The International Journal Of Engineering And Science (IJES)
||Volume|| 3 ||Issue|| 01 || Pages || 60-65 || 2014 ||
ISSN (e): 2319 – 1813 ISSN (p): 2319 – 1805

Investigation on Low Speed Rectangular Jet

Gurunath K[1], Maniiarasan P[2], Senthilkumar S[3], Arulselvan K[4]
[1] Post Graduate Scholar, Department of Aeronautical Engineering,
[2] Professor, Department of Mechanical Engineering,
[3] Assistant Professor, Department of Aeronautical Engineering,
[4] Professor, Department of Mechatronics Engineering,
Nehru Institute of Engineering and Technology, T.M. Palayam, Coimbatore – 641 105, Tamil Nadu, India.

ABSTRACT
The jet noise in the aircraft has detrimental effects on the environment, causing economic hardships to those living in and around airports and in the case of non-ideally expanded screeching jets, damage to the structure of the aircraft. Proven methods for reducing jet noise do exist, but at great expense to the aircraft overall performance, increasing the weight and drag of the vehicle. Researchers have been developing various noise reduction techniques, which work on the premise that by increasing the mixing of the jet flow with a bypass flow, and thereby reducing the velocity, noise will be reduced. Other methods for increasing the mixing of jet flows with the ambient air, which have demonstrated the ability to decrease the overall radiated noise in both subsonic and supersonic jet flows include tabs or vortex generators, non-axially symmetric nozzles and simple shaping of the nozzle trailing edge.

Keywords *- Aspect Ratio, Decay, Mixing, Rectangular Jet, Spread.*

Date of Submission: 24 December 2013 ⟺ Date of Acceptance: 10 January 2014

I. INTRODUCTION

Rectangular jets have been gaining importance owing to their desirable mixing, entrainment and acoustic characteristics compared to circular jets. They find applications in combustors and propulsive devices of high-speed systems. Rectangular jets in a row are extensively used for various industrial applications. In chemical engineering the process of mixing, separation and transport among species can be intensified by using a series of jets instead of a circular jet. In power boilers, mixing between fuel and air is improved with a row of combustor jets. In aerospace engineering, the mixing of jets is used for variety of engineering applications, ranging from thrust augmenting equipment to fuel mixing in combustion processes. The examples are thrust augmenting ejectors for VTOL / STOL aircraft. The Multi-element nozzles for noise suppression and improved static augmentation in short augmentors [1].

The problem of noise suppression of turbojet engines has shown a need for turbulence data within the flow field of various types of nozzles [2]. So, the rectangular jets are used to give improved propulsion and reduced noise level. Relatively complete information of the jet properties in the far field is available, but most of the mixing processes of technical interest take place in the near field. Although the number of investigations in this part of the flow has increased remarkably in the last few years. Nevertheless no reliable theoretical prediction scheme is available today to describe the mixing of rectangular jets in the near field satisfactorily [3]. At present, the most effective way for the development of a quantitative prediction method is presumably the numerical investigation of some basic configuration and the generalization of the numerical results by some scaling laws.

II. THE PRESENT STUDY

In the present investigation, experiments are conducted on incompressible jets issuing from a rectangular orifice (120 x 20 mm) at an exit velocity of 50 m/s. The Reynolds number based on the width and exit velocity of the jet is 45000. The experiments are carried out in the near field of the jet.

The computational results such as velocity contour and velocity streamline helps to study the effects:

- Jet mixing
- Jet decay
- Jet spread

Comparing the results of rectangular and circular orifices of same area would spectacle the viable of rectangular orifice.

III. CFD ANALYSIS

The experimentation carried out in the field of Computational Fluid Dynamics (CFD). The objective and scope of the project is to investigate the flow property of parallel jets issuing from rectangular orifice. The near field flow is considered as an important area for investigation. As like practical experimentation CFD also have some methodologies which should be followed. CFD can be used to determine the performance of a component at the design stage, or it can be used to analyse difficulties with an existing component and lead to its improved design. The process of performing a single CFD simulation is split into four components:

- Creating the Geometry/Mesh
- Defining the Physics of the Model
- Solving the CFD Problem
- Visualizing the Results in the Post-processor

These steps were carried out for rectangular and circular orifices and velocity contours of rectangular orifice only presented since, the circular orifice is classical.

