

CFD Analysis of a Turbulent Jet Flow

C. Bhargavi

PG Student in Department of Mechanical
SIETK, Puttur,
A.P, India
udaybhargavi09@gmail.com

G.Ravindra Reddy M.E , MISTE.

Associate Professor, Department of Mechanical,
SIETK, Puttur,
A.P, India
gururavindrareddy@gmail.com

Abstract— The Turbulent jets are stumble upon in industrial apparatus, combustion chambers and different types of mixers. The co-Axial turbulent jets can create a complex flow with an outer jet developing under asymmetric conditions by considering high jet velocities for both core and annular jets. The main objective is to increase the penetration length of the nozzle by modify or designing the outlet shape of the jet. The external and internal nozzle area ratio was varied as well as velocity issuing from the two nozzles. In all these cases the calculations of turbulence intensities, shear stresses distribution of the average velocities .The state of flow field and the state of approach to a self-preserving condition is analyzed. In order to improve the Thrust efficiency the Reynolds numbers based on various shapes of nozzles outlet such as Circle and square with and without annular was determined and studied by using CFD software.

Keywords- Turbulence, co-Axial, CFD software.

I. INTRODUCTION

The co-axial medium which allows in two separate waterways to fit in to one nozzle in inside and the other. The smooth-bore waterway passes directly through the center of the nozzle cavity to produce a solid stream and the fog waterway passes in the outer portion of the nozzle cavity emerge in a full fog stream. The technology which combines the two nozzle designs one to achieve and the effectiveness of the both. The co-axial design will be used by water, by the Class A foam or by compressed air foam. It is valuable application in CAFS because the unclear smooth-bore way produces a tight stream that maintains a good bubble volume with an excellent reach. This allows firefighters to keep a safer standoff distance when contacted a direct attack to effectively coat to all parts of a structure from the position on the ground in a defensive mode.

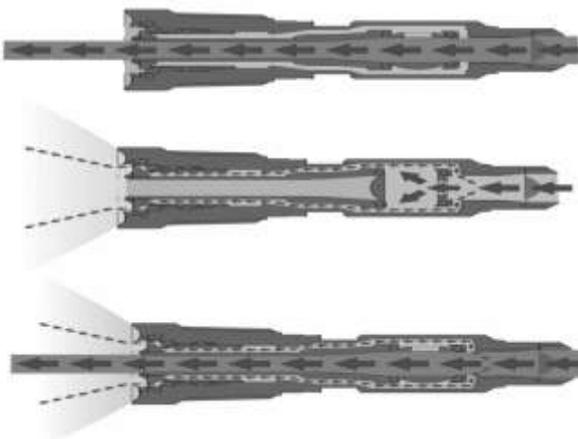


Fig 1 : Nozzle stream flows

II. INTRODUCTION TO CFD

In the present study computation methods , computation has been attempted for a flow through a nozzle with different types of cross section models.

CFD analysis broadly divided into three broad categories namely, preprocessing, solver settings and post processing. GAMBIT is support 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 the consideration. GAMBIT receives user input data by means of its graphical user interface (GUI).

The GAMBIT GUI makes the basic steps of building, meshing, and assigning zone types to a model in a simpler and simple intuitive manner, and also it is versatile enough to accommodate a wide range of modeling applications The FLUENT solver has the modeling capabilities:

Flows in two or three dimensional geometries using solution-adaptivetri/tetrahedral, quadrilateral/hexahedral, or mixed (hybrid) grids that include prisms (wedges) or pyramids (Both conformal and hanging-node meshes are acceptable), Incompressible or compressible flows, Steady-state or transient analysis, Inviscid, Laminar and Turbulent flows. Both single-precision and double-precision versions of FLUENT are available in all the computer platforms.

III. DESIGN OF COAXIAL NOZZLE

Convergent-divergent nozzles can accelerate fluids that have choked in the convergent section in to supersonic speeds. This Convergent- Divergent process is more efficient

than allowing in a convergent nozzle to expand in to supersonically externally.

The shape of the divergent section is also ensures in the direction of escaping gases is in directly backwards, sideways component would not contribute to the thrust. Outer diameter of the nozzle is 35mm and inside diameter of the nozzle is 16mm and width is taken as 4mm.

