

Design and analysis of a clustered nozzle configuration and comparison of its thrust

¹Abdul Hadi Butt, ²Asfandyar Arshad
 Department of Aeronautics and Astronautics
 Institute of Space technology, Islamabad
¹hadi_ul_lisan@yahoo.com
²asfand939@gmail.com

Abstract—The objective of this paper is to study the thrust losses (if any) encountered in clustered configuration of nozzles. It involves the design and analysis of a clustered configuration of nozzles using Fluent. First the single nozzle configuration is analyzed for axial thrust. Further clustered configuration is analyzed provided boundary conditions i.e. chamber pressure, temperature and area ratios. It includes the comparison of single nozzle and clustered nozzles.

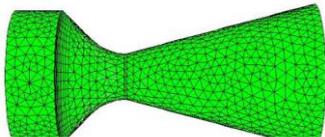
Keywords—CD nozzle, cluster, thrust, fluent, ansys.

I. INTRODUCTION

Exit flows of clustered nozzles configuration is very considerable study in aero-propulsion applications. Zero pressure operating condition and floating exit pressure results in a no shock flow across the nozzle. Firstly a single nozzle analyzed in Ansys fluent® gives us the exhaust values of thrust coefficient and Mach number. The twin nozzle jet modeled is analyzed in the same manner. Mixing length for single and twin nozzle is calculated by the equations used for mixing length in acoustic behaviours analyzed by Kandula and Bruce [1] showing that clustering also decreases the mixing length..

II. SETUP OF MODEL

The flow field inside and outside high-pressure nozzle is numerically simulated. The cylindrical tube is 1m long. The inlet diameter of nozzle is 0.46m and the exit diameter of the cylindrical tube is 0.506m. External rectangular block is considered as far field of outlet where external flow is analyzed. In cluster distance between nozzle was specified.



Model

Fluid
air with

Figure 1-single nozzle geometry

Mathematical

used in simulation is ideal properties. Real

fluid is viscous. It should comply with mass conservation law and Newton's second law.

A. Continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x} (\rho V x) + \frac{\partial}{\partial r} (\rho V r) + \frac{\rho V r}{r} = 0$$

B. Energy Equation

$$\frac{\partial}{\partial t} (\rho E) + \nabla \cdot (v \cdot (\rho \cdot E + p)) = 0$$

III. COMPUTATIONAL METHODS

A convergent divergent nozzle configuration is used for computational analysis with an outlet to throat area ratio of 10

A. Boundary Conditions

Inlet pressure of 55 bars and an inlet temperature of **2900** k were provided in the boundary condition setup of the fluent software. At the outlet backpressure was set free to float where as in operating conditions we have zero operating pressure in order to minimize the reverse flow in compressible conditions.

Providing the reference conditions, we have inlet area of **0.16** and length of our convergent divergent nozzle to be **1** meter. Fluent calculates the velocity itself from the given boundary conditions.

The Turbulence models used in CFD analysis were k-ε and energy equation was implemented. A density-based solution was run with a courant number range of 5-50 .Solution was initialized from inlet. Named selections were inlet, outlet and wall.

B. Mixing Criteria

The simulations in the literature review showed that mixing in cluster causes the thrust to decrease .we used the model of Kandula and Bruce[1] showing that clustering causes mixing of flows and proposed a mathematical model for calculations of mixing regions.

The equations used for calculating mixing length are as follows which suggested that mixing length comes out to be 18 meters in a twin jet nozzle but the actual thrust, which can affect the exhaust performance, is the thrust at the exit mean centerline. Mixing length is given by the ratio

$$\frac{X_4}{X_t} = \frac{2\left(\frac{r_2}{r_{e3}} - 1\right)}{\sqrt{n} - 1}$$

Where

$X_4 =$ Mixing length

$X_t =$ Core length

$n =$ Number of nozzles

$\frac{r_2}{r_{e3}} =$ ratio of cluster radius to exit radius of individual nozzle

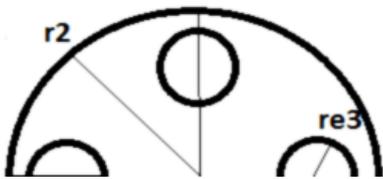


Figure 2-cluster radius-individual radius of nozzle

Conclusion in the results clearly matches up with the theory that mixing starts in cluster reducing thrust performances relatively.

