International Journal of Chemical Engineering and Applications, Vol. 1, No. 4, December 2010
ISSN: 2010-0221

CFD Analysis of Four Jet Flow at Mach 1.74 with Fluent Software

K.M.Pandey and Virendra Kumar

***Abstract*—The variations in mean velocity profiles of the *x* component along *x*-axis of the four jets at the designed Mach number is discussed in this research paper. It is found that the velocity profiles are fairly symmetrical about *y*=0. The velocities and its gradients decay along *x*-axis. Due to the effect of entrainment in the shear layer, the velocity between four jets increases with *x*-axis. It observed that the dynamic pressure inside the nozzle, just before the exit of the jet is maximum where the velocity of flow is maximum while in space between two nozzles is less and near the nozzle wall the static pressure is maximum. The static pressure inside the nozzle is minimum. It decreases with velocity increase, while static pressure is more in the space between the two jets in compression to flow domain. The jet is designed for streamline flow and hence the intensity of turbulence is less inside nozzle as compared to atmosphere. The turbulence intensity has value of 3.43x10^3 (%) at nozzle exit. This is because of the eddy creation and reversal of flow at the base region of circular duct. Further downstream, as flow gets agitated and the turbulence intensity increases. A maximum of 6.78x10^3(%) is attained and beyond which the turbulent intensity goes on decreasing steadily.**

***Index Terms*—four jet, Mach number, De Laval nozzle, turbulence intensity, flow reversal.**

I. Introduction

A jet is formed by flow issuing from a nozzle into ambient fluid, which is at a different velocity. If the ambient fluid is at rest the jet is referred to as a free jet; if the surrounding fluid is moving, the jet is called a co-flowing jet. A jet is one of the basic flow configurations which have many practical applications such as in jet engines, combustors, chemical lasers, ink-jet printer heads, among others. The velocity at the exit of the nozzle of a typical laboratory jet has a smooth profile and a low turbulence level, about 0.1% - 0.5% of the mean velocity. Due to the velocity difference between the jet and the ambient fluid, a thin *shear layer* is created. This shear layer is highly unstable and is subjected to flow instabilities that eventually lead to the formation of large-scale vortical structures. The interaction of these structures produces strong flow fluctuations, entrains ambient fluid into the jet flow and enhances the mixing. The shear layer and consequently, the jet spread along the direction perpendicular to the main jet flow. The central portion of the jet, a region with almost uniform mean velocity, is called the *potential core*. Because of the spreading of the shear layer, the potential core eventually disappears at a distance of about four to six diameters downstream from the nozzle. The entrainment process continues further beyond the end of the potential core region such that the velocity distribution of the jet eventually relaxes to an asymptotic bell-shaped velocity profile as illustrated in Fig. 2.1. Also shown in Fig. 2.1 is the half-width of the jet, $y_{1/2}$, defined as the distance between the axis and the location where the *local velocity* equals half of the local maximum or centerline velocity, U_0. The increase in the jet half-width with downstream distance provides a measure of the spreading rate of the jet. Due to the spreading, the jet centerline velocity decreases downstream beyond the potential core region.

II. Literature Review

K.M.Pandey[1] worked on the topic of "Wall Static Pressure Variation in Sudden Expansion in Flow through De Laval Nozzles at Mach 1.74 And 2.23: A Fuzzy Logic Approach" and his findings are- The analysis of wall static pressure variation with fuzzy logic approach to have smooth flow in the duct. There are three area ratios chosen for the enlarged duct, 2.89, 6.00 and 10.00. The primary pressure ratio is taken as 2.65 and cavity aspect ratio is taken as 1 and 2. The study is analyzed for length to diameter ratio of 1, 2, 4 and 6. The nozzles used are De Laval type and with a Mach number of 1.74 and 2.23. The analysis based on fuzzy logic theory indicates that the length to diameter ratio of 1 is sufficient for smooth flow development if only the basis of wall static pressure variations is considered. Although these results are not consistent with the earlier findings but this opens another method through which one can analyze this flow. This result can be attributed to the fact that the flow coming out from these nozzles are parallel one.

K.M.Pandey[2] worked on the topic of "Wall Static Pressure Variation in Sudden Expansion in Cylindrical Ducts with Supersonic Flow : A Fuzzy Logic Approach" and his findings are- the analysis of wall static pressure variation with fuzzy logic approach to have smooth flow in the duct. Here there are three area ratios chosen for the enlarged duct, 2.89, 6.00 and 10.00. The primary pressure ratio is taken as 2.65 and cavity aspect ratio is taken as 1 and 2. The study is analyzed for length to diameter ratio of 1, 2, 4 and 6. The nozzles used are De Laval type and with a Mach number of 1.74 and 2.23 and conical nozzles having Mach numbers of 1.58 and 2.06. The analysis based on fuzzy logic theory indicates that the length to diameter ratio of 1 is sufficient for smooth flow development if only the basis of wall static pressure variations is considered.

