Journal of Basic and Applied Engineering Research
Print ISSN: 2350-0077; Online ISSN: 2350-0255; Volume 1, Number 2; October, 2014 pp. 28-32

http://www.krishisanskriti.org/jbaer.html

Numerical Simulation of Subsonic jet through Circular, Square, and Rectangular Nozzles

Piyush Vijayan[1], Sandhya M.[2], Arthur John[3]

[1]*P.G. Student, Government Engineering College, Thrissur, India*
[2]*Government Engineering College, Thrissur, India*
[3]*PG Student, Government Engineering College, Thrissur, India*

Abstract: **Numerical Simulation of a circular, square and rectangular jet is being carried out at subsonic Mach number and various flow properties and turbulence characteristics are being obtained. Simulations are performed in a three dimensional computational domain using steady RANS equations and SST k-ω turbulence model. The computational domain is discretized using hexahedral mesh .The velocity decay of all the three nozzles are being analyzed with the main aim of attaining better mixing characteristic. Results show that the rectangular jet due to its better entrainment and small scale mixing property is more efficient when it comes to mixing enhancement characteristic due to its non axi-symmetric geometry. Furthermore, the importance of U_{rms} and V_{rms} in mixing enhancement is also analyzed. Shortening of potential core and velocity decay in rectangular jet ultimately also result in reduction of sound from the jet exit.**

1. INTRODUCTION

Fluid flow through nozzles has been researched extensively due to its large application in fighter jets and also in the chemical industry where different chemicals need to be thoroughly mixed. The reason for nozzle being such an intriguing area of interest for researchers is due to its varied application and its widespread availability. Over the past decade or so, flow regimes of various converging, diverging and converging-diverging nozzles have been well established. The research in this field was further helped by commencement of jet application in aircrafts and due to its easy implementation with its propulsive units, gas turbines.

In the beginning, the area of research was limited to circular jet due to its uniform flow pattern, ease of manufacturing and ease of incorporation with gas turbines. But with increase in demand for better maneuverability and stealth capabilities in a jet, the need to look beyond circular jets increased. This accompanied by various other potential areas lead to investigation into the feasibility and application of non-circular nozzles. Besides, towards the end of 20^{th} century, focus also started shifting towards noise reduction and its impact on the aircraft. All these factors lead to the feasibility study of non-axi symmetric nozzles, especially rectangular, triangular and elliptic exits.

In this paper, the main focus of study would be the mixing characteristics associated with three nozzles having square, rectangular and circular exits. Mixing between a jet and its surroundings take place in two stages- macro level and micro level. Initial stage is the macro level stage where large quantity of fluid is brought together followed by micro level stage where mixing is being induced by the development of small scale rotational velocity at molecular level.

The entrainment rate, i.e. the rate at which the mixing of the jet takes place with the surrounding is controlled by the large scale coherent vortical structures. Mixing in a nozzle can be enhanced in two ways- 1) manipulating the coherent vertical structures by means of actuators, vortex generators, tabs, etc. or 2) modifying the geometry of the jet nozzle which directly alter the flow development downstream relative to using a circular nozzle. Research on flow control with non- circular jets was carried by Gutmark & Grinstein (1999) [3]. The reason why rectangular jet configurations are of particular interest is because it offers better mixing at both ends of the spectrum: enhanced large scale mixing due to axis-switching, and enhanced small mixing near corner regions and due to faster breakdown of vortex ring coherence and hence faster transition to turbulence. This is also generally accompanied by a higher spreading rate of the jet. Also, the noise generated by the rectangular nozzle is usually lower than that of the circular nozzle of the same exit cross sectional area, although this may not always be the case as proved by Knowles and Saddington (2006) [4].

The study of rectangular nozzles assume more importance as it offers better mixing capability and thrust vectoring, both of which compliments the two requirement of current breed of fighter aircrafts, stealth and agility. A rectangular nozzle in its simplest configuration may provide better one dimensional thrust capabilities. Also, the nozzle profile can be easily altered to ensure that the jet is always ideally expanded at all flight conditions to minimize thrust losses. The only downside of using rectangular nozzles is thrust loss when compared to a circular nozzle. The loss is expected to be around 5% for a 6:1 aspect ratio (AR) [4], but the loss is sometimes acceptable due to the other advantages offered by rectangular nozzle.

