COMPUTATIONAL ANALYSIS TO ENHANCE THE JET MIXING CHARACTERISTICS USING CIRCULAR GROOVES IN A SQUARE NOZZLE

L.Natrayan [a], M.Prasanth [b], M.Anbalagan [c], R.prabhu [d]

[a,b] *PG Scholar, Engineering Design., Selvam college of Technology, Namakkal, India.*
[c] *Associate Professor, Department of Mechanical Engineering, Selvam college of Technology, Namakkal, India,*
[d] *Principal, Selvam college of Technology, Namakkal, India,*

Abstract

A Computational study has been carried out to understand jet flow development from plain and grooved rectangular nozzles of aspect ratio 2:1. Grooves of square cross section of side 4mm and axial length 5mm were introduced at the exit of the nozzle in three different orientations as (i) minor-axis, (ii) major-axis and, (iii) in both minor and major axes. The computational studies were carried out using computational software ANSYS CFX for a nominal jet exit velocity of 20m/s. Velocity distribution along the axis of the jet is observed from computational results and compared with the available experimental results. Grooves seem to have very negligible effect on the near field region but significantly influence the jet decay in the far field.
Keywords: Passive Control, Jet Mixing Enhancement, Rectangular Jet and Centre Line Velocity Decay

I. Introduction

There are many engineering devices in which jet producing Nozzles form an important component. Circular nozzles are most commonly used but non-circular nozzles are also used primarily due to their desirable mixing characteristics. They are found to exhibit enhanced entrainment of ambient fluid. In view of their favourable qualities many researchers have carried out extensive study of rectangular jets, both experimentally and computationally.

A. NOZZLE:

A Nozzle (from nose, meaning 'small spout') is a tube of varying cross-sectional area (usually axisymmetric) aiming at increasing the speed of an outflow, and controlling its direction and shape. Nozzle flow always generates forces associated to the change in flow momentum, as we can feel by handholding a hose and opening the tap. In the simplest case of a rocket nozzle, relative motion is created by ejecting mass from a chamber backwards through the nozzle, with the reaction forces acting mainly on the opposite chamber wall, with a small contribution from nozzle walls. As important as the propeller is to shaft-engine propulsions, so it is the nozzle to jet propulsion, since it is in the nozzle that thermal energy (or any other kind of high-pressure energy source) transforms into kinetic energy of the exhaust, and its associated linear momentum producing thrust.

The flow in a nozzle is very rapid (and thus adiabatic to a first approximation), and with very little frictional loses (because the flow is nearly one-dimensional, with a favourable pressure gradient except if shock waves form, and nozzles are relatively short), so that the isentropic model all along the nozzle is good enough for preliminary design. The nozzle is said to begin where the chamber diameter begins to decrease (by the way, we assume the nozzle is axisymmetric, i.e. with circular cross-sections, in spite that rectangular cross-sections, said two-dimensional nozzles, are sometimes used, particularly for their ease of direction ability). The meridian nozzle shape is irrelevant with the 1D isentropic model, the flow is only dependent on cross-section area ratios.

B.CONVERGING NOZZLE:

In a converging nozzle, cross-section area smoothly decreases from a larger value (usually assumed a plenum chamber with $M \rightarrow 0$, pc=pt) to a smaller value (exit section Ae, with Me and pe). The mass flow rate in terms of static or total conditions at any stage, with the isentropic relations.

C. COMPUTATIONAL FLUID DYNAMICS:

Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of mathematical modelling (partial differential equations) numerical methods (discretization and solution techniques) software tools (solvers, pre- and post-processing utilities). CFD enables scientists and engineers to perform ‘numerical experiments

Fluid (gas and liquid) flows are governed by partial differential equations which represent conservation laws for the mass, momentum, and energy.

Computational Fluid Dynamics (CFD) is the art of replacing such PDE systems by a set of algebraic equations which can be solved using digital computers.