K.M. Pandey and A.P. Singh [3] worked on the topic of "CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software" and they found that the variation in the Mach no., pressure ratio. For 40 angles the Mach number at exit is very low compared to other nozzles and the turbulence intensity is very high for 160 at exit and it is nearly 3.045e+05. While when the angle is 80 the Mach number at nozzle exit is 2.91 and same as 120 angle but an angle 160 it gives the mach number at nozzle exit is 2.92 and it is lowest at an angle 40. The degree of angle for conical nozzle can be large as 12 to 18 degree maximum so for maximum thrust we can go with 120 or 160 conical nozzle. K. M. Pandey et.al [4] worked on the topic of "Studies on Pressure Loss in Sudden Expansion in Flow through Nozzles: A Fuzzy Logic Approach" and there findings are minimum pressure loss takes place when the length to diameter ratio is one and it is seen that the results given by fuzzy logic formulation are very logical and it can be used for qualitative analysis of fluid flow in flow through nozzles in sudden expansion. K. M. Pandey and E.Rathakrishnan [5] worked on the topic of "Influence of Cavities on Flow Development in Sudden Expansion" and there findings are - Flow from nozzles expanding suddenly into circular pipes with and without cavities was experimentally investigated for a Mach number range of 0.6 to 2.75. The research indicates that the introduction of secondary circulation by cavities reduces the oscillatory nature of the flow more in subsonic region than in supersonic region. K. M. Pandey and E.Rathakrishnan [6] worked on the topic of "Annular Cavities for Base Flow Control" and there findings are – Air flow from a Mach 1.74 convergent – divergent axi-symmetric nozzle expanded suddenly into circular duct of larger cross sectional area, provided with annular rectangular cavities, was studied experimentally, focusing attention o the base pressure, and the flow development in the enlarged duct. It was found that the base pressure is strongly influenced by the expansion level at the nozzle exit, the area ratio of the passage, the length to diameter ratio of the enlarged duct. For low area ratio, the annular cavities result in the base pressure. Also, the cavity aspect ratio influences the base pressure significantly for low area ratio.

K. M. Pandey et.al [7] worked on the topic of "Studies on Supersonic Flows in the De Laval Nozzle at Mach No. 1.5 and its flow Development into a Suddenly Expanded Duct" and there findings are – Solution of supersonic flow fields of flow development in 2D De Laval nozzle with a duct. The study is aimed with 1.5 Mach numbers for various L/D, into a duct. The nature of the flow is smooth when the flow gets attached and streamlined. The suddenly expanded cavity not only causes head losses but also is accompanied by flow oscillations due to phenomenon called vortex shedding. K. M. Pandey et.al [8] worked on the topic of "Study on Rocket Nozzles with Combustion Chamber Using Fluent Software at Mach 2.1" and there findings are – The pressure and Temperature parameter depend upon air-fuel ratio. Loss of pressure and temperature above two fuel inlet for same quantity of air fuel ratio. K. M. Pandey et.al [9] worked on the topic of "Studies on Supersonic Flows in the De Laval Nozzle at Mach No. 1.5 and its Base Pressure into a suddenly Expanded Duct" and there findings are – Supersonic flow fields of wall static pressure in 2D De Laval Nozzles with a duct. Expansion of supersonic flow in De Laval nozzle, with 1.5 Mach numbers for various L/D, into a duct. The flow field of an axi-symmetric flow on sudden expansion is accompanied with flow reversal, flow separation and vortex shedding near the nozzle exit region. K. M. Pandey et.al [10] worked on the topic of "Study on Supersonic Free Single Jet Flow: A Numerical Analysis with Fluent Software" and there findings are – to review the basic aspects of free jet flow and to contribute additional data concerning effects on the free jet flow map of efflux Reynolds number and orifice geometry.

K.M. Pandey and A.P. Singh [11] worked on the topic of " Design and Development of De Laval nozzle for Mach 3 & 4 using methods of Characteristics with Fluent Software" and there findings are –gas flows in a De Laval nozzle using 2D axi-symmetric models, which solves the governing equations by a control volume method. The throat diameter is same for both nozzles and designed using method of characteristics. Detailed flow characteristics like the centerline Mach number distribution and Mach contours of the steady flow through the converging – diverging nozzle are obtained.

Bouderah, Gasmi and Serguine [12], studied *on 'Zero Gravity of Free-Surface Jet Flow'* the flow due to a jet against infinite vertical plate on the free surface, where the effects of gravity and surface tension is not taken into account. We use initially the method of the free streamline theory based on the hodograph method and Schwarz-Christoffel transformation technique to obtain the exact solution. The problem of determining the free surface due to a jet against a vertical plate is considered. The classical problem of free streamline flow of an ideal fluid has been studied by many authors. The first work in this type of problem is characterized by the use of the Schwartz-Christoffel formula. The latter can treat the flows of border, which combines rectilinear paroies and a free surface. A two dimensional flow of a jet ideal fluid encroaching on a wall neglecting the forces of gravity studied by Peng and Parker using the integral equations method. YIN Zhao-qin[13], studied on, '*Experimental Study on The Flow Field Characteristics in the Mixing Region of Twin Jets'*, twin jets flow, generated by two identical parallel axi-symmetric nozzles, has been experimentally investigated. The dimensionless spacing *(B)* between two nozzles were set at 1.89, 1.75 and 1.5. Measurements have been carried out at several free-stream velocities ranging from 10 m/s to 25 m/s or Reynolds numbers (based on the nozzle diameter of 44 mm) ranging from 3.33×104 to 8.33×104. The results show that the twin jets attract each other. With the increasing Reynolds number, the turbulence energy grows, which indicates that the twin jets attract acutely. The jet flow field and the merging process of two jets vary with *B*. The width of the twin jets flow spreads linearly downstream and grows with *B*. The merging between two jets occurs at the location closer to the nozzle exit for the cases with smaller spacing between nozzles and higher Reynolds number.