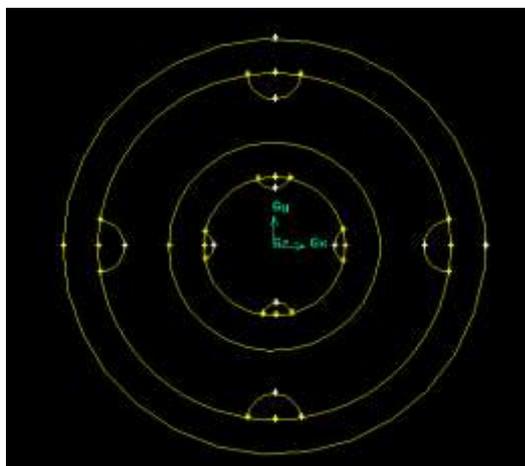


Fig 2: Creation Of The Geometry

IV. NOZZLE GRID GENERATION WITH DIFFERENT OUTLET CROSS SECTIONS

A structured grid was generated for 2D simulation of flow around the nozzle model. 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. Total of 50,000 cells were used in the grid system.

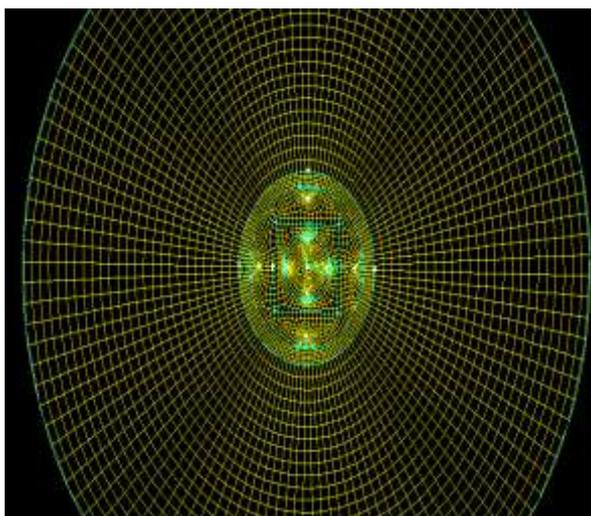


Fig 3. Nozzle Grid Generation With Square Cross Section

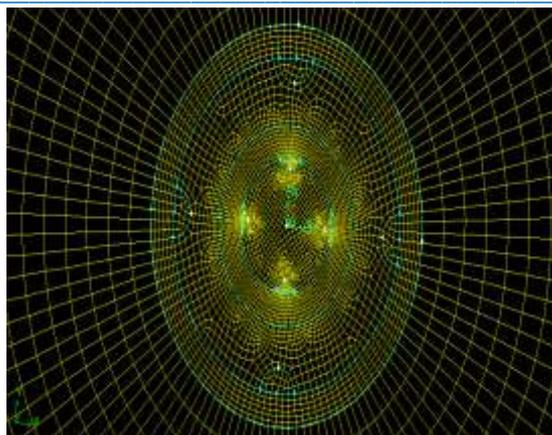


Fig 4. Nozzle Grid Generation With Circle Cross Section

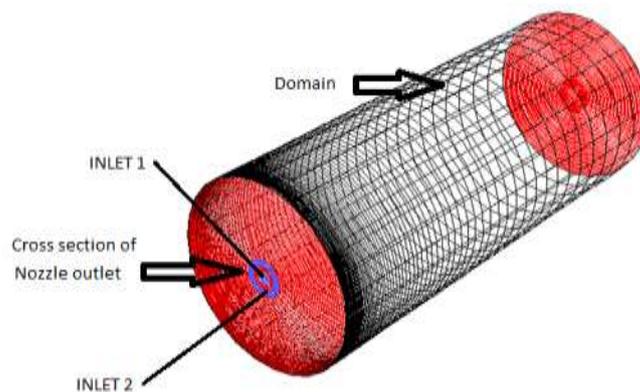


Fig 5 Computational Design model

V. STRUCTURAL MESH AND NOZZLE SOLID VIEW

Created geometry imported into GAMBIT software and then meshed. In GAMBIT meshing can be done in various types as edge meshing, face meshing and volume meshing. To perform grading and meshing operation on the edges the following parameters can be used edge(s) in which the grading specifications, schemes, mesh node spacing in number of intervals, and also in edge meshing options.