The need to develop and integrate the use of non-axisymmetric nozzles into aircraft requires the presence of extensive database for the properties of free jets emerging from such nozzles. Rectangular nozzles have already been incorporated in aircrafts such as F-22 and B-2 as thrust nozzles. They are also used as roll posts in F-35 as these experience conditions of high Nozzle Pressure Ratio (NPR) during vertical take-off and landing.

The characteristics of the flow field issuing from the rectangular nozzles depend upon various factors such as inlet geometry of the nozzle, the Reynold's number, exit aspect ratio (AR_e), condition of the ambient medium and the type of exit velocity profile(Krothrapalli et al. [5]). Our main focus by the medium of this paper is to investigate the effect of exit velocity profile on the flow properties.

Investigation was carried by Hammond [6] where the effect of changing the nozzle exit on their decay rates was being studied (Figure 1.1). It can be seen that the rectangular nozzle shows higher decay when compared to the circular nozzle and the slot nozzle being further modification in the form of chevrons or slots to the rectangular nozzle shows the highest decay among the three.

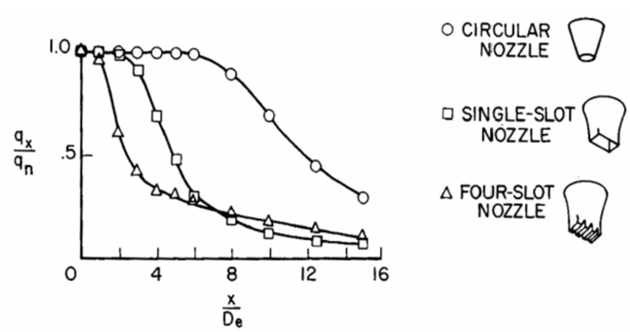

Fig. 1. Effect of nozzle geometry on jet velocity decay [6]

Comparison was also carried out in the performance of a Stratford nozzle and a conical nozzle. The Stratford nozzle's main characteristics is its smoothly profiled converging section which ensure minimum flow separation and hence the exit turbulent intensity. It was found that although there was no dependence on the potential core length but being affected by the NPR and temperature. Due to the different exit turbulent intensity, considerable difference was being observed in the spreading rate of the nozzle downstream.

Zhou and Sang Joon Lee [9] performed experiments and obtained fluid flow and heat transfer characteristics of a rectangular jet impinging on a heated flat plate. From the experiments, it was found that the jet Reynolds Number, the nozzle to plate spacing as well as the turbulent intensity have an important influence on the heat transfer of impinging rectangular jets. They also derived a correlation for local Nusselt number the free stream turbulence intensity. Sfeir [10] by using hot-wire anemometry determined the mean velocity and temperature profiles of rectangular jets with different aspect ratios and nozzle geometries.

In general, non-circular jet has been found to spread and mix faster than circular jets. Ho and Gutmark [11] investigated the flow characteristics of a 2:1 aspect ratio elliptic jet and found that the mass entrainment before the end of the potential core to be three to eight times than that in an axi-symmetric jet. Husain and Husain [12] carried out a study comparing aspect ratio in the range of 2:1 – 8:1. It was observed that for a given equivalent diameter, the aspect ratio is an important characteristic controlling the deformation and typological changes, i.e. bifurcation of large-scale vertical structures in elliptic jets.

Research in the field of non-axisymmetric nozzles show that the flow can be divided into three regions as shown in Figure 1.2 below:

1) The potential core region, where the centreline velocity, U_c is constant;

2) The two-dimensional transition region, where $(U_c)^2 \propto x^{-n}$ with $n \approx 1$

3) The region extending to infinity, where the centreline velocity decay is characteristic of axisymmetric jets, i.e. $U_c \propto x^{-1}$

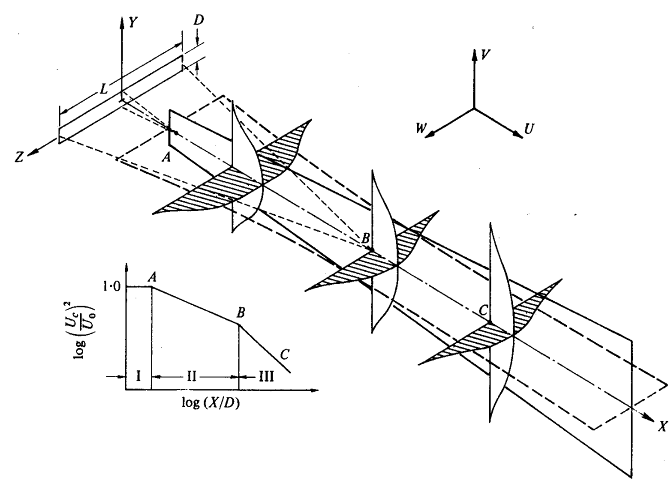

Fig. 2. Schematic representation of a subsonic free jet flow field [5]