Berg, Ormiston, and Soliman[14], studied on '*Prediction of the flow structure in a turbulent rectangular free jet'*, a numerical analysis is conducted for turbulent flow of a rectangular free jet with an aspect ratio of 2:1. The computations were performed using two standard two-equation turbulence models (the k–ε and the k–ω models). Two inflow boundary conditions were evaluated with each model: a uniform inlet velocity profile and a profiled inlet velocity fitted to experimental data. The results show that the k–ε model with the profiled inlet velocity succeeded in predicting the main features of the flow, including the vena contract and the saddle-shaped velocity profiles in the near-field region, and the rate of velocity decay in the far-field region. Turbulent rectangular jets can be found in a great deal of engineering applications.

Abburi and Dixit[15] together generated a knowledge based system for predicting the surface roughness in turning process. In which they generated IF-THEN rules using Fuzzy set theory. Soft computing-based techniques are used to impart capabilities. Rathakrishnan[16] *A Numerical Approach to Single and Twin Supersonic Jet Flows'*, for steady in viscid flow the physical character of the flow is elliptic in the subsonic region and hyperbolic in the supersonic region. For steady supersonic flows that contain no embedded subsonic regions it is possible to construct a space marching scheme in the hyperbolic direction, which is usually the approximate flow direction. Then the marching direction has the same role as time in an unsteady problem. Such marching techniques prove to very efficient. For supersonic flows with shock, shock fitting or shock capturing techniques can be employed depending on the solution requirement. By casting the equations in conservation form and using discretized form that conserves the mass, momentum, etc, it is possible to obtain solutions which satisfy the weak form of the governing equations. In the weak form the governing equations automatically satisfy the Rankine-Hugoniot jump conditions across any discontinuity like shock. Shock capturing schemes are the most widely used techniques for studying in viscid flows with shocks. In this method, the Euler equations are cast in conservation form. An effective finite difference scheme for in viscid supersonic flows which serves as the basis for most of the steady flow shock capturing techniques in the explicit predictor corrector scheme. Zhang[17] 'An inclined rectangular jet in a turbulent boundary layer-vortex flow', a model test study was performed on stream wise vortices generated by a rectangular jet in an otherwise at plate turbulent boundary layer. The study was conducted in a low speed wind tunnel. The rectangular jet had a cross-section size of 28 mm by 5.5 mm. The coming boundary layer had a 99.5 percent thickness of 25 mm. The free stream speed of the oncoming flow was 20 m/s. Measurements were performed with a three-element LDA system. The effects of skew angle and stream wise development of vortex were investigated and the mean flow properties are presented. The study showed that the rectangular jet was able to produce a stream wise vortex of higher strength than that of a round jet, while at the same time keeping the same size and shape as that of a round jet. A 63% increase in the maximum vorticity was found. The 45°skew angle was identified as the optimal skew angle for vortex production. An experimental study was performed on stream wise vortices generated by a rectangular jet in an otherwise flat-plate turbulent boundary layer. The circulation level of the vortex was typical of flow control vortices but was one order of magnitude lower than those found in vortex/ aircraft interactions. Measurements were performed with a three-element LDA system in a low speed wind tunnel. The effects of skew angle and stream wise development of vortex were investigated. A comparison was also made with a round jet. The study helped to improve the current understanding of the inclined jets-in-cross flow and contributed to the establishment of a reliable database for numerical model validation. Wilcox[18] 'Passive venture system for modifying cavity flow fields at supersonic speed', Experimentally he showed that a passive venting system could be employed to control cavity flow field at supersonic speed, specifically the passive venting system had been used to extend the L/H value before the onset value of high drag producing closed cavity flow. In his experiment the porous flow eliminated the large drag increase for L/H >12. There is tremendous increase in drag coefficient for L/H > 12 but for porous flow having more diameters the decrease in drag coefficient is comparatively very less with the floor having fewer diameters. Rathakrishnan [19] demonstrated that a control in the form of annular ribs can serve as an effective controller to control the base pressure for axi-symmetric sudden expansion of under expanded sonic flows. It was found that the rib, when placed at an appropriate location downstream of the base prevents the reverse flow through boundary layer into the base zone. This enables the free-shear layer expanding at the base to have its process delinked from influence of the reverse flow.

III. Methodology

As all dimensions are known, the geometry is designed. Inserting the respective data the geometry is constructed, similarly the rest of them are constructed for a length to diameter ratio and never to forget, the solver is to be chosen. On saving the finished geometry, files with extension *jou, dbs* and *trn* are formed. The Geometry is then to be meshed for studying in smaller control volumes as briefly described. Then on exporting mesh, they form msh. file which is later used for simulation in Fluent software. The boundary conditions are set in the same software kit for the geometry, like wall, pressure inlet, pressure outlet etc. The procedure of constructing the complete geometry and mesh would be stored in journal file which when imported to gambit shall open up the geometry constructed earlier.