Mixing length was observed when an animation for the streamlines of dual nozzle was run. The far field flow of the

Nozzle was analyzed by sketching a rectangular surface just outside the nozzle.

This rectangular surface was named as outlet in meshing surfaces. Mixing length came out to be out of the constraints of geometry but the reasoning says that thrust at the mean exit line is the actual thrust of the nozzle and if there is any influence it is calculated but putting a probe at the exit.

IV. METHODOLOGY

Geometry from the known dimensions is designed. Geometry generated in an .igs extension is further moved to meshing. A solver is to be chosen for meshing. For 3 dimensional models meshing in ICEM is very helpful which was chosen.

A. Meshing

Then on exporting mesh, they form msh. file which is later used for simulation in fluent software. The meshing of nozzle is done with inflation control fixed “Program Controlled” and named selections are defined as inlet outlet and wall. The meshing is updated. After this the scaling is done. Millimeter is set as the scale. Grid created was changed to mm. The boundary Conditions are positioned in the same software for the geometry, like wall, pressure inlet, pressure outlet etc.

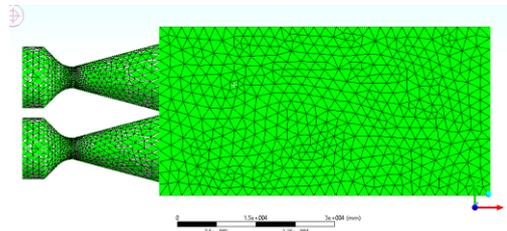


Figure 3-cluster of nozzles

B. Solver Specifications

The solver is defined first. Solver is taken as density based and formulation as implicit, ISO as 3D and time as steady. Velocity formulation as absolute and gradient options as “Green Gauss cell based” are taken. Energy equation is taken into consideration. The viscous medium is also taken. They analysis is carried out using **K-epsilon** turbulence model. The selection of material is done. Material selected is gas. The properties of gas taken as follows:

- _ Density as Ideal gas
- _ Cp (Specific heat capacity) = 2429.3 J/Kg.K
- _ Thermal Conductivity = 0.434W/m.K
- _ Viscosity = 9.24 e-5 Kg/m.s

The analysis is carried out under operating condition Of zero Pascal. Gravity is not taken into consideration.

Simulation Results (single nozzle)