2. PROBLEM DESCRIPTION AND SOLUTION METHODOLOGY

2.1 Nozzle geometry

The nozzle considered in the present work is the same as the one considered by Anderson et al. [1]. The exit diameter D_j of the circular nozzle is 50 mm. For the nozzles with the rectangular and square exit cross section, dimension was calculated keeping the exit area same as that in the case of circular nozzle. For the rectangular (2:1 Aspect ratio) and square nozzle (1:1 Aspect ratio), the exit dimensions were 31.3328mm X 62.6657mm and 44.310mm X 44.310mm respectively.

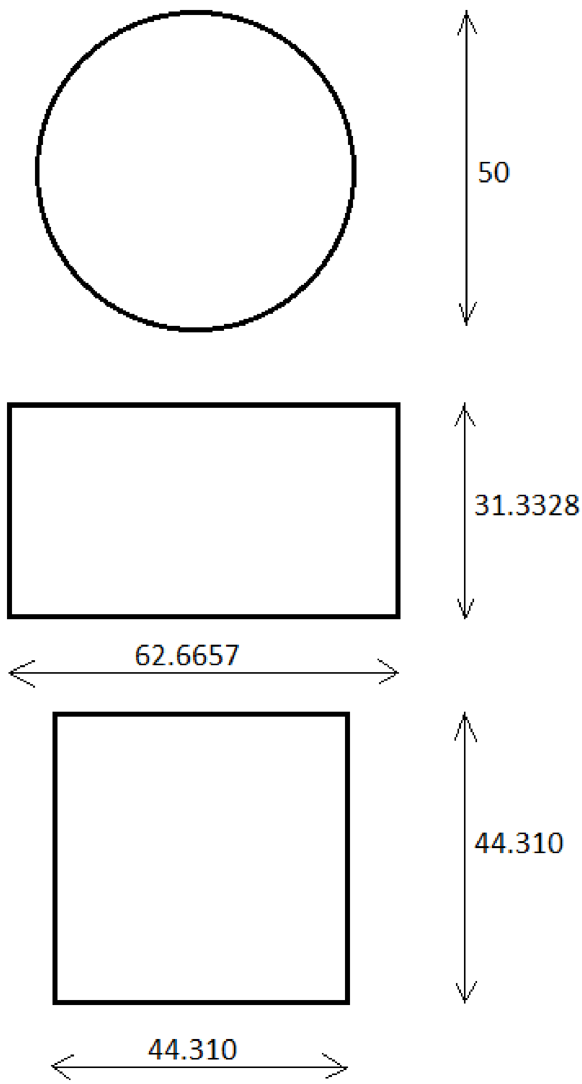

Fig. 3. Geometry dimensions of circular, rectangular and square nozzle exit (All dimensions in mm)

2.2 Computational domain

The computational domain used for the three-dimensional simulation is a rectangle with length breadth and height of 3m, 3m and 1.5 m respectively. Domain independent study was carried out to ensure least computational time. Gambit was used to model the geometries and for meshing purpose. Grid independent study was carried out and optimum number of nodes was fixed at almost 1.5 million with minimum orthogonal quality being 0.998705 and maximum aspect ratio in the range of 44 to 55 depending on the geometry.

2.3 Boundary conditions and Numerical Scheme

The flow is three-dimensional, compressible and turbulent, Reynold's averaged Navier-Stokes equation along with SST K-ω turbulence equations have been solved. Bousinessq hypothesis is used which links Reynold's stresses with the mean rate of deformation. The nozzle inlet has been characterized as pressure inlet with static and stagnation pressure specified. The rest of the domain inlet and the upper outer entrainment surface have been characterized as pressure inlets at the ambient pressure and temperature. The domain outlet at the downstream end of the domain is taken to be a pressure outlet at the ambient pressure and temperature. The nozzle wall has been treated as adiabatic surfaces and standard wall functions have been used. The bottom boundary is treated as a symmetry surface for the sake of minimizing computational time.