3.1 Creating the Geometry/Mesh

This interactive process is the first pre-processing stage. The objective is to produce a mesh for input to the physics pre-processor. Before a mesh can be produced, a closed geometric solid is required. The geometry and mesh can be created in the Meshing application or any of the other geometry/mesh creation tools. The basic steps involved are:

- Defining the geometry of the region of interest.
- Creating regions of fluid flow, solid regions and surface boundary names.
- Setting properties for the mesh.

3.1.1 Defining Geometry of the Region of Interest

As for as the investigation is concerned the rectangular orifice was taken for experimentation.
The profiles was created in ICEM CFD.

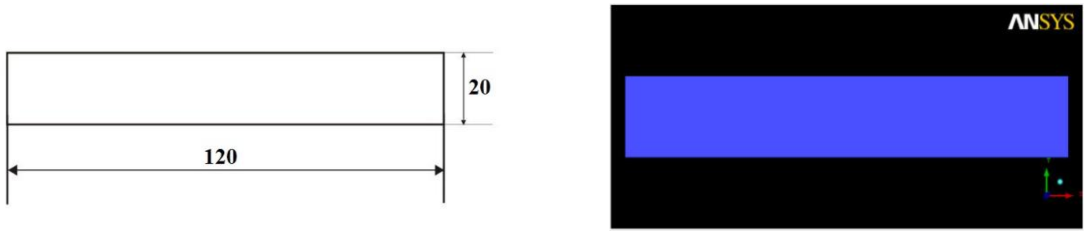

Fig. 1 Rectangular orifice

The rectangular orifice with dimension 120 × 20 mm was created and considered as inlet of the control volume. The geometry of rectangular orifices are shown in Fig. 1.

3.1.2 Creating Regions of Fluid Flow, Solid Regions and Surface Boundary Names

For the second pre-processing stage it needs a control volume to export created geometry in to solver. This control volume can be generated by forming domain which has physical and chemical properties as same as control volume. Due to axis pitching the jet does not hold its rectangular profile, so we need an equally spaced domain. Cylindrical domain satisfies the requirements. In atmospheric condition effect of the jet have some boundary distance, up to this distance the atmosphere will experience the disturbance of the jet. So for this concern the domain has been shaped as cylindrical surface which have the dimension of radius 100 mm and the length 1000 mm (Fig. 2).

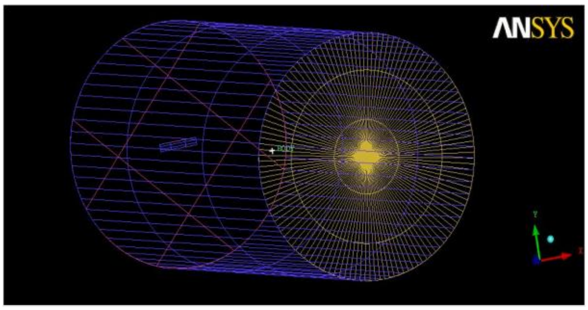

Fig. 2 Cylindrical Domain with Rectangular orifice

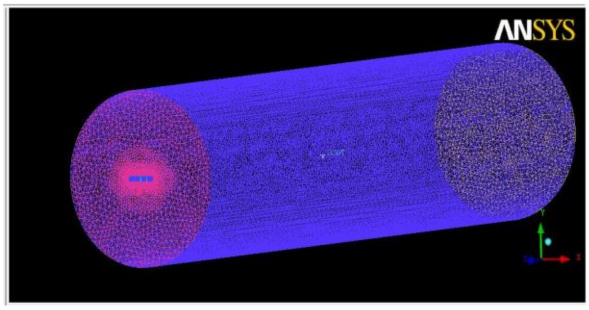

Fig. 3 Meshed geometry

The flow is assumed to be live inside the domain. As early said the conditions inside the domain are similar to the atmosphere. Though the control volume is a closed structure some parts are used to maintain the experimental conditions. The parts present in the flow field are inlet, symmetry, opening and outlet.