A. Geometry and Computational Grid Structure

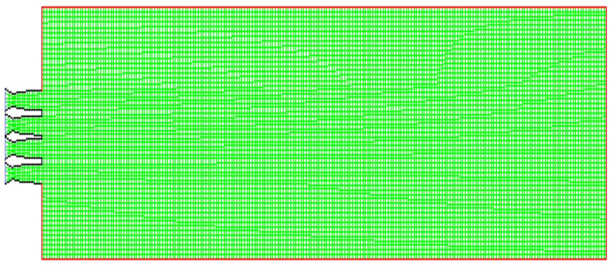

Fig.1. Grid arrangement of free four jet flow.

The project involved setting up the geometry of the situation using the program GAMBIT. The governing Fluid Dynamics equations (continuity, momentum and energy) were then solved over the grid developed in GAMBIT using the program FLUENT. For both programs there was a moderate learning curve . The geometry of the situation being examined is shown below. The initial goal of the project was to replicate this set up and solve for the flow conditions. In order to arrive at the precise solution, we need to describe the inputs that are to be fed to the simulation. Our descriptive study is for the conical nozzle of Mach numbers 1.74, for which it is important to know the dimensions of the respective set ups which invariably includes the convergent divergent nozzle and a circular duct with and without passive control. The remaining factors such as turbulent parameters and pressure ratio are to be fed for initiating the flow within the created geometry. With these parameters established we can proceed towards building a geometry using the Gambit Software.

IV. Geometry and Grid Arrangement

A 2d axi-symmetric computational domain was considered, the initial design parameters for jet nozzle of Mach number 1.74. The nozzle choosen is taken as De Laval.

V. Results And Discussions

A. *Supersonic Free Four Jet Flow*

A four jet supersonic flow with application to high performance VTOL aircraft is investigated. Flow visualization studies were conducted with air as the working fluid. Results at different Mach numbers of the flow are presented to describe the properties. Four jet flow, generated by four identical parallel axi-symmetric nozzles, has been investigated with the help of FLUENT software. The dimensionless spacing (*B*) between two nozzles were set at 8 to 16. The results show that the four jets attract each other. The jet flow field and the merging process of four jets vary with *B*. The width of the four jets flow spreads linearly downstream and grows with *B*. The merging between four jets occurs at the location closer to the nozzle exits for the cases with smaller spacing between nozzles and higher Reynolds number.

B. *Velocity*

The variations in mean velocity profiles of the *x* component along *x*-axis of the four jets at *B*=9 and M_e 1.74 are shown in Fig.2,3 and 4, where maximum velocity $5.98x10^2$ at the exit of the jets. It is found that the velocity profiles are fairly symmetrical about *y*=0. The velocities and its gradients decay along *x*-axis. Due to the effect of entrainment in the shear layer, the velocity between four jets increases with *x*-axis. At the same time, the width of the jets was found to increase.

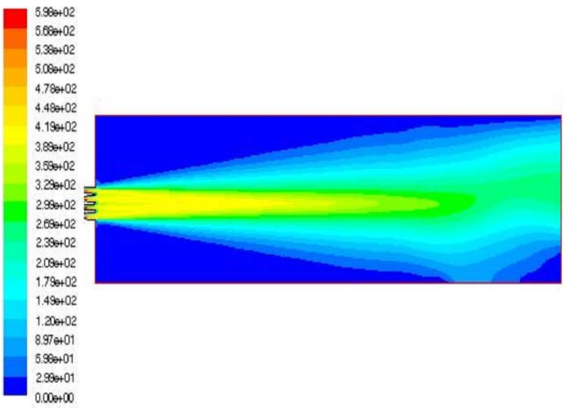

Fig. 2. Contour of Velocity Magnitude (m/s)

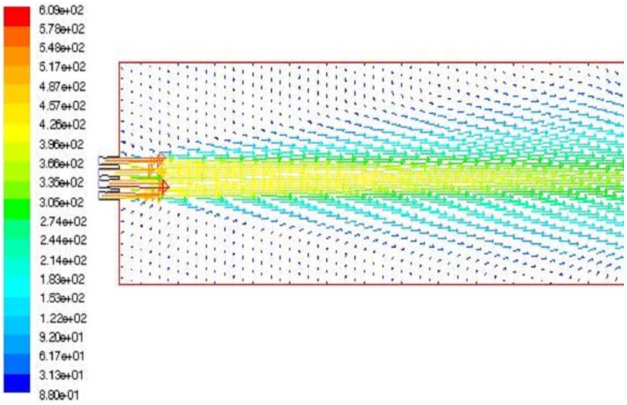

Fig. 3. Contour of the Velocity Vector (m/s)

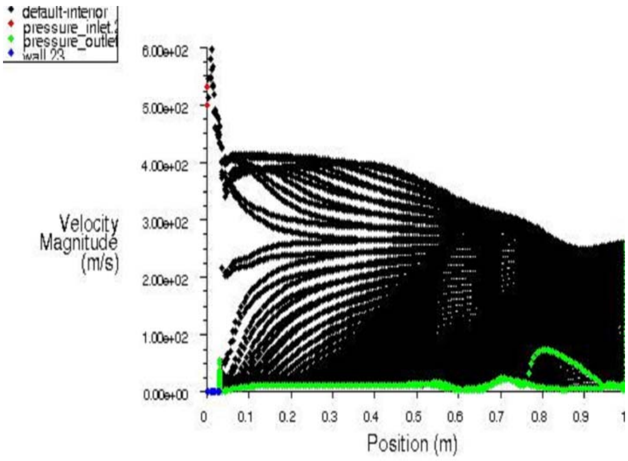

Fig.4. X-Y plot of the Velocity Magnitude (m/s)