Fluent 13 was used for simulation purpose. Due to its suitability in subsonic flows, pressure based solver have being used. The flow was assumed to steady. The convergence criteria of energy was 1e-06 where for all the other parameters, it was kept as 1e-05. The boundary conditions applied is given in Table 1.

Table 1

Flow Properties	Jet
Mach Number	0.75
T_j/T_∞	1
P_∞	101325 Pa
C_∞	340.174 m/sec
T_∞	288 K
T_{0j}	320.4 K
Re_D	5×10^5
NPR	1.456

3. RESULTS AND DISCUSSION

3.1 Comparison of mean axial velocity

The variation of axial velocity along the centerline of computational data of square and rectangle nozzle data is being compared with that of experimental data of a circular nozzle from Ref. [1] in Fig. 4. The results show that there is over prediction in case of potential core length in numerical data although the velocity decay of the three shows that the velocity decay is more in case of rectangle nozzle.

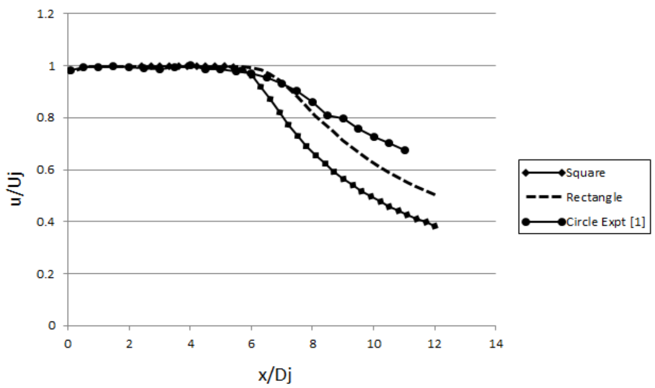

Fig. 4. Variation of axial velocity along the centerline

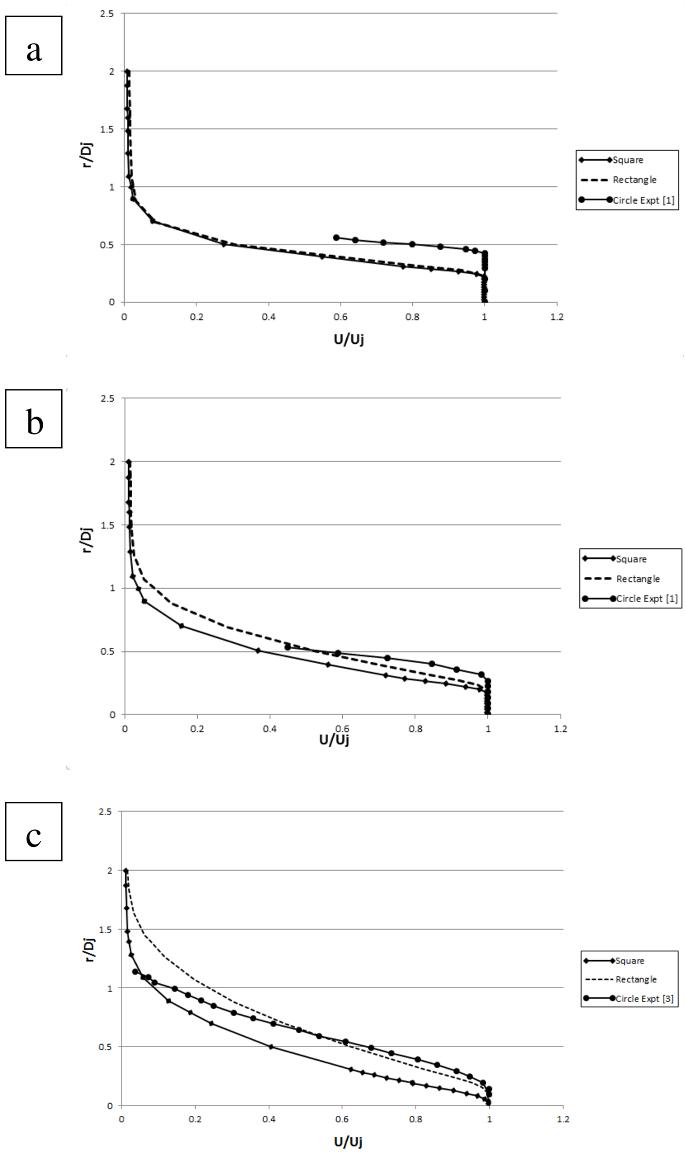

Fig. 5. Radial profile of velocity at axial stations (a) Z=1 (b) Z=2. 5 and (c) Z=5

Variation of the axial velocity in the radial direction at three axial stations, namely, x/D_j =1, 2.5 and 5 for all the three geometries is shown in Fig. 5. It shows how as we go further downstream, the slope of axial velocity becomes steeper.