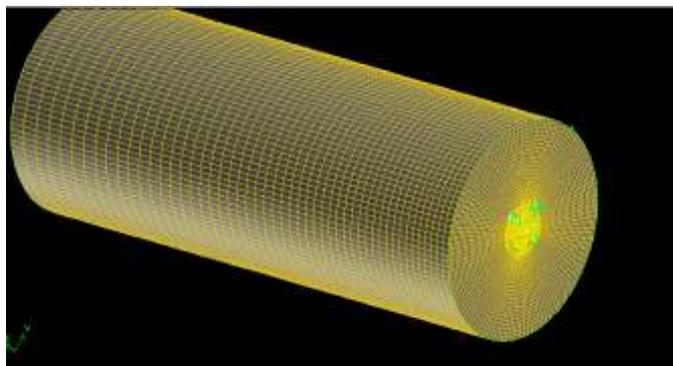


Fig 6 . Solid View Of Entire Domain With Element

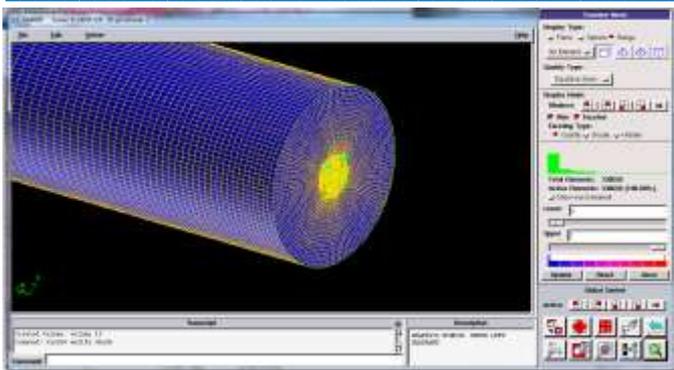


Fig 7. Elemental Examinations

VI. BOUNDARY CONDITIONS

The boundary pressure field condition was set to the pressure outlet, in order to specify the atmospheric pressure. The 1 and 2 inlet boundary condition set to velocity surface inlet in which the flow enter, for the model surfaces wall boundary fixed and interior extends to domain boundary

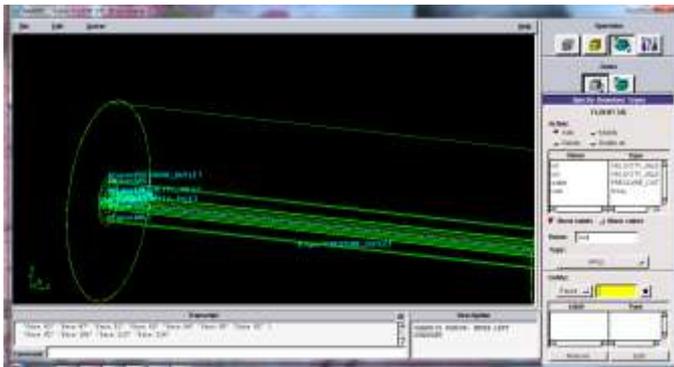


Fig 8. Applying Boundary Conditions

VII. RESULTS AND DISCUSSIONS

Computations were performed to in order to flow field in around a scaled down model of typical nozzle with and without annular. Computations using the software FLUENT were carried out for two dimensional and also in three dimensional fully developed flows. A test for validation is performed by referring to same type of model at a same inlet velocity. A fine standard S-A turbulence model was adopted for viscous computation after checking with other turbulence models like namely $k-\omega$ and standard $k-\epsilon$.

The solver settings in operational conditions, in material properties, and also in boundary conditions were set according to typical space launch vehicle problem. And problem was iterated until the residuals of continuity, momentum, and energy converged to the value of 10^{-3} and also the scalars nut (SA) residuals converged to the value of 10^{-5} .

A Typical time taken for Inviscid problem 250 iterations per hour and for viscous three dimensional problems were 150 iterations / hour. For all these cases, an average 3000 iterations were performed until desired convergence to be obtained. The

Inviscid analysis also performed for comparing the results obtained for viscous flows.

Verification and validation are mainly by means to assess accuracy and reliability in computational simulations. The operating inlet pressure is taken as 2MPa and Number of computational tests are also performed on a typical nozzle of 35 m/s inlet 1 and 25 m/s inlet 2 in order to verifying and validating the grid computational and turbulence models are going to be adopted for this present work.

A) VELOCITY MAGNITUDE OF JET STREAM :