C. *Pressure*

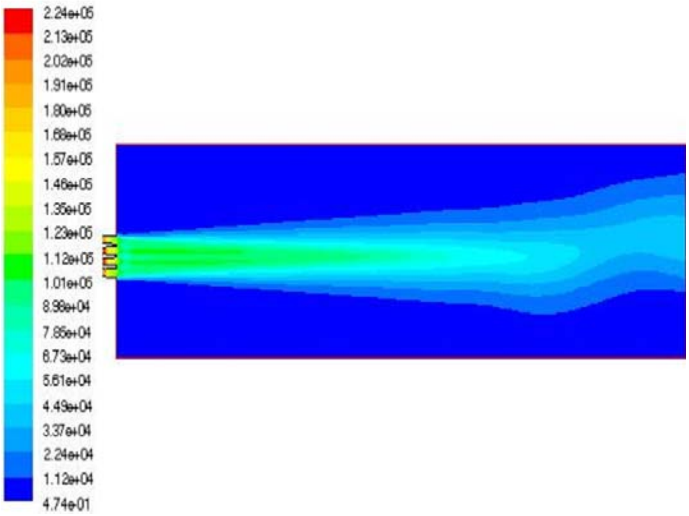

Fig. 5. Contour of Dynamic Pressure (Pascal)

From Fig. 5, it observed that the dynamic pressure inside the nozzle, just before the exit of the jet is maximum where the velocity of flow is maximum while in space between two nozzles less and near the nozzle wall the static pressure is maximum. In Fig. 7, the static pressure inside the nozzle is minimum it decreases with velocity increases while static pressure is more in the space between the two jets in compression to flow domain. The maximum static pressure is $1.17x10^5$ Pa. in the Fig. 7, the total pressure in the order of $1.01x10^5$ to $2.39x10^5$ Pa.

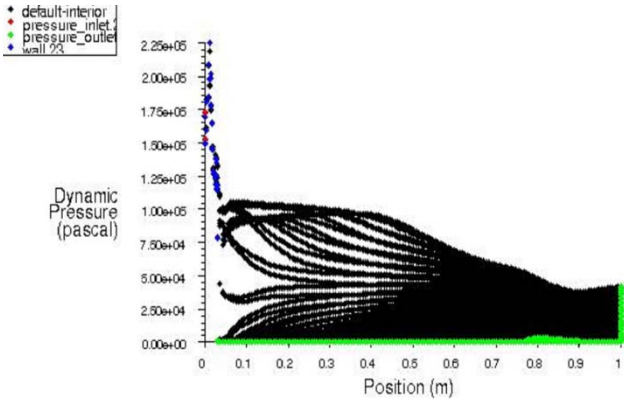

Fig. 6. X-Y Plot of Dynamic Pressure (Pascal)

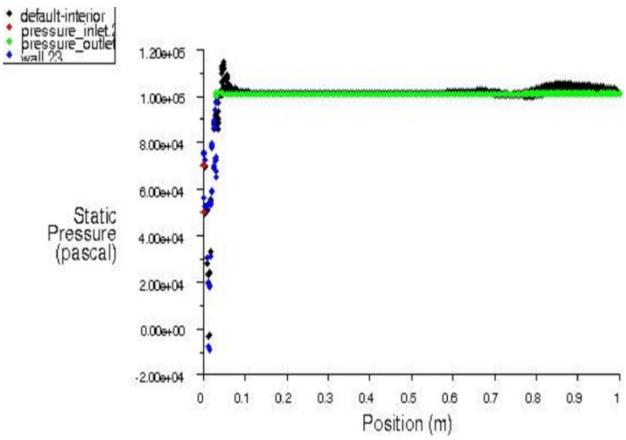

Fig.7.1f, X-Y Plot of the Static Pressure (Pascal)

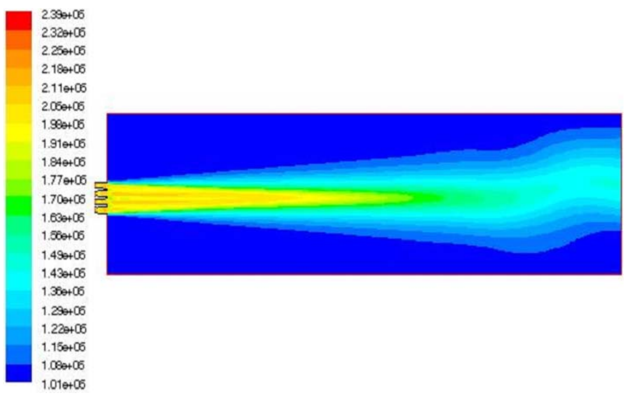

Fig. 8. Contour of the Total Pressure (Pascal)