3.2 Comparison of fluctuating quantities

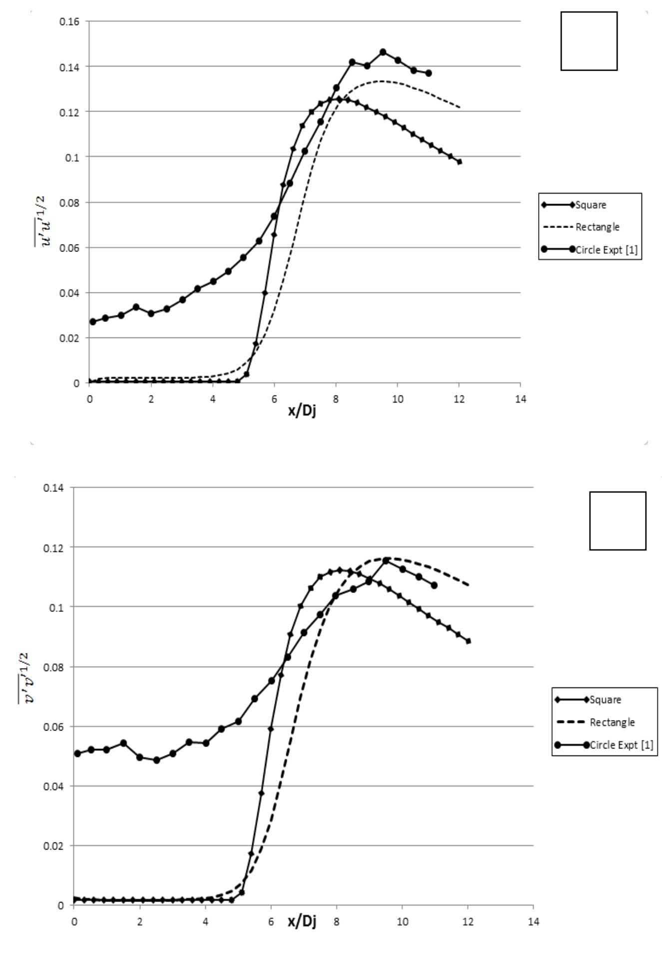

Fig. 6. Variation of (a) $\overline{\acute{u}\acute{u}}^{1/2}$ and (b) $\overline{\acute{v}\acute{v}}^{1/2}$ along the centerline

Variation of $\overline{\acute{u}\acute{u}}^{1/2}$ and $\overline{\acute{v}\acute{v}}^{1/2}$ along the axis from the present work is shown in Fig.6. To achieve the greatest mixing characteristics, it is desirable to have more turbulence in the fluid flow. In case of U_{rms} value there is under prediction of square and rectangular nozzle but it predicts the respective peaks accurately. It can be seen that the peak in case of rectangular nozzle is greater than that in case of square nozzle suggesting more turbulence. In case of V_{rms}, it predicts the maximum peak value of rectangular nozzle accurately. Turbulence in both horizontal and vertical direction decides the extent of mixing downstream. Due to axis switching phenomenon and sharp corners, it helps in inducing more turbulence in large scale mass entrainment as well as at the microscopic level.

