DESIGN AND OPTIMIZATION OF OUTLET SHAPE ON COAXIAL TURBULENT JETS USING CFD

G.Sivaraj[1], D.Lakshmanan[2], S.Nithish[3]

[1,2,3] *Department of Aeronautical Engineering, Bannari Amman Institute of Technology, (India)*

ABSTRACT

The computational analysis was carried out to investigate the effect of flow field generated by two coaxial jets using Ansys Fluent software. The area ratio between the external and internal nozzle was varied as well as the velocity issuing from each of the nozzles. The distribution of the mean velocities, turbulence intensities, and shear stresses were determined for the various cases. The development of the flow field and its approach to a self-preserving state is discussed. The Reynolds numbers based on various shape of nozzle outlet such as Circle, square, and polygon shapes and the velocities were low enough that the flow can be considered incompressible.

Keywords: Annular, Coaxial Jet, Nozzle, Turbulence Intensities

1. INTRODUCTION

A nozzle is a device designed to control the direction or characteristics of a fluid flow as it exits an enclosed chamber or pipe via an orifice. Nozzles are frequently used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that emerges from them

A computational study was carried out to investigate the effect of co-axial nozzle technology with varying cross sections. The co-axial design can be used with water, Class A foam, or compressed air foam. It is especially valuable in CAFS applications because the unobstructed smooth-bore passageway produces a tight stream that maintains a good bubble volume with an excellent reach. This allows firefighters to keep a safer standoff distance when making a direct attack or to effectively coat all parts of a threatened structure from a position on the ground in a defensive mode.

The background study was carried out about co-axial nozzles with their applications. The computational and experimental results were also carried out. Based on the literature survey coaxial nozzle were a designed with varying cross sections. Computations were performed to understand the flow field around a scaled down model of a typical nozzle with annular. Computations using the commercially available software FLUENT 6.3.26 were carried out for two dimensional and three dimensional fully developed flows. A validation test was performed by referring to the same type of model at a same inlet velocity

II. COAXIAL NOZZLES

Much data have been accumulated for laser cutting using various nozzle shapes with various standoff distances and pressures. But it appears to be little agreement on methods for obtaining a consistently high cutting quality. Usually the nozzle is positioned close to the work piece (0.3-1.3 mm) and with low nozzle pressures. When higher nozzle pressure is used, the cutting speed is raised but the cutting repeatability cannot be ensured.

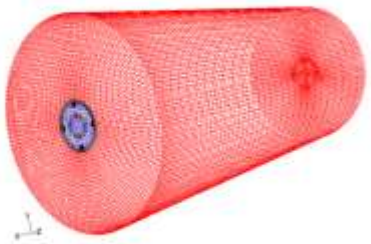
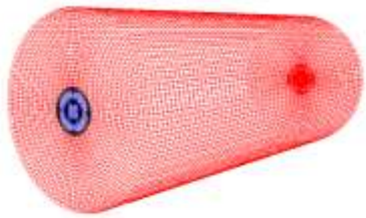
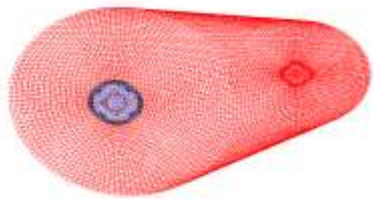
III. DESIGN OF NOZZLE

Frequently the goal is to increase the kinetic energy of the flowing medium at the expense of its pressure and internal energy. Nozzles can be described as convergent (narrowing down from a wide diameter to a smaller diameter in the direction of the flow) or divergent (expanding from a smaller diameter to a larger one). A de Laval nozzle has a convergent section followed by a divergent section and is often called a convergent-divergent nozzle ("con-di nozzle").

3.1 Different Shapes of Nozzle Outlet Cross Sections

There are nozzle outlet shapes classified by circle and square shapes with annular.

1. Circle without annular
2. Square without annular
3. Square with annular

Fig(1). Circular without annular **Fig(2). Square with annular** **Fig(3): Square without annular**

IV. COMPUTATIONAL ANALYSIS

For thorough understanding of the complex flow field around typical Co-axial Nozzle with annular, axi-symmetric computational simulation of the flow field can be made useful along with the experimental testing. This can be done either by developing a code or by available commercial CFD software. In the present study computation has been attempted for flow over a nozzle with different types of cross section model with the commercially available software FLUENT 6.3.26. Present chapter describes briefly about the solver, solution methodology, preprocessing, solver settings and post processing.