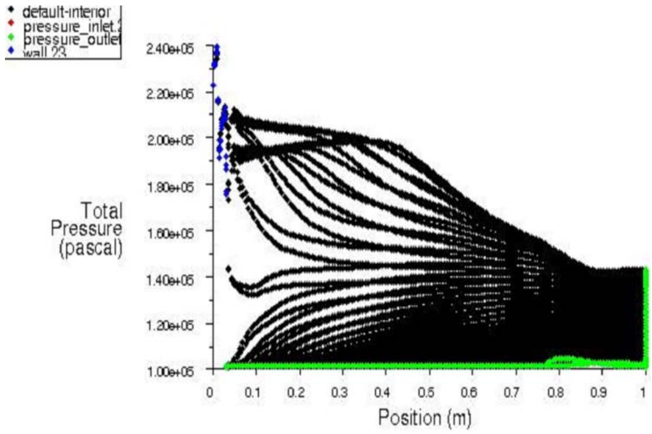

Fig.9. X-Y Plot of the Total Pressure (Pascal)Turbulence

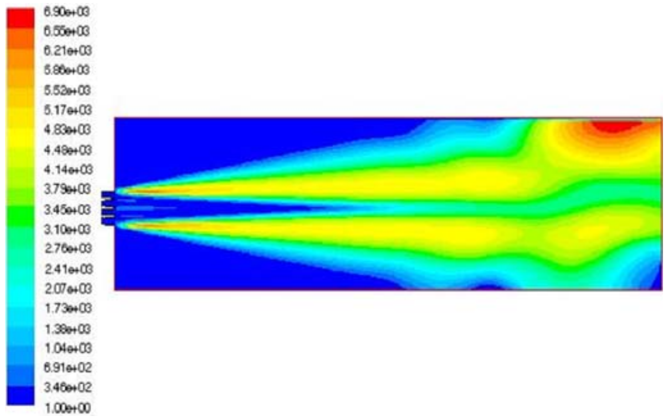

Fig. 10. Contour of the Turbulence Kinetic Energy (m^2/s^2)

The jet is designed for streamline flow and hence the intensity of turbulence is less inside nozzle as compared to atmosphere. The turbulence intensity has value of $3.43x10^3$ (%) at nozzle exit. This is because of the eddy creation and reversal of flow at the base region of circular duct. Further downstream, as flow gets agitated and the turbulence intensity increases. A maximum of $6.78x10^3$(%) is attained and beyond which the turbulent intensity steadily decreases. While the turbulence of kinetic energy range 1 to $6.90x10^3$ m^2/s^2.

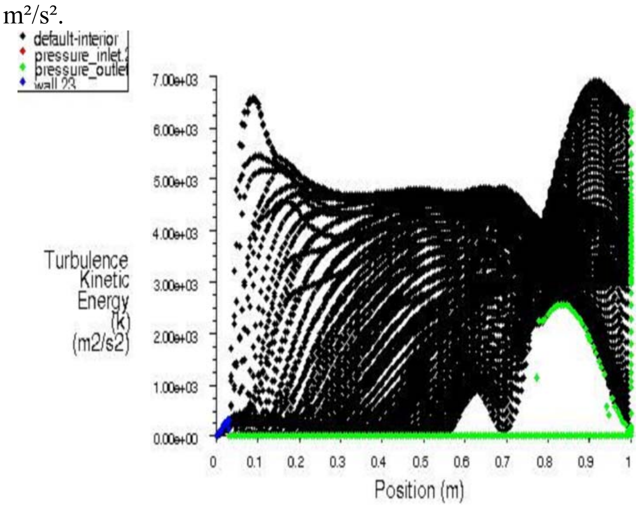

Fig. 11. X-Y Plot of the Turbulence Kinetic Energy (m^2/s^2)

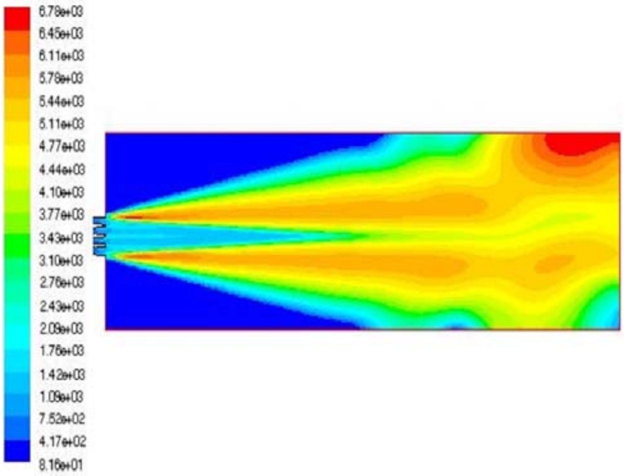

Fig. 12. Contour of the Turbulence Intensity (%)