3.1.3 Setting Properties for the Mesh

When the geometry is completely created and it shows as a fully closed structure then it is ready to mesh. Meshing is the process by which the part which we are going to analyse is being covered by very small net like structures. In order to obtain the required parameters such as velocity, Mach number etc., by providing node size and refinement. Based on the shapes meshing is divided into two, one is structured mesh and another one is unstructured mesh. In this work the type of mesh used is unstructured tetrahedral mesh. By decreasing element or node size, refinement can be improved so as to get the accurate results wherever needed. The Mesh property differs for each part. This is done because to obtain the accurate values which are required. The prism mesh is created near the orifice inlet, since more accuracy is needed near the inlet. The global mesh parameters are given as, max element size 8 and 1 as scale factor. Finally the needed and the effective mesh were generated (Fig. 3).

3.2 Defining the Physics of the Model

This interactive process is the second pre-processing stage and is used to create input required by the Solver. The meshed files are loaded into the physics pre-processor, CFX-Pre. The physical models that are to be included in the simulation are selected. Fluid properties and boundary conditions are specified.

Boundary conditions for domain:
- Location – Body
- Type of domain – Fluid domain
- Fluid – Air at 25˚C
- Reference pressure – 1 atm
- Process – Total energy
- Turbulence type – Shear stress transport

Boundary conditions for inlet:
- Boundary type – inlet
- Fluid velocity – 50 m/sec
- Fluid temperature – Static temperature (288 K)

Boundary conditions for opening:
- Boundary type – opening
- Location – opening
- Relative pressure – 0 atm
- Opening temperature - 25˚C

Boundary conditions for outlet:
- Boundary type – Outlet
- Location – Outlet
- Relative pressure – 0 atm

Boundary conditions for symmetry:
- Boundary type – Symmetry
- Location – Symmetry

3.3 Solving the CFD Problem

The component that solves the CFD problem is called the Solver. It produces the required results in a non-interactive/batch process. A CFD problem is solved as follows:

i. The partial differential equations are integrated over all the control volumes in the region of interest. This is equivalent to applying a basic conservation law to each control volume.

ii. These integral equations are converted to a system of algebraic equations by generating a set of approximations for the terms in the integral equations.

iii. The algebraic equations are solved iteratively. An iterative approach is required because of the non-linear nature of the equations, and as the solution approaches the exact solution, it is said to converge. For each iteration an error or residual, is reported as a measure of the overall conservation of the flow properties.

The final solution is the exact solution depends on a number of factors, including the size and shape of the control volumes and the size of the final residuals. Complex physical process turbulence, is often modelled using empirical relationships. The solution process requires no user interaction and is, therefore, usually carried out as a batch process. The solver produces a results file which is then passed to the post-processor.

3.4 Visualizing the Results in the Post-Process

The post-processor is the component used to analyse, visualize and present the results interactively. Post-processing includes anything from obtaining point values to complex animated sequences. Examples of some important features of post-processors are Visualization of the geometry and control volumes, Quantitative numerical calculations, Animation, Charts showing graphical plots of variables.

3.4.1 Rectangular orifice

The visualization of rectangular orifice shows the Velocity contour and how the jet accelerated at orifice which is shown in Fig. 4.

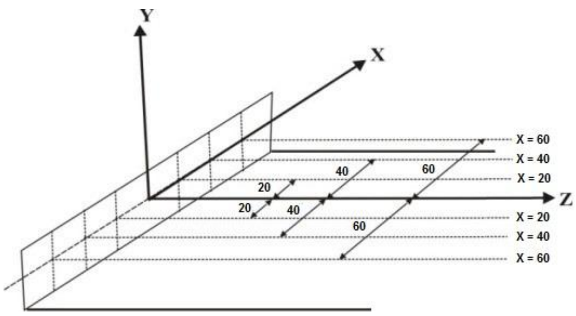

Fig. 4 Schematic representation of co-ordinate system adopted for total velocity measurements in the flow field for rectangular jets.