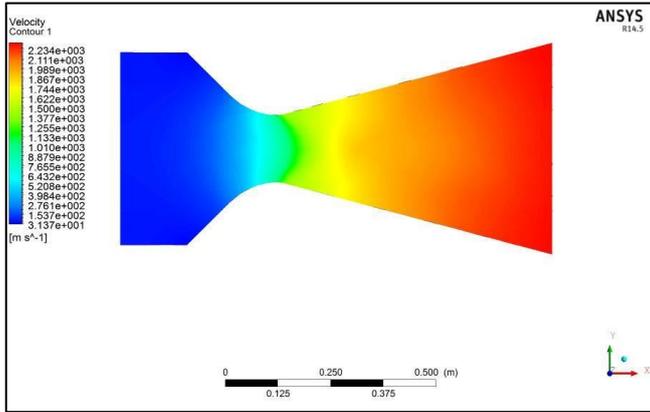


Figure 4-velocity contour

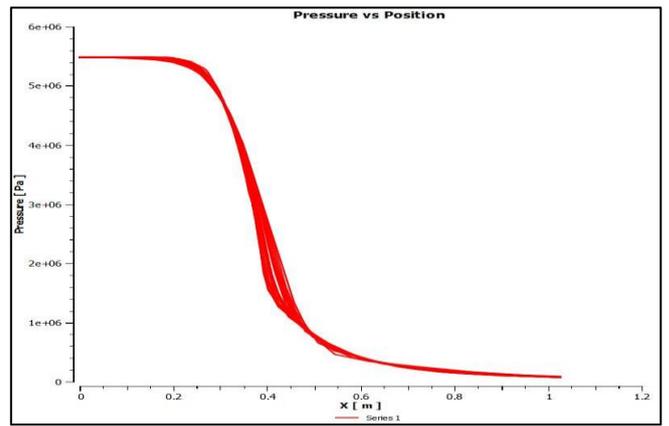


Figure7-Pressure vs. position

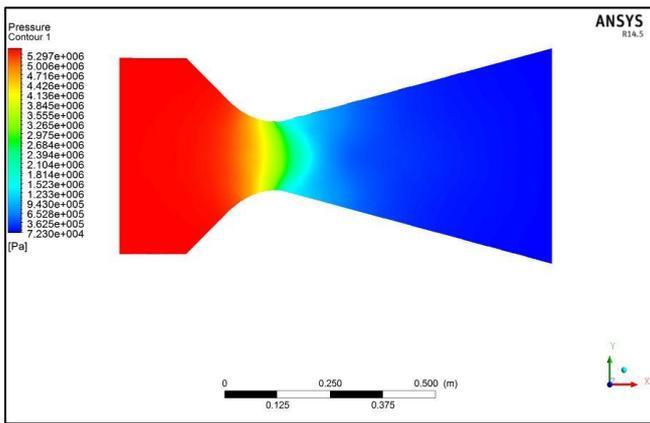
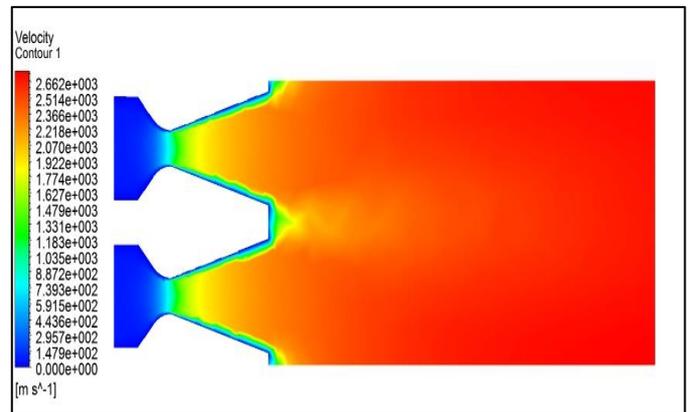


Figure 5-pressure Contour



Simulation Results (Cluster)

Figure8-Velocity Contour(Cluster)