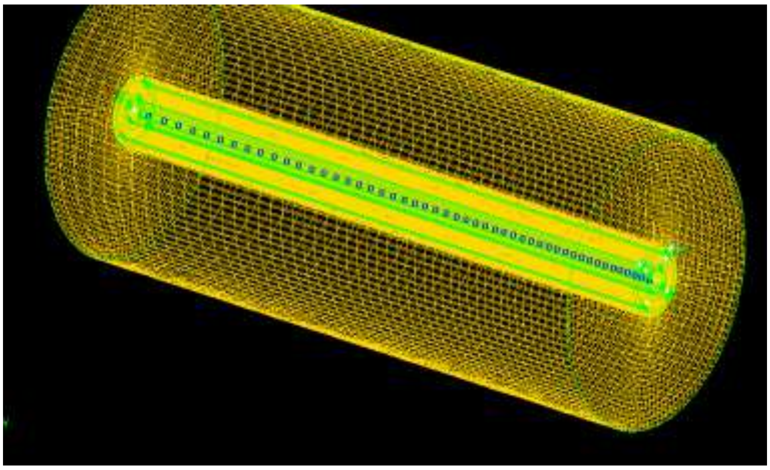
4.1 Geometry Design

Any CFD analysis can be divided into three broad categories namely, preprocessing, solver settings and post processing. GAMBIT is an aid for preprocessing of the problem in FLUENT. It is a software package designed to help analysts and designers build a mesh around the geometry under consideration. GAMBIT receives user input by means of its graphical user interface (GUI). Geometry creation in GAMBIT is done with the help of required commands from the geometry creation tool pad. The geometry creation tool pad contains command buttons that allows performing operations which include creating vertices, lines, faces, volumes etc.

Meshing in GAMBIT can be done in various forms namely edge meshing, face meshing and volume meshing. Meshed edges, faces and volumes can be copied, moved, linked or disconnected from one another.Creation of the geometry can also be done in any of the available software like AutoCAD, CATIA, and PRO-E etc. Created geometry can then be imported into GAMBIT and meshed. After meshing is completed and the zone types are specified, the created mesh can be exported as a mesh file in to various different solvers like FLUENT 5/6, FLUENT 4, POLYFLOW, ANSYS etc. The exported mesh is ready to be fed into the solver for obtaining a solution of the given problem.

4.2 Grid Generation for Nozzle Model

A structured grid was generated for 2D simulation of flow around the nozzle model. The grid was made very fine at the geometry surfaces and coarsens away from the body. The overall domain was selected based on several iterations, boundary conditions and finally a domain extending 5 times the major length of the nozzle model. The extents of the domain were evaluated from inviscid simulation of the problem. A total of 50,000 cells were used in the grid system. Figure shows the grid system with boundary conditions, in the vicinity of the geometry and a close up surface grid near main body and strap-on nose regions respectively. In order to capture the shocks more accurately, finer mesh cells were created near the surface of the model using appropriate edge mesh distribution.



Fig(4).Solid View of Entire Domain With Model

4.3 Boundary Conditions

Pressure far field boundary condition was set for the pressure outlet (outer domain), where it is needed to specify the atmospheric pressure. Inlet 1 and inlet 2 boundary condition was set to the velocity inlet (surface from which the flow enter), Wall boundary condition is assigned for the model surfaces and domain boundary extents other than interior.

V. COMPUTATIONAL RESULTS

Computations were performed to understand the flow field around a scaled down model of a typical nozzle with annular. Computations using the commercially available software FLUENT 6.3.26 were carried out for two dimensional and three dimensional fully developed flows. A validation test was performed by referring to the same type of model at a same inlet velocity. The details of the computations are discussed in this chapter.

Two dimensional and three dimensional computational simulations were carried out for studying the flow field around typical nozzle geometry at 30 m/s inlet 1 and 25 m/s inlet 2. in detail in the following sections.

5.1 Validation of Computational Procedure

Verification and validation are the primary means to assess accuracy and reliability in computational simulations.

