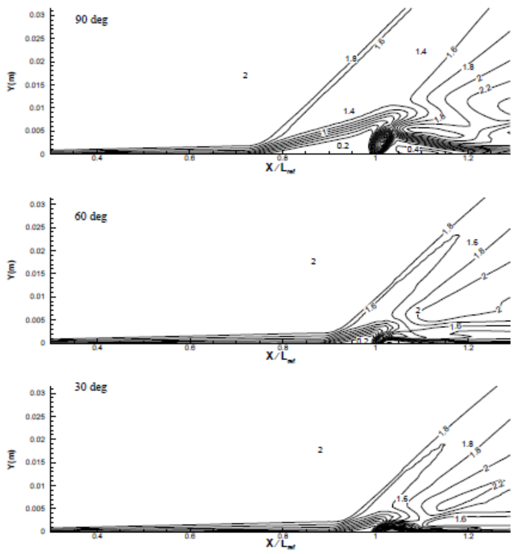

Fig. 5 Comparison of Mach contours at different injection angles in downstream

5.1 Effect of Injection angle

The influence of injection angle is reported here; results at different injection angles are shown in figures 5,6&7 for free-stream Mach number =2 and sonic injection conditions. The injection angle varies from 30 deg to 150 deg. Here Injection angle below 90 degrees indicates that fuel is injected downstream and above 90 degrees indicates that fuel is injected against upstream. In Fig. 5, Mach contours are shown for downstream injection. From the Mach contours, it is observed that both upstream and downstream recirculation regions are larger for 90-degree injection angles, and their size is reduced with a decrease in injection angle from 90 to 30 degrees. The inclined injection is likely to reduce or eliminate auto ignition and stabilization because of the absence of recirculation regions, especially at flight speeds lower than Mach 10.

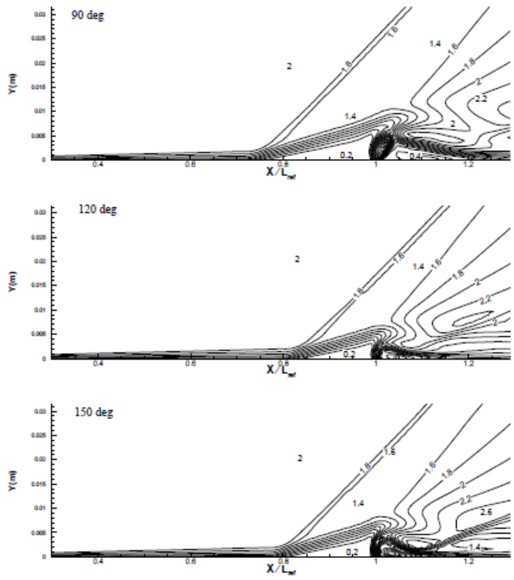

Fig. 6 Comparison of Mach contours at different injection angles against upstream.

In Figure 6, Mach contours are shown for fuel injection upstream. An increase in the angle of injection from 90 to 120 degrees resulted in a decrease in the upstream recirculation region (see Fig 6.). But with a 150 deg case, the upstream recirculation region is observed slightly more than 120 deg. The width of the Mach disc has reduced with injection against upstream due to a reduction in shock strength. Figure 7 compares wall pressure for all different injection angles ranging from 30 to 150 deg. The figure shows that as the injection angle is decreased from 90 to 30 degrees, the peak pressure is reduced. Secondly, the length of the upstream separation region is also reduced. The upstream separation is found more in fuel injection against upstream than the downstream side of fuel injection. The upstream separation length and peak pressure are more with an injection angle of 150 deg than 120 deg. The intensity of pressure disturbances upstream of the injection has reduced when the injection angle is reduced from 90 to 30 degrees. At a higher flight Mach number, the pressure losses due to normal injection are very high. So it would be better to use an angled injection at a higher flight Mach number.

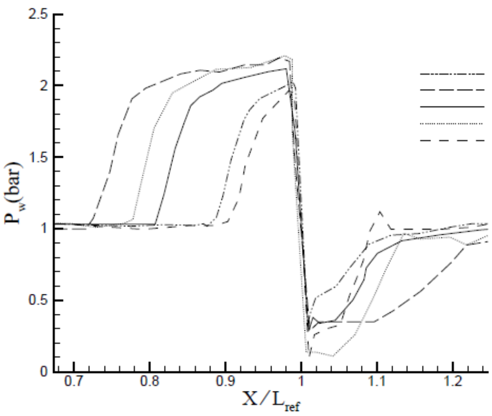

Fig. 7 Comparison of wall pressure distribution at different injection angles.