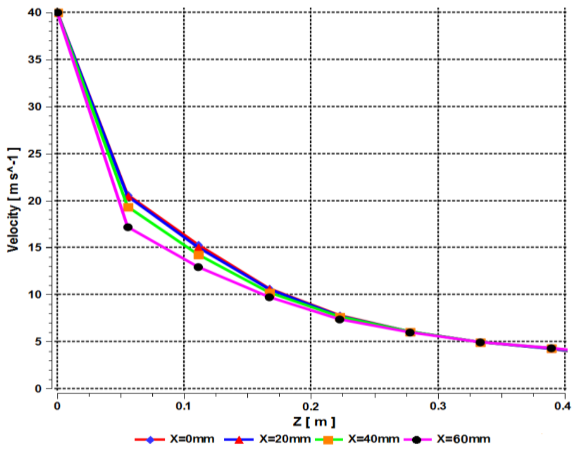

Fig. 5 Span-wise velocity graph for Rectangular orifice

From the graph we came to know about the variation in velocity at x = 0, 20, 40, 60 mm respectively (Fig. 5).The properties such as jet mixing, jet decay and jet spread is also visualized satisfactorily. The result shows how the velocity distribution and potential core changes. The velocity streamline describes how effectively the atmosphere disturbs the flow. Some of the obtained results are shown in Fig. 6 to Fig. 9.

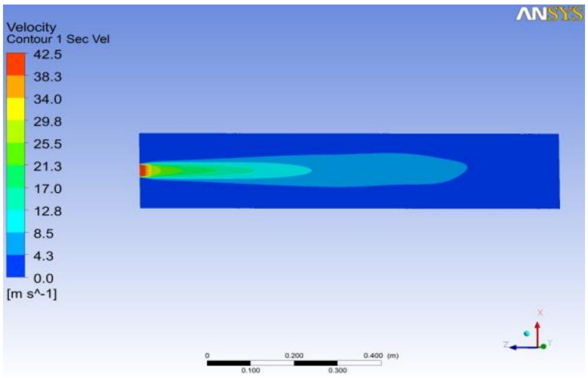

Fig. 6 Velocity contour for Rectangular orifice

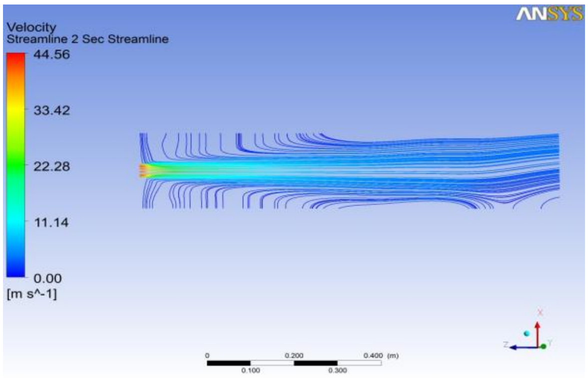

Fig. 7 Velocity streamline for Rectangular orifice

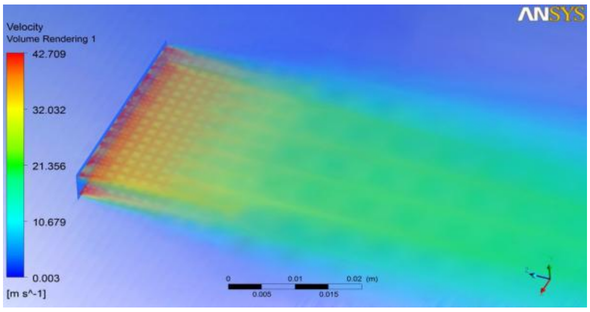

Fig. 8 Volume rendering at near field of the orifice

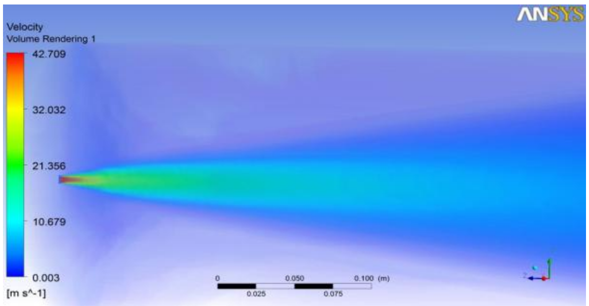

Fig. 9 Jet spread and Jet decay

The Fig. 9 shows the layer between a uniform stream and still air is called a "mixing layer" or "half jet" [4].