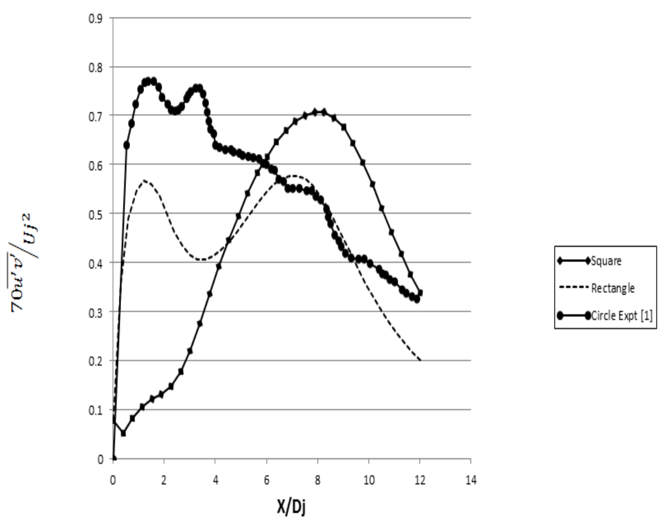

Fig. 7. Variation of peak turbulent viscosity

Fig.8. shows the axial variation of peak value of $\overline{\acute{u}\acute{v}}$ for the three nozzles. It basically suggests the shear stress offered at the nozzle wall and there seems to be under prediction in case of the square and rectangular nozzles as has been the trend in case of RANS simulation. In case of rectangular nozzle, there are two peaks similar to the circular nozzle. In case of square there is only one peak which leads to less mixing. Although, it must be noted that turbulent viscosity does not play the main role in deciding the extent of mixing characteristics.

4. SUMMARY AND CONCLUSIONS

Numerical simulations of a Mach 0.75 jet using the SST k-ω model have been carried out. Both rectangular as well as square nozzle is considered. Grid independent and domain independent study have being carried out to ensure optimized computational time. Almost 1.5 million cells were generated during meshing and steady, pressure based solver was applied to solv RANS turbulence model. Comparison of mean axial velocity at the centerline and along the radial direction at several locations was carried out. The comparison of turbulence characteristics, namely $\overline{\acute{u}\acute{u}}^{1/2}$, $\overline{\acute{v}\acute{v}}^{1/2}$ and $\overline{\acute{u}\acute{v}}$ along the jet centerline was also carried out. In all the above cases, numerical data attained is being compared with experimental data from a circular nozzle from Ref. [1] with the same nozzle outlet area. Results show that although there is under prediction in the simulated RANS data, it is within the permissible limit and can be used for comparison with experimental data of circular nozzle.

Results show that the rectangular nozzle shows the most mixing enhancement characteristic compared to the other nozzles. This can be correlated to the axis switching phenomenon in case of non axi-symmetric nozzles. This phenomenon can be clearly seen in the plotting of turbulence characteristics fluctuating plots. Also it is evident that the SST K-ω model can be used to predict mean flow and turbulence characteristics. The predictions for both square and rectangle nozzle need to be compared with experimental work. Once this is accomplished and the numerical data is being successfully compared with experimental data, RANS calculations can be used effectively for mixing enhancement study and also jet noise aerodynamics.

REFERENCES

[1] Andersson N, Eriksson LE, Davidson L. A study of Mach 0.75 jets and their radiated sound using large eddy simulation. AIAA-2004-3024.

[2] P.S. Tide, V. Babu. Numerical predictions of noise due to subsonic jets from nozzles with and without chevrons.

[3] Gutmark, E. J. & Grinstein, F. F. (1999). Flow control with noncircular jets. Annual Review of Fluid Mechanics, 31, 239–272

[4] Knowles, K. & Saddington, A. J. (2006). A review of jet mixing enhancement for aircraft propulsion applications. Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, 220(2), 103–127.

[5] Krothapalli, A., Bagano_, D., & Karamcheti, K. (1981). On the mixing of a rectangular jet. Journal of Fluid Mechanics, 107, 201–220.

[6] Hammond, A. D. (1966). Thrust losses in hovering for jet vtol aircraft. Technical Report SP-116, NASA

[7] Curtis, P. (1987). Investigation into the behaviour of a single jet in free air and impinging perpendicularly on the ground. Technical Report BAe-KAD-R-RES- 3349, British Aerospace.

[8] Kassab, S. Z., Bakry, A. E., & Warda, H. A. (1996). Laser doppler anemometry measurements in an axisymmetric turbulent jet. Review of Scientific Instruments, 67(5), 1842–1849.

[9] Zhou, D.W., Sang-Joon Lee., Forced convective heat transfer with impinging rectangular jets, International Journal of Heat and Mass Transfer, Vol. 50, (2007), pp. 1916–1926

[10] Sfeir, A. A. 1979: Investigation of three-dimensional turbulent rectangular jets. AIAA J. 17, 1055–1060

[11] Kassab, S. Z., Bakry, A. E., & Warda, H. A. (1996). Laser doppler anemometry measurements in an axisymmetric turbulent jet. Review of Scientific Instruments, 67(5), 1842–1849.

[12] Hussain, F. & Husain, H. S. (1989). Elliptic jets. part 1. characteristics of unexcitedand excited jets. Journal of Fluid Mechanics, 208, 257–320