From Fig. 5, it can also be observed that as the injection angle is reduced from 90 to 60 degrees, the Mach disc is partially deformed and inclined downstream. Whereas for injection angle of 30 deg with a horizontal plate downstream, no Mach disc is observed. The degree of separation of the boundary layer is less as the fuel is injected at a lower angle, indicating that the shock strength has also reduced.

5.2 Effect of free-stream Mach number

The effect of free-stream Mach number are reported here; results at different free-stream Mach numbers are shown in Figures 8,9 &10. In Fig. 8, Mach contours are shown at different free-stream Mach numbers. Fig. 8 indicates that the Mach disc, jet penetration, and height reduce with an increase in free-stream Mach number. The size of both upstream and downstream separation regions decreases with an increase in free-stream Mach number. This can be explained as when the free stream Mach number is increased, the pressure waves generated due to the introduction of the jet won't get sufficient time to travel move against upstream. Thus, the shock waves form closer to the jet as the Mach number is increased. The formation of the upstream separation region is due to adverse pressure gradients created by the shock wave. When the shock is closer to the jet, the size of the separation region also reduces. For a given injection pressure ratio, as the relative speed between the free stream and jet increases, the penetration depth of the jet decreases. Because the normal jet can travel relatively less speed than the main flow due to the higher free-stream Mach number.

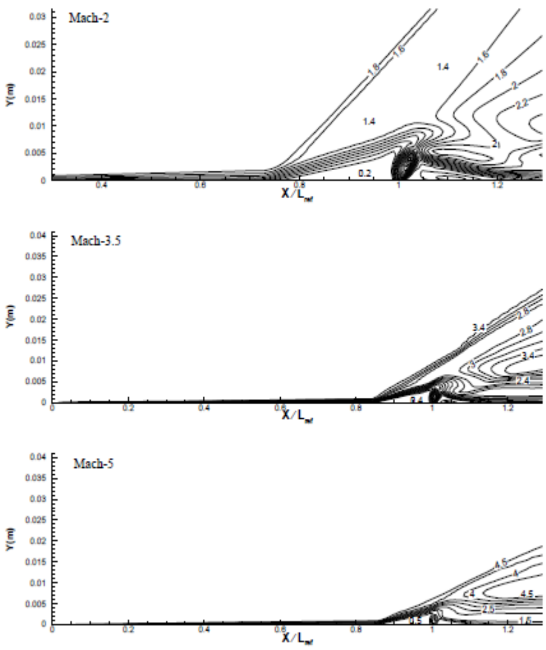

Fig. 8 Comparison of Mach contours at different inflow conditions

In Fig. 9, the pressure contours are shown with different free-stream Mach numbers. From Fig. 9, the width of the pressure rise upstream of the injector is more at free-stream Mach 2 inflow conditions, while the concentrated pressure rise is observed with

Mach 5. In all the cases, expansion is followed by pressure rise, and this expansion is found quick at Mach 5 and is slow at Mach 2. Fig. 10 gives the quantitative comparison of wall pressure with different free-stream Mach numbers. The width of the pressure disturbance is found more at the lower free-stream Mach number.

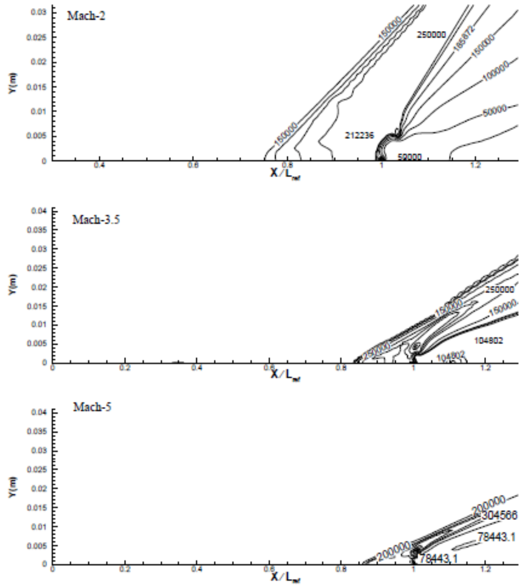

Fig. 8 Comparison of Mach contours at different inflow conditions.

