Centerline velocity decay at the plane jets

Science, Geography

The results of computational analysis denoting the centerline velocity decay and the spreading rate of plane jets of air issuing into ambient air conditions are presented.

The air flow is discharged from a circular inlet and through the gradual contour of the nozzle, it exits at two different case scenarios-square and rectangular cross-sections at the outlet. The exit area of both cross-sections is kept same in order that the mass flow rate remains constant. Here, the dimensions have been calculated as multiples of the diameter of the inlet circular cross-section. The developing zone of the free jet has been the main concern of this study. An ' Air Flow Bench' have been fabricated for this study and the experimental observation justifies the result. Pitot-static tube and manometer arrangement were used to acquire data in the experimental analysis. The Reynolds number, based on the normal atmospheric pressure conditions was 73230 in the jets. The results show that the length of the potential core of the rectangular jet is smaller relative to the square one. The spreading rate for the rectangular jet is also higher compared to the other shape.

Introduction

Jet is formed by a flow issuing from a nozzle into the ambient fluid, which can be at rest (free jet), in motion (co-flowing or, counter flowing jet) or, tangential to a solid surface (wall jet). Fluid usually discharges from a high pressure region to that of a low pressure one. Most of the jet flows are typically turbulent in nature and has got so many applications ranging from daily activities to industrial applications such as fuel injections, combustor, jet engine propulsion exhaust etc. One of the most important considerations for jet flow is its entrainment and mixing process with its boundaries. Depending on the initial velocity profile, beyond certain distance, the jet fluid discharging from the nozzle develops flow oscillations in the shear layer. As a result, vortices are formed which increase in sizes and strength with the axial distance. Non-circular jets with corners, such as rectangular, square, and triangular, provide both bulk and molecular mixing.

Various studies have been done over the years on the free jet characterization. A few studies have been devoted to a particular shape only, while others provide a general overview. (Quinn, Azad & Groulx, 2013) analyzed centerline velocity decay and entrainment of triangular and circular jets. Their analysis was done on sharp edged orifice circular jet as well as contoured nozzle following with triangular jets having apex angle of 30° and 160°. Using hot wire anemometry and pitot-static tube they concluded that the velocity decay rate for 160° triangular jet is 0. 18 whereas for the 30° variant, it is only 0. 165. Also the near field entrainment for 160° triangular jet was higher than the other one. (Peram and Boomera, 2016) concluded that the geometrical modification is effective in case of rectangular and square models compared to that of the circular one. They experimented at two pressure heads i. e. at 30 cm of H2O and 40 cm of H2O.

Overall pressure drop is more significant in rectangular jet at 30 cm of H2O, compared to the square one. Centerline pressure drop is more noteworthy in case of the square jet at 40 cm of H2O, compared to the circular and the rectangular jets. The effect of the geometrical variation is minimal at exit for the pressure head of 40 cm of H2O, whereas for the pressure head of 30 cm of H2O, the effect is significant. The incompressible fluid jets can be utilized to increase mixing tendency by a passive control method, using vortex generators like non-circular cross-sections. The comparison was made in terms of the circular one.

Non-circular jets have the potential to entrain ambient fluid more effectively than comparable circular jets. (Ko and Davies, 1971) investigated the pressure field inside the potential cone of a subsonic jet with Mach number less than 0. 45. The existence of a near field inside the core, except that in the first half of the relative diameter downstream also justified the computational results for their experiment. Differences in the characteristics of the pressure field within the potential core resided in the vortex generated noise at a very low jet velocity and eddies were also developed at higher velocities. Near the orifice, high frequency noise was developed and low frequency noise was formed in the downstream location. (Atmaca et al., 2016) investigated flow characteristics around open jet with different outlet area (circular, ellipse, triangular, square, rectangular, pentagonal, and hexagonal) via experimental and numerical considerations.