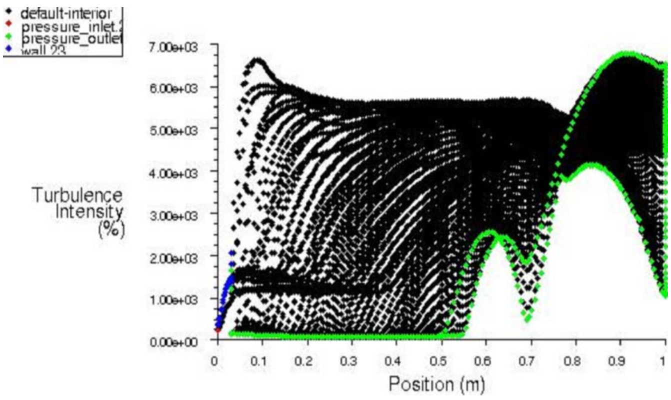

Fig. 13. X-Y Plot of the Turbulence Intensity (%)

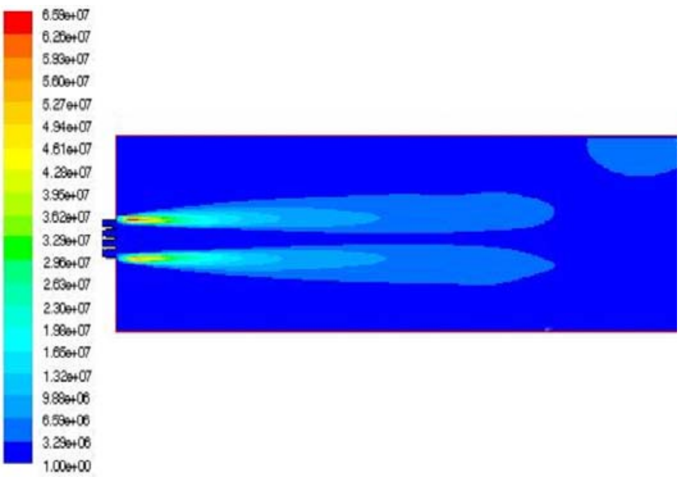

Fig. 14. Contour of the Turbulence Dissipation Rate (□) (m^2/s^3)

From the Fig.10, 11,12,13,14 and 15, it is observed that turbulence dissipation rate has maximum value 6.59x10^7 m^2/s^3 at the point of maximum turbulence Intensity.

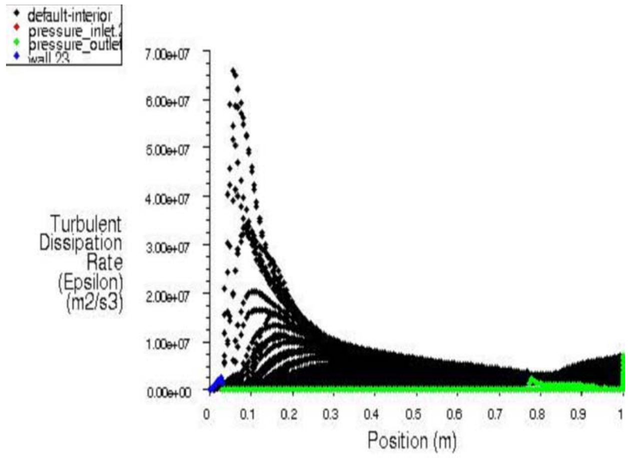

Fig. 15. X-Y Plot of The Turbulence Dissipation Rate (□) (m^2/s^3)

VI. Conclusion

FLUENT analysis was carried out to investigate the flow field of the four jets at Mach number of 1.74. The flow behaviors were observed to assist in understanding the change in the flow. The effects of Mach numbers (M_e), pressure ratio (P_e/P_a) and the spacing (B) between two jets on the flow field along the flow and lateral directions were examined. The conclusions are drawn as follows:

The jet boundary, the shape of the shock cell, the Mach disc and the intercepting shock are clearly seen from results. It is seen that increase in jet Mach number M_e results in increase of the shock cell length, ie, the Mach disc shifts downstream with increase of M_e. Further, the Mach disc diameter also increases with increasing Mach number, as expected. It is seen that both M_e and P_e/P_a have similar effect on the jet width. In general, the jet width increases with increase of both M_e and P_e/P_a. The distance where the inter boundaries coalesce, the jets behave as single jet. Beyond this point, the Mach number distribution in the jets is totally different from that of single jet. The reason for the difference is that the single jet is axi-symmetric throughout, whereas the jets in four jets are influenced by the wave pattern which tries to establish a slip stream right from the merging point.

In the flow direction, the four jets are clearly separated near the nozzle exits. Away from this region, the four jets mix and merge to appear as a single jet. The velocity and turbulent energy profiles are fairly symmetrical around the centre line of the four jets for various Mach number and pressure ratio and the spacing between the four nozzles. The velocities between the four jets change along the lateral direction. The interference between four jets increases with increase of L. The stronger the interference is, the larger the Reynolds shear stress and turbulent energy are. The interference between the four jets increases as the spacing between four nozzles decrease. Furthermore the width of the four jets spreads linearly downstream and grows with B. The merging length of the four jets can be increased either by reducing B or increasing Mach number.

REFERENCES

[1] K.M.Pandey., Wall Static Pressure Variation in Sudden Expansion in Flow through De Laval Nozzle at Mach 1.74 and 2.23: A Fuzzy Logic Approach, 2010 Second International Conference on Machine Learning and Computing, IEEE, DOI 10.1109/ICMLC.2010.75, pp. 243- 247.