IV. RESULTS AND DISCUSSION

The experiments are carried out on rectangular orifice with dimensions 120 x 20 mm using CFD techniques. The exit velocity of the jet is 50 m/s. The Reynolds number based on the width and exit velocity of the jet is 45000. Total velocity measurements are taken along both the jet axis i.e. Z-direction and the transverse direction X-axis of the jet flow field. Jet decay plots are made by taking measurements along the jet centreline. For rectangular orifice the measurements are taken along the jet axis at various X locations of 0, 20, 40 and 60 mm respectively. The readings are taken in both positive and negative directions of X-axis of the orifice. Measurements are taken at various X locations of rectangular orifice for the purpose of comparison. Span wise total velocity measurements are made along the transverse direction of the orifice at Z/De = 1to 12.

4.1 Effect on Jet Mixing

The mixing of a jet can be significantly altered by applying a suitable excitation at the jet orifice. The graph shown in Fig. 10, plotted between the velocity along X axis of rectangular orifice and circular orifice. The Fig. 10 shows the effect of rectangular orifice in span wise velocity. At near field flow the jet mixing increased when the rectangular orifice is introduced. The result shows how the rectangular orifice helps the atmospheric air to flow closer as possible with near field flow.

4.2 Effect on Jet Decay

In addition to the quantities measured directly, the mean stream wise centreline velocity decay is measured. The graph shown in Fig. 11 shows how the jet decay varies for circular and rectangular orifices. And in addition the Fig. 11 shows how the research succeeded as for as the rectangular orifice is concerned. The trend in jet decay shown in Fig. 11 for a rectangular jet seems to agree well with those observed by [5] and [6].