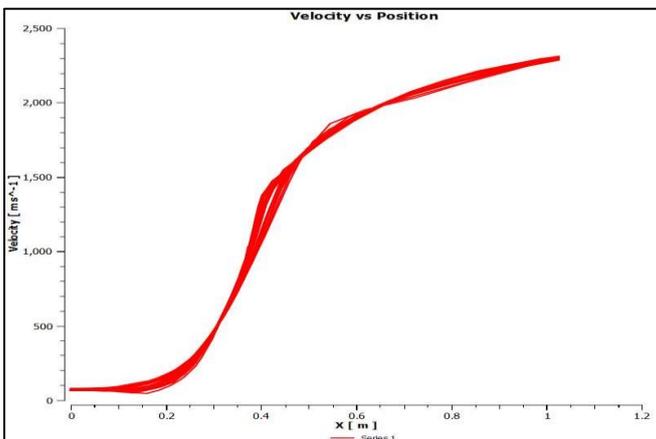


Figure-6 Velocity vs. position

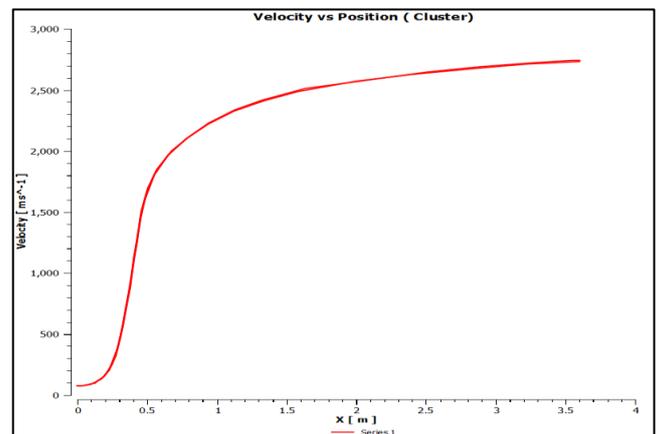


Figure9-Velocity v.s Position(Cluster)

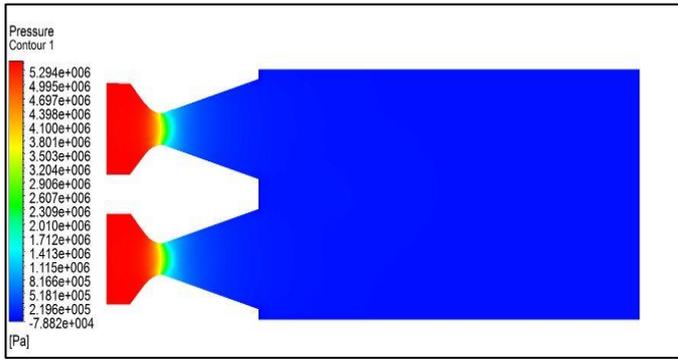
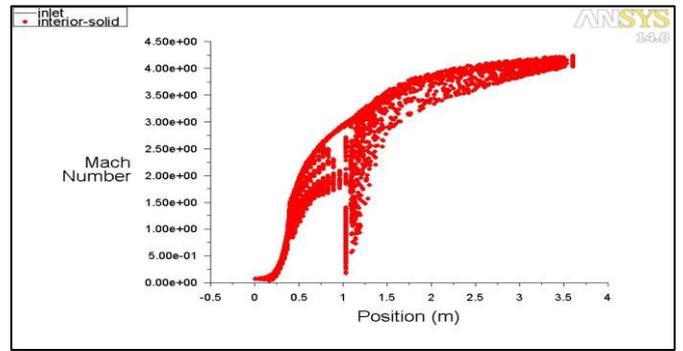


Figure10-Pressure Contour (Cluster)



Mach no vs. position (cluster)

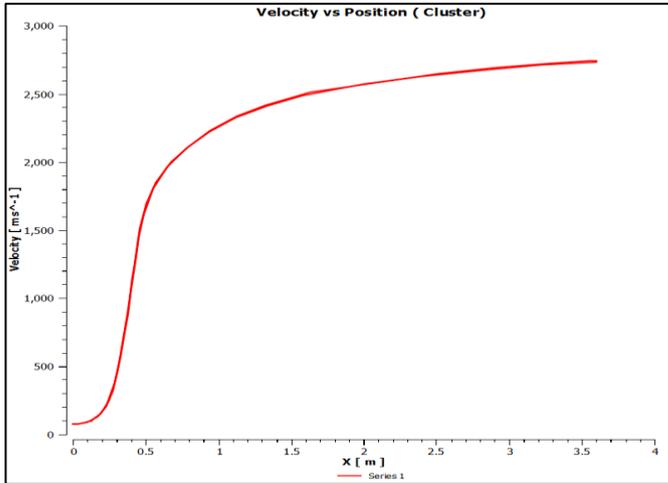


Figure11-velocity vs. position (cluster)