[2] K.M.Pandey., Wall Static Pressure Variation in Sudden Expansion in Cylindrical Ducts with Supersonic Flow: A Fuzzy Logic Approach, 2010 Second International Conference on Machine Learning and Computing, IEEE, DOI 10.1109/ICMLC.2010.74, PP. 237-242.

[3] K.M. Pandey and A.P. Singh., CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software, International Journal of Chemical Engineering and Applications, Vol. 1, No. 2, August 2010, pp.179-185.

[4] K. M. Pandey, R. Jagannath and N. G. Naresh., Studies on pressure loss in sudden expansion in flow through nozzles: a fuzzy logic approach, ARPN Journal of Engineering and Applied Sciences, vol. 2, no. 2, April 2007, pp.50-91.

[5] K. M. Pandey and E.Rathakrishnan., Influence of Cavities on Flow Development in Sudden Expansion, International Journal of Turbo And Jet Engines, vol. 23, no.2, 2006, pp. 97-112.

[6] K. M. Pandey and E.Rathakrishnan., Annular Cavities for Base Flow Control, International Journal of Turbo And Jet Engines, vol. 23, no.2, 2006, pp.113-127.

[7] K. M. Pandey, Prateek Shrivastava, K.C.sharma and A.P.Singh., Studies on Supersonic Flows in the De Laval Nozzle at Mach No. 1.5 and its flow Development into a Suddenly Expanded Duct, The 10th Asian Symposium on Visualization, SRM University, Chennai, March1-5,2010., pp. 164-169.

[8] K. M. Pandey, Surendra Yadav, and A.P.Singh., Study on Rocket Nozzles with Combustion Chamber Using Fluent Software at Mach 2.1, The 10th Asian Symposium on Visualization, SRM University, Chennai, March1-5, 2010, pp. 171-177.

[9] K. M. Pandey, Prateek Shrivastava, K.C.sharma and A.P.Singh., Studies on Supersonic Flows in the De Laval Nozzle at Mach No. 1.5 and its Base Pressure into a suddenly Expanded Duct, The 10th Asian Symposium on Visualization, SRM University, Chennai, March1-5, 2010, pp.624-631.

[10] K. M. Pandey, Virendra Kumar, and A.P.Singh., Study on Supersonic Free Single Jet Flow: A Numerical Analysis with Fluent Software, The 10th Asian Symposium on Visualization, SRM University, Chennai, March1-5, 2010, pp.179-184.

[11] K.M. Pandey and A.P. Singh., Design and Development of De Laval nozzle for Mach 3 & 4 using methods of Characteristics with Fluent Software, ISST Journal of Mechanical Engineering, Vol.1, Issue 1, 2010, pp. 61-72.

[12] B. Bouderah, A. Gasmi and H. Serguine, Zero Gravity of Free-Surface Jet Flow, *International Mathematical Forum*, 3273 – 3277, **(2007)**

[13] Yin Zhao-qin, Experimental Study on The Flow Field Characteristics in the Mixing Region of Twin Jets, *journal of Hydrodynamics*, 309-313**, (2007)**

[14] J.R. Berg, S.J. Ormiston, H.M. Soliman, Prediction of the flow structure in a turbulent rectangular free jet, *International Communications in Heat and Mass Transfer*,**(33),** 552–563, **(2006)**

[15] Abburi, N.R., Dixit, U.S., A knowledge-based system for the prediction of surface roughness in turning process, *Robotics and Computer-Integrated manufacturing,* **(22),** 363- 372, **(2006).**

[16] E. Rathakrishnan, A Numerical Approach to Single and Twin Supersonic Jet Flows, *IE (I) Journal AS, January,* 32-34, **(2004).**

[17] X. Zhang, 'An inclined rectangular jet in a turbulent boundary layer-vortex flow, *Experiments in Fluids***, (28)**, 344-354, (**2000).**

[18] Wilcox, J.F. Jr., 'Passive venting system for modifying cavity flow fields at supersonic speeds', *AIAA Journal*, Vol. 26, No. 3, 1988, p. 374-376.

[19] Rathakrishnan, E., 'Effect of ribs on suddenly expanded flows', *AIAA Journal*, Vol. 39, No. 7, 1998, p.1402-1404.

AUTHORS' INFORMATION

Dr. K.M.Pandey did his PhD in Mechanical Engineering in 1994 from IIT Kanpur. He has published and presented 170 papers in International & National Conferences and Journals. Currently he is working as Professor in Mechanical .Engineering Department, National Institute of Technology, Silchar, Assam, India.. He also served the Department in the capacity of head from July 07 to 13 July 2010. He has also worked as faculty consultant in Colombo Plan Staff College, Manila, Philippines as seconded faculty from Government of India.

His research interest areas are the following; Combustion, High speed flows, Technical Education, Fuzzy logic and Neural Network , Heat Transfer, Internal Combustion Engines, Human Resource Management, Gas Dynamics and Numerical Simulations in CFD with commercial soft wares. Email:kmpandey2001@yahoo.com

Virendra Kumar-Virendra Kumar did his M.Tech. in Thermal Engineering from Department of Mechanical Eengineering of NIT Silchar, Assam. He served in Department of Mechanical Engineering at Gurgaon College of Engineering, Gurgaon. Currently he is doing Ph.D. at Mechanical Engineering Department of Indian Institute of Technology, Delhi.