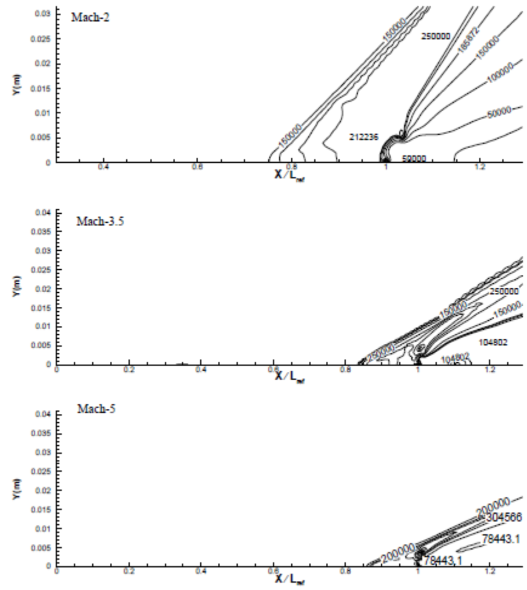

Fig. 9 Comparison of pressure contours at different inflow conditions

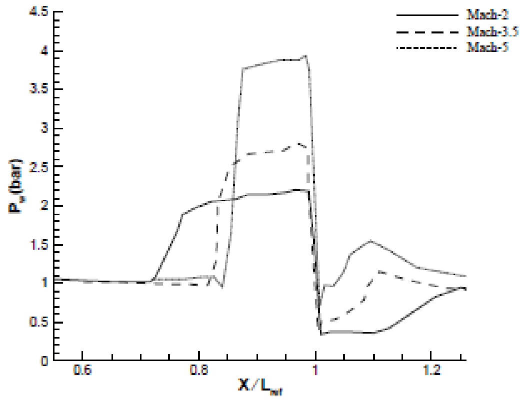

Fig. 10. Comparison of wall pressure distribution for different inflow conditions.

Pressure is low downstream of the injector for free stream Mach 2 inflow condition. Reduction in separation length with a simple wall injection system suggests the need of creating additional local recirculation zones, especially at higher Mach number flows.

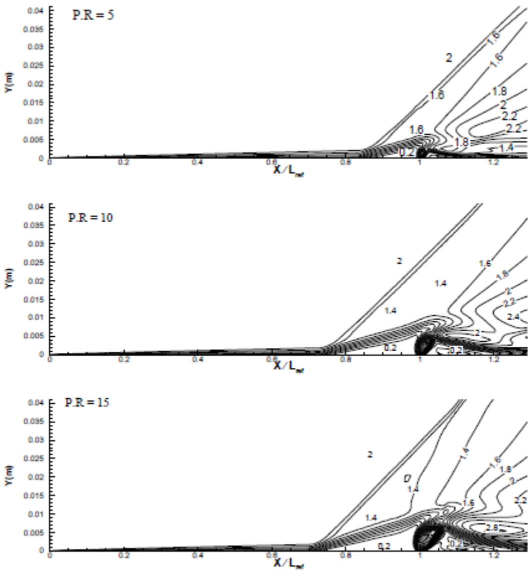

Fig. 11. Comparison of Mach contours for different injection pressure ratios.

5.3 Effect of injection pressure ratio

The effect of injection pressure ratio (P.R) is explored in this section, results at different

injection pressure ratios are shown in Figs. 11-13. In Fig. 11 Mach contours are shown for different injection pressure ratios, the size of the Mach disc increases with an increase in injection pressure.

As the injection pressure ratio increases, the jet will have more pressure energy that can be converted into kinetic energy; thus, the penetration depth and size of the Mach disc increase. Higher injection pressure means the pressure disturbances are more substantial. The pressure waves generated due to the introduction of the high-pressure jet can now travel a larger distance against upstream. Thus as the injection pressure ratio increases, shock forms slightly far from the injection slot, as shown in Fig. 12. An increase in injection pressure ratio resulted in a more significant separation region. The separation starts where the boundary layer encounters a shock wave. Hence both the upstream and downstream separation regions increase with the increase in injection pressure. Fig. 13 gives a quantitative comparison of wall pressure for different injection pressure ratios. Both width of the pressure disturbance upstream and the peak - pressure are higher for higher pressure ratios.