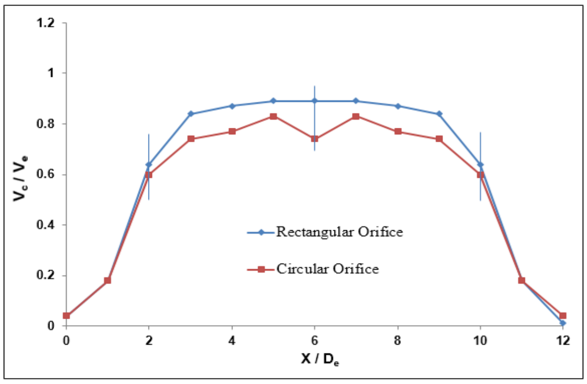

Fig. 10 Comparison of Jet Mixing for Circular and Rectangular jets

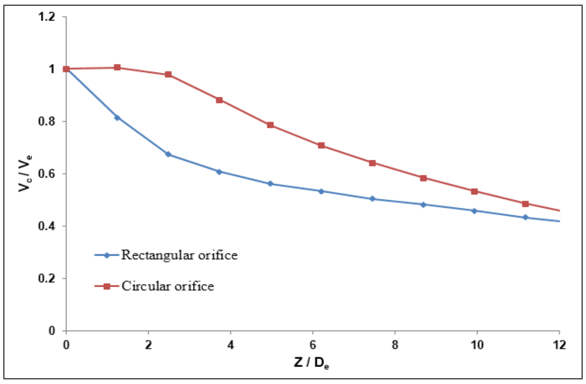

Fig. 11 Comparison of Jet decay for Circular and Rectangular jets

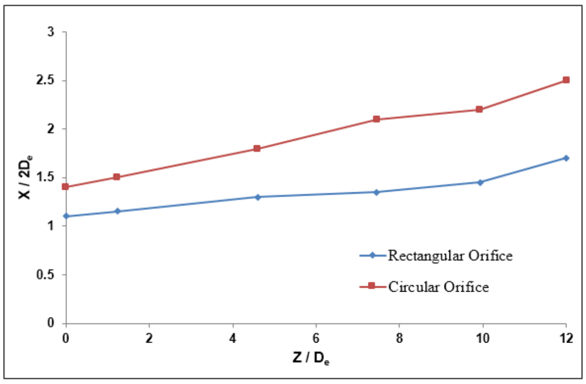

Fig. 12 Comparison of Jet Spread for Circular and Rectangular jets

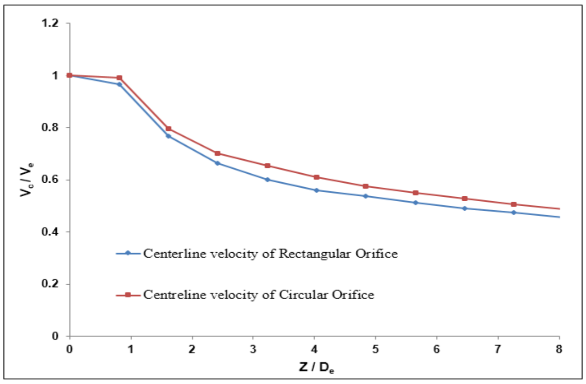

Fig. 13 Variation of jet centreline velocity along jet axis

4.3 Effect on Jet Spread

The spreading of the jet as measured by our objective function is larger at higher Reynolds numbers. If the exit Reynolds number is 20000, the turbulence will be 1 % [7]. The large spreading of the jet can be achieved with a small mass flow actuation if suitable frequencies are chosen, suggesting that the spreading of the jet is larger from the Re = 6000 jet. The graph Shown in Fig.12 shows how the jet spread takes place when the half width velocity of the jet is concerned. The Fig. 12 shows good variation in jet spread between the rectangular orifice and circular orifice. The graph plotted between half width velocities along X-Z plane. In all the experiments, total velocity measurements have been taken along both the centreline (line through intersection point of the diagonals of the rectangular orifice) and the transverse direction. Using atmospheric pressure as the reference pressure, the local velocities have been calculated. It has been found that the near-field mean stream wise velocity distribution is upright for rectangular jet than circular jet [8]. The Fig. 13 shows how the jet decay reduced along Z axis. Up to Z/De=1 both maintained straight line, this length is considered as potential core. The calculated Potential core for the jet is 15.89 mm.

V. CONCLUSION

The major conclusion derived from the present investigation is effect of rectangular orifice on Jet decay, Jet mixing and Jet spread of the jet.

- Increase in aspect ratio result in rapid jet decay. For instance, introduction of rectangular orifice with aspect ratio of 6 increases the jet decay by 4.3 % at Z/De=8. In case of increasing the aspect ratio of the rectangular orifice, the near field flow velocity of the jet is decreased by 12.05 % compared to that velocity of circular jet at the same location.
- Jet mixing is greatly enhanced with increase in aspect ratio.
- Induction of a rectangular orifice results in increased half-width velocities along X-Z plane (Fig. 12), except up to a certain distance closer to the exit of the orifice. Increasing the aspect ratio of the rectangular jets and studying the effect of aspect ratio on fluid dynamic properties of the jet would be the future work of this study.

REFERENCES

[1] Aiken, N.T., "Aerodynamics and Noise Measurements on a Quasi – Two Dimensional Augmenter Wing Model with Lobe – Type Nozzles," NASA TMX – 62, 237, Sept. 1973.
[2] Lawrence, C.J., "Turbulence Studies of a Rectangular Slotted Noise Suppressor Nozzle," NASA TND – 294, 1960.
[3] Junye Wang, Geoffrey H.Priestman and Dongdi Wu., "An Analytical Solution for Incompressible Flow through Parallel Multiple Jets," Journal of Fluids Engineering, Transactions of the ASME, Vol.123, June 2001, pp.407-410.
[4] Bradshaw, P., "An Introduction to Turbulence and its Measurement," Pergamon Press 1971.
[5] Krothapalli, A., Baganoff, D., and Karamcheti, K., "Development and Structure of a Rectangular Jet in a Multiple Jet Configuration," AIAA Journal, Vol.18, No.8, August 1980, pp.945 – 950.
[6] Sarma M.S., Sundararajan.T and Ramjee. V., "Study of Incompressible Jets from various Orifices," Journal of Aerospace Sciences and Technologies, Vol.55, No.2 May 2003, pp 97 – 111
[7] Elbanna, H.Gahin, S., and Rashed, M.I.L., "Investigation of two plane parallel jets," AIAA Journal, Vol.21, No.7, 1983, pp.986 – 990.
[8] Quinn, W.R., "Passive Near-Field Mixing Enhancement in Rectangular Jet Flows," AIAA Journal, Vol.29, No.4, April 1991, pp.515 – 519.