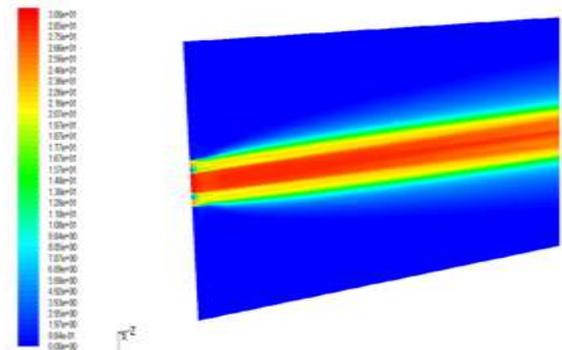


Fig 9. Circular Cross Section Without Annular

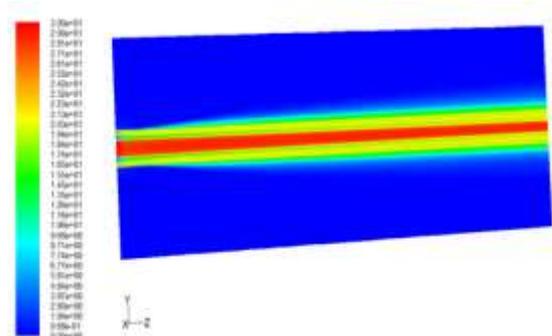


Fig 10. Circular Cross Section With Annular

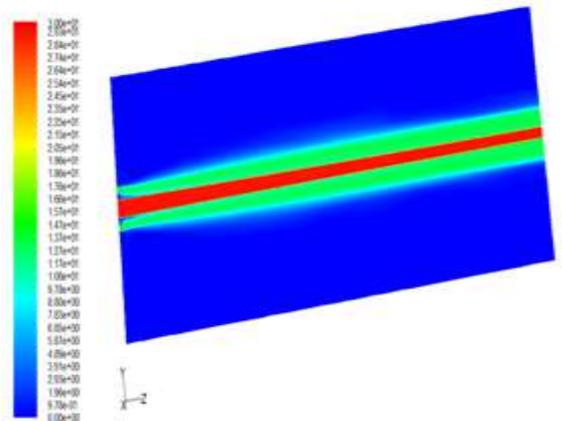


Fig 11. Square Cross Section Without Annular

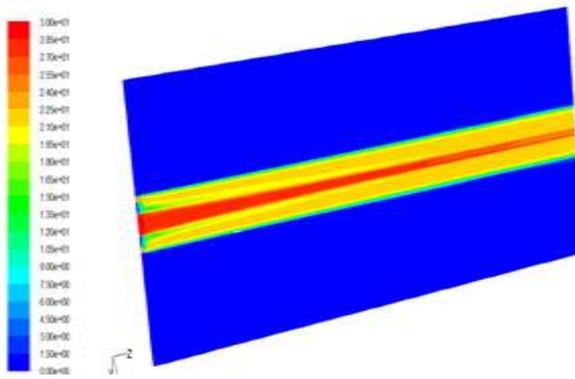


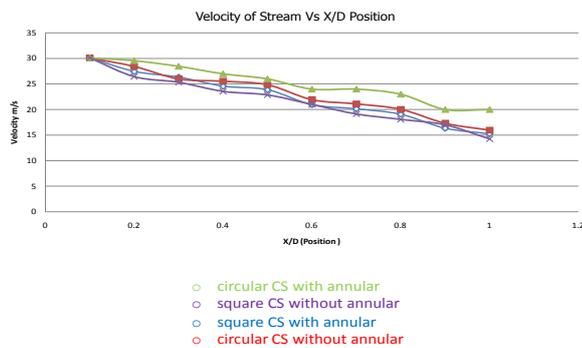
Fig 12. Square Cross Section With Annular

- [3] SEUNG TAEKOH, SETSURO HIRAOKO, YUTAKA TADA "Heat Transfer Inlet Mixing Vessel With Rotating Nozzle Around The Vessel Axis."
- [4] KOICHI TSUJIMOTO, KOJIAO, TOSHIHIKO SHAKOUCHI & TOSHITAKE ANDO "Numerical Investigation Of Flow Structures & Mixing Performances Of Vector - Controlled Free Jet Using DNS".
- [5] YUMIKO OTOBE SHIGERUMATSUO, MASANORI TANAKA, HIDEKO KASHIMURA "A study On Characteristics Of Under Expanded Condensing Jet".
- [6] T.J.CHUNG, "A Text book of Fluid dynamics" Cambridge university press.
- [7] JOHN ANDERSON, "Computational Fluid dynamics"
- [8] H VERSTEG, W .MALALASEKERA, "An Introduction to Computational Fluid Dynamics: The Finite Volume Method" Pearson prentice hall.

B) GRAPH

x-axis is the ratio of length to diameter (X/D) position.
y-axis is the velocity in m/s.

GRAPHS



VIII. CONCLUSION

The simulation results are based on the viscous model and implicit formulation. the computational results obtained by using different nozzle cross sections such as circle and square with annular, without annular.

The co-axial technology allowing two separate waterways to fit in one nozzle-one inside the other. This technology blends the two nozzle designs into one and achieving the effectiveness of both.

The best results obtained during circular cross section with annular.

REFERENCES

- [1] TAKAHIRO KIWATA, TAICHIUSUZAWA " Flow Visualization & Characteristics Of A Co Axial Jet With A Tabbed Annular Nozzle".
- [2] TAKAHIRO KIWATA ,SHIGEOKIMURA "Flow Structure Of A Coaxial Circular Jet With Axisymmetric & Helical Instability Modes".