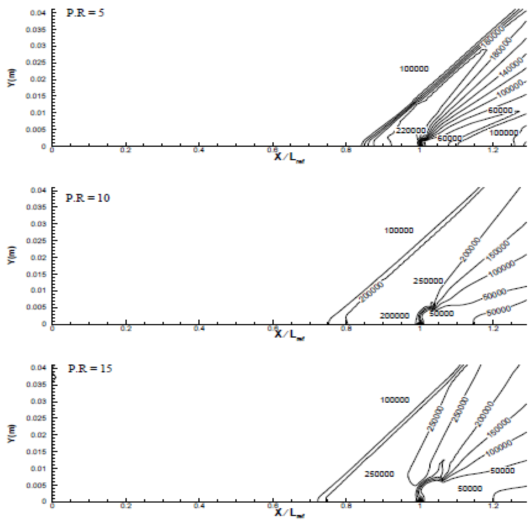

Fig. 12. Comparison of pressure contours for different injection pressure ratios.

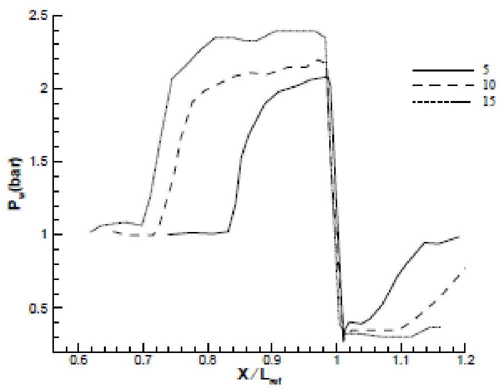

Fig. 13 Comparison of wall pressure distribution for different injection pressure ratios.

6. Conclusions

The jet-induced shock waves and boundary layer separation in supersonic flows are investigated numerically and reported. The separation length and the peak pressure are validated well with the experimental results.

Three different turbulence models are used; RNG-k-ϵ model predicted the results closer to the experimental than k-ω and Reynolds stress model. The recirculation region upstream and downstream of the jet in the transverse injection system significantly influences mixing and flame holding. The flow field characteristics of transverse fuel jet injection in supersonic cross-flow are reported for various parameters such as free stream Mach number, jet injection angle, and jet injection pressure. Multiple plots are drawn to give the quantitative study of variation in separation length against different parameters. Some of the essential qualitative observations are reported below

- With the increase in free stream Mach number, the jet-induced shock formed closer to the jet with more significant wall peak pressure, and as a consequence, separation length decreased.
- Injecting the fuel at an angle downstream resulted in decreased separation length and wall peak pressure as well due to reduced shock strength.
- Injecting fuel at an angle against upstream resulted in deformation of Mach disc, and the separation length is found large for normal injection. The upstream separation is observed more in the case of fuel injection against upstream than downstream.
- With the increase in injection pressure ratio, the jet-induced shock wave formed at a greater distance from the injector. As a result, upstream separation length increased, and higher peak wall pressure is recorded.

References

[1] V. Viti, R. Neel, J.A. Schetz, Detailed flow physics of the supersonic jet interaction flow field, Physics of Fluids 21 (2009) 1-16.

[2] C. F. Chenault, P. S. Beran, R. D. W. Bowersox, Numerical investigation of supersonic injection using a Reynolds-stress turbulence model, AIAA J. 37(10) (1999)1257-1269.

[3] S.Alexander and D. Jean-Paul, Turbulent shear layers in supersonic flow, New York, (2006).

[4] E. Erdem, K. Kontis, Numerical and experimental investigation of transverse injection flows, J. of Shock waves 20 (2010) 10-118.

[5] P. Kutschenreuter, E.T. Curran, S.N.B. Murthy (Eds.), Supersonic Flow Combustors, Scramjet Propulsion, AIAA, Reston, VA, (2000).

[6] W.H.Heiser, D.T. Pratt, D.H. Daley, and U.B. Mehta, Hypersonic Airbreathing Propulsion, AIAA Education Series, Washington, DC. (1994).

[7] J. A. Schetz, F. S. Billing, S. Favin, Analysis of slot injection in hypersonic flow, J. of Propulsion and power 7(1) (1991) 115-122.

[8] T.F. Fric, and A. Roshko, Vortical structure in the wake of a transverse jet, J. of Fluid Mechanics, 279 (1994) 1-47.

[9] F.S., Billig, Research on supersonic combustion, J. of Propulsion and Power 9 (4) (1993) 499-514

[10] MD. Habib, U. Khan, Z. U. Ahmed, Fluid flow and heat transfer characteristics of multiple Swirling Impinging Jets at various impingement distance, Int. J. of Thermofluid Sci. and Tech. 6(4) (2019) 19060403.

[11] D.W. Riggins, C.R. McClinton, R.C. Rogers, R.D. Bittner, Investigation of Scramjet injection strategies for high Mach number flows. J. of Propulsion and Power 11(3) (1995) 409-418.