D. FLUID FLOW:

Fluid flows encountered in everyday life include Meteorological phenomena (rain, wind, hurricanes, floods, fires) Environmental hazards (air pollution, transport of contaminants) Heating, ventilation and air conditioning of buildings, cars etc. Combustion in automobile engines and other propulsion systems interaction of various objects with the surrounding air/water complex flows in furnaces, heat exchangers, chemical reactors.

II. CFD ANALYSIS PROCESS :

Problem statement information about the flow

- Mathematical model IBVP = PDE + IC + BC
- Mesh generation nodes/cells, time instants
- Space discretization coupled ODE/DAE systems
- Time discretization algebraic system Ax = b
- Iterative solver discrete function values
- CFD software implementation, debugging
- Simulation run parameters, stopping criteria

- Post processing visualization, analysis of data
- Verification model validation / adjustment

A. MATHEMATICAL MODEL:

Choose a suitable flow model (viewpoint) and reference frame.

- Identify the forces which cause and influence the fluid motion.
- Define the computational domain in which to solve the problem.
- Formulate conservation laws for the mass, momentum, and energy.
- Simplify the governing equations to reduce the computational effort: use available information about the prevailing flow regime check for symmetries and predominant flow directions (1D/2D) neglect the terms which have little or no influence on the results model.

B. POST PROCESSING AND ANALYSIS:

Post processing of the simulation results is performed in order to extract the desired information from the computed flow field calculation of derived quantities (stream function, vortices)

- Calculation of integral parameters (lift, drag, total mass)
- Visualization (representation of numbers as images)
- 1D data: function values connected by straight lines
- 2D data: streamlines, contour levels, color diagrams
- 3D data: cut lines, cut planes, iso surfaces, iso volumes arrow plots particle tracing, animations.
- Systematic data analysis by means of statistical tools Debugging, verification, and validation of the CFD model

C. UNCERTAINTY AND ERROR:

Whether or not the results of a CFD simulation can be trusted depends on the degree of uncertainty and on the cumulative effect of various errors Uncertainty is defined as a potential deficiency due to the lack of knowledge (turbulence modelling is a classic example) Error is defined as a recognizable deficiency due to other reasons Acknowledged errors have certain mechanisms for identifying, estimating and possibly eliminating or at least alleviating them. Unacknowledged errors have no standard procedures for detecting them and may remain undiscovered causing a lot of harm Local errors refer to solution errors at a single grid point or cell Global errors refer to solution errors over the entire flow domain Local errors contribute to the global error may move throughout the grid.

III. DESIGN

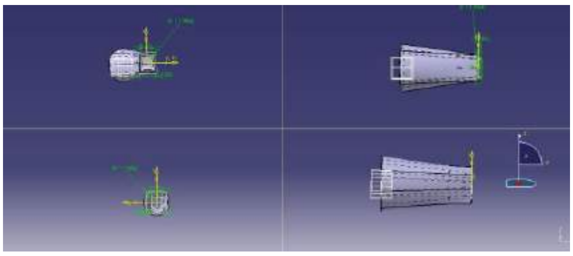

Fig 3-D View of Square Nozzle with Groove 1mm

MESHING:

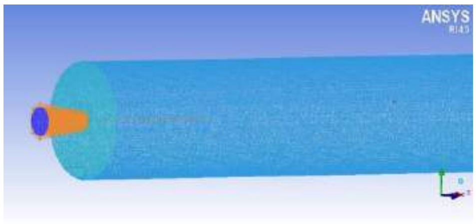

Fig: 1 Finite element model

By importing the IGS file from CATIA, Meshing is carried out using CFD 14.5 and the mesh sizes are given below for the parts. The mesh density is increased in the Nozzle exit area, hence to get the accurate results at the exit area. The Solver selected in ANSYS and ANSYS CFX for further process as follows

Mesh ICEM CFD :

Number of Nodes	:	353319
Number of Elements	:	1872453
Volume	:	$3.02129*10^6$ m^3

Mesh Size :

Domain	:	3
Domain inlet	:	3
Inlet	:	1
Nozzle	:	0.5
Outlet	:	3

CFX Pre-processor