This study concluded that the highest jet thickness occurred in square and pentagon respectively. The lowest jet lowest thickness occurred in ellipse and circular jets respectively. In upstream flow, the highest velocity occurred in open jet with square outlet and the lowest velocity occurred in open jet with triangle outlet at centerline. In downstream flow, the highest velocity occurred in open jet with rectangle outlet and lowest velocity occurred in open jet with triangle outlet. (Azad, Quinn & Groulx, 2012) investigated the variation of apex angles of triangular jets regarding near field mixing of incompressible and isothermal air. This experimental investigation was carried out for 5 different isosceles triangular orifices (10°, 20°, 30°, 70° & 160°). The Reynolds number was 1670 & Strouhal number was 0. 81 in all the cases. Mean stream-wise centerline velocity decay, spreading and mass entrainment ratios were analyzed and it was concluded that the jet with 160° apex angle has the longest perimeter and has highest mixing rate in the near field. The one-dimensional energy spectra showed that this jet also has the most rapid development. (Mi & Nathan, 2009) observed the turbulent statistical properties of nine differently shaped nozzles.

All nozzles had identical opening area and the AR varied in between 1 to 2. 5. They concluded that the jet asymmetry at the outlet is generally lost hence the mean velocity decay rate is faster. As the fluctuating intensity grows in the near field, it increases the overall entrainment rate. The change of nozzle shape at the outlet had no effect on asymptotic decay rate of centerline velocity on the far field. The analysis was done with a Reynolds number of 15000 approx The changes in exhaust geometry and the study of its effects in the developing zone of the jet is the main concern of this study. It is to be mentioned here, that the various applications of this sort are already in use but its scopes are not acknowledged that much. This is undeniably the most important part of the study because the outcome has to be validated.

Experimental Setup and Data Acquisition

An ‘ Air Flow Facility’ is to be fabricated for studying the air flow measurements as well as the air flow characteristics. The experimental setup consists of the blower, the settling chamber, the flow straightener and the reducer (or, contraction cone). The air flow in the ' Jet Flow Facility' is generated by a centrifugal blower. An axial fan could also have been used. The outgoing airflow from the blower is directed to the settling chamber. It connects the blower to the straightener region. The settling chamber provides time for the generated flow to become stable and reduce the turbulence. Laminar flow is developed in this uniform pipe duct. The inner and outer pipe diameter of the settling chamber is matched with that of the blower in order to mitigate flow loss and turbulence. The length of the pipe should be large enough to allow better flow development. However, this experimental set-up used a blower for better duct flow and a well-designed straightener was also attached alongside. Hence, the length of the settling chamber was curtailed to establish the cost effectiveness of the project. Followed by the settling chamber, the flow straightener is attached. It consists of honeycomb structure of circular cells. Uniform, circular pipes of 1. 5 cm diameter act as the cell of the honeycomb structure. The purposes of using the straightener section are to straighten the flow and to ensure that the flow is free from large eddies. It also allows straight and smooth flow to the discharge section.

Air Flow Facility

The flow from the straightener is channeled through a reducer. The entry dimension of the reducer supposedly corresponds to that of the diameter of the settling chamber. The purpose of using the reducer is to connect two ducts of different diameters and to accelerate the flow. The reducer connects the flow straightener and the nozzle to be attached for the experimentation.

Pipe Assembly

3D printing facilities were used for the square and rectangular cross-section exhaust nozzles. It mostly ensured the gradual shape change from circular to square and rectangle respectively. The axial length is 13. 17cm for both the nozzles. Threaded arrangement was made at the nozzles to ease the experimental readings. However, it could also have been done using flanges alike the other parts of the assembly.

Geometry Specifications (Square)

The dimensions for the both the square and the rectangular cross-sections have been taken as multiples of the circular inlet. The axial length of the nozzle also manifests a dimension relative to the diameter of the circular inlet. This gives an overall control on the change of the dimensions. The pitot static tube was used to get velocity at different location and ultimately figuring velocity profiles at different lines for various ratios.

Geometry Specifications (Rectangle)

The numerical analysis was also done in ANSYS Fluent to see whether the values closely correspond to each other. The outlet faces at the exhaust were visualized from the results obtained from this computational approach. The results of the computational analysis were compared to the experimental ones as well.

IV. Results

The main analysis was focused on the computational approach but initially values (e. g. velocity profile) were taken experimentally to verify the result that is obtained in ANSYS Fluent.