[12] D. Papamoschou, D.G. Hubbard, Visual observations of supersonic transverse jets, Experiments in Fluids 14 (1993) 468-476.

[13] W. Huang, J. Liu, L. Jin, L. Yan, Molecular weight and injector configuration effects on the transverse injection flow field properties in supersonic flows, Aerospace Sci. and Tech. 32 (2014) 94–102.

[14] Ali Akbar, Sadegh Tabejamaat, Yashar Shoraka, Effects of incident shock wave on mixing and flame holding of hydrogen in supersonic airflow, Int. J. of Hydrogen Energy, 39 (2014) 10284-10292.

[15] M. Ali, A.K.M. Sadrul Islam, S. Ahmed, Mixing and flame holding with air inlet configuration in scramjet combustor, Int. communication in Heat and Mass Transfer 31 (2004) 1187–1198.

[16] A. Ben-Yakar, R.K. Hanson, Hypervelocity combustion studies using simultaneous OH-PLIF and schlieren imaging in an expansion tube. 36th AIAA Aerospace Sci.s Meeting

and Exhibit, Aerospace Sci.s Meetings, (1999) 99-2453.

[17] K.M. Kim, S.W. Baek, C.Y. Han, Numerical study on supersonic combustion with cavity-based fuel injection. Int. J. of Heat and Mass Transfer, 47,271-286, (2004).

[18] W. Huang, W Liu, S. Li, Z. Xia, J. Liu and Z. Wangn, Influences of the turbulence model and the slot width on the transverse slot injection flow field in supersonic flows, Acta Astronautica 73 (2012) 1-9.

[19] E. Khali, Y. Yao, Mixing flow characteristics for a transverse sonic jet injecting into a supersonic crossflow. 53rd AIAA Aerospace Sci.s Meeting: AIAA Sci-Tech Conference, Kissimmee, Florida, USA (2015) 5-9.

[20] M. Yousuf, A. Farrukh, Jet arrays in supersonic crossflow —An experimental study, Physics of Fluids 27 (2015) 126102.

[21] M. Nithiyaraj, R. Govardhan, Particle image velocimetry study of sonic jet injected into a supersonic crossflow, Fluid Mechanics and Fluid Power – Contemporary Research (2014) 177-184.

[22] D. Ruofan, C. Yan, H. Yu, Z. Han, Partially-averaged navier-stokes method for turbulence simulations: A jet injected into a supersonic crossflow. IEEE, 7th Int. Conference on Mechanical and Aerospace Engg. (ICMAE), London, UK, (2016).

[23] R.Soni, D. Ashoke, Investigation of strut-ramp injector in a Scramjet combustor: Effect of strut geometry, fuel and jet diameter on mixing characteristics. J. of Mechanical Sci. and Tech. 31 (3) (2017) 1169-1179.

[24] A. Anazadehsayed, G. Barzegar, Y Amini, R. Moradi, Mixing augmentation of transverse hydrogen jet by injection of micro air jets in supersonic crossflow, Acta Astronautica 137 (2017) 403-414.

[25] L. Maddalena, T.L. Campioli, J.A. Schetz, Experimental and computational investigation of light-gas injectors in Mach 4.0 crossflow, J. of Propulsion and Power 22 (5) (2006) 1027–1038.

[26] M. Gerdroodbary, M. Imani, D. Ganji, Investigation of film cooling on nose cone by a forward-facing array of micro-jets in hypersonic flow , Internation Communication of Heat and Mass Transfer 64 (2015) 42–49.

[27] G. Aswin, D. Chakraborty, Numerical simulation of transverse side jet interaction with supersonic free stream, Aerospace Sci. and Tech. 14 (2010 295–301.

[28] A.S. Pudsey, R.R. Boyce, Numerical investigation of transverse jets through multiport injector arrays in supersonic crossflow, J. of Propulsion and Power 26 (2010) 1225–1236.

[29] F. Genin, S. Menon, Dynamics of sonic jet interaction into supersonic crossflow, J. of Turbulence 11(4) (2010).

[30] D.A. Dickmann, Shock/boundary layer interaction effects on transverse jets in crossflow over a flat plate , J. Spacecr. Rockets 46 (2009) 1132–1141.

[31] L. Asproulias, RANS Modeling for Compressible Turbulent Flows Involving Shock Wave Boundary Layer Interactions (PhD thesis),University of Manchester (2014) 68-69.X