Several computational tests were performed on a typical nozzle at a 30 m/s inlet 1 and 25 m/s inlet 2 for verifying and validating the computational grid and turbulence model that are going to be adopted for the present work.Inviscid, laminar, S-A, standard k-ε and standard k-ω models were tested for verifying the most suitable turbulence model for present case of a typical nozzle configuration. The solver settings operational conditions, material properties, and boundary conditions were set according to the present typical space launch vehicle problem. The problem was iterated till the residuals of continuity, momentum, and energy converged to a value of 10^{-3} and the scalars nut (SA) residuals converged to a value of 10^{-5}.

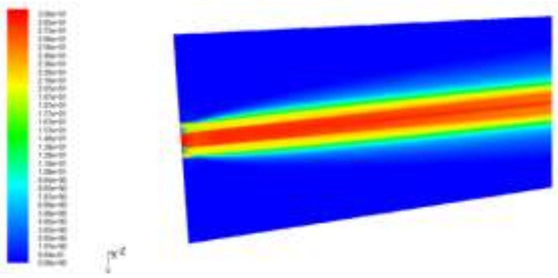
5.1.1 Comparison of Circular Cross Section with and Without Annular

Force Report						
zone name	pressure Force	Viscous Force	total force	pressure coefficient	viscous coefficient	total coefficient
Without Annular	35984.81	0.024961	35984.83	65.27856	4.53E-05	65.278608
With Annular	29511.12	0.03901	29511.16	77.09026	0.000102	77.090366

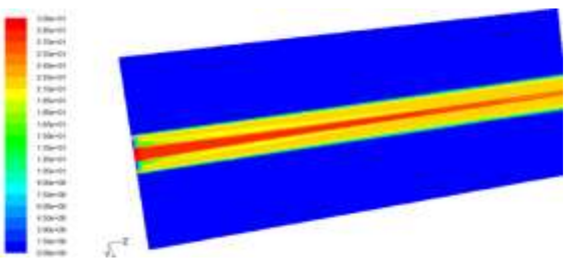
5.1.2 Comparison of Rectangular Cross Section with and Without Annular

Force Report						
zone name	pressure Force	Viscous Force	total force	pressure coefficient	viscous coefficient	total coefficient
Without Annular	30332.31	0.038046	30332.35	49522.14	0.062116	49522.2
With Annular	30282.81	0.0437046	30282.85	59622.17	0.062116	59622.5

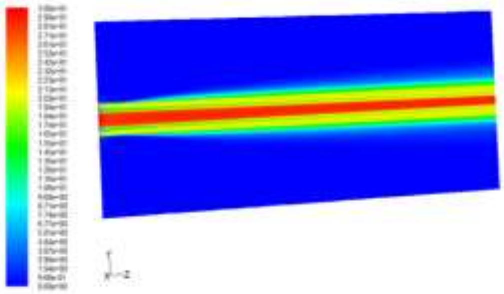
VI. VELOCITY MAGNITUDE OF JET STREAM



Fig(5).Circular Cross Section Without Annular



Fig(6).Rectangular Cross Section With Annular



Fig(7). Rectangular Cross Section Without Annular

VII. CONCLUSION

Based on computational investigations of co axial circular flow jet using annular, the following conclusions have been observed:

- The potential core length of Circular cross section without annular jet was less than the Circular cross section with annular Co-axial jet and it was longer for circular co-flow jet when compared to the non-circular co-flow.
- The velocity decay of single jet was less than the Co-axial jet. It was significant in size for crusi form jet. The velocity decay along the radial direction of single circular jet was lesser than the Co-axial circular jet, and it was faster for non-circular co-flow jet when compare to circular co-flow jet compared with circular jet and related to vortices generated at the corner of jetting.

REFERENCE

[1]. Nob hide Kasagi, “Jsme International Journal”, Publication Year:2006

[2]. ParvizBehrouzi And James J, McGuirk ,”Journal Of Fluid Science And Technology”, Publication Year:2007

[3]. Toru KosoAnd Takuya Kinshasa, “Journal Of Fluid Science And Technology”, Publication Year:2008

[4]. Truong V.V, Hideyuki Takakura, Jonn.C.Wells And Takashi Minemot,“Journal Of Solid Mechanics And Materials Engineering”, Publication Year:2010

[5]. Jeff Kastner, Chris Harries And Ephraim Gut mark,”Journal Of Fluid Science And Technology”, Publication Year:2011