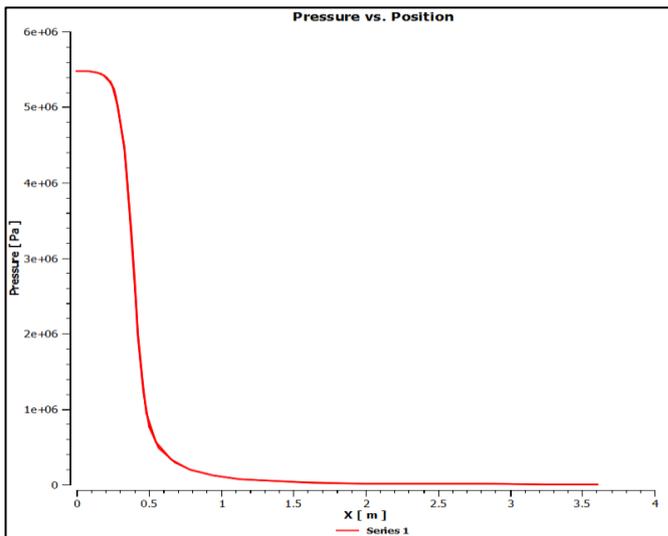


Figure12-pressure vs. position (cluster)

V. RESULTS AND DISCUSSION

Thrust Equation used for calculations is given by

$$F = \dot{m} V_e + (P_e - P_a)A_e$$

Provided the exit area, ambient pressure, exit pressure (simulation probe value) and mass flow (simulation value) the thrust calculated is given below

Thrust comparison of single and clustered nozzle

Configuration	Thrust (KN)	Velocity(m/s)
Single Nozzle	192.4	2234
Single Nozzle (cluster)	186.5	2080

The actual values for the pressure and velocity at the exit of single nozzle and exit of single nozzle in a dual cluster were calculated by putting a probe at same exit coordinate, which showed that there, occurs a loss of **3 percent** in thrust of single nozzle due to mixing when clustered. Thrust equation was used to calculate the thrust. CFD analysis has been done on Convergent-Divergent nozzles of different arrangements like single and a dual cluster. It has been found that single nozzle gives a velocity of 2234 m/s whereas single nozzle in cluster gives a velocity of 2080 m/s indicating a five percent decrease in velocity.

VI. ACKNOWLEDGMENT

The authors acknowledge the valuable suggestions and help from Dr. Khalid Pervez, and Ma'am Nida Ahsan, Department of Aerospace Engineering, IST Islamabad, Pakistan.

Dept of Aerospace Engg,IIT Kanpur, Mar 15-16, 2000

VII. REFERENCES

- [1]Max kandula and Bruce t Vu , Near-Field Acoustical Characterization of Clustered Rocket Engines, Sierra Lobo, Inc. (USTDC), Kennedy Space Center, FL 32899
- [2]Potter, R.C., and Crocker, M.J., Acoustic prediction methods for rocket engines, including the effects of clustered engines and deflected exhaust flow, NASA CR-566, 1966.
- [3] Pandey K. M., S. A. Khan and Rathakrishnan. National Seminar on *Recent Advances inExperimental*
- [4] Chen, C. L., Chakravarthy, S. L. and Hung, C. . "Numerical investigation of separated Nozzle Flows", AIAA Journal Vol.32, 1836-1843, 1994.
- [5] David C.Wil Cox, "Turbulence modeling for CFD" Second Edition 1998.
- [6] M.M.Atha vale and H.Q. Yang, "Coupled field thermal structural simulations in Micro Valves and Micro channels" CFD Research Corporation.
- [7] Lars Davidson, "An introduction to turbulence Models", Department of thermo and fluid dynamics, Chalmers university of technology, Goteborg, Sweden, November, 2003.
- [8] Kazuhiro Nakahashi, "Navier-Stokes Computations of two and three dimensional cascade flow fields", Vol.5, No.3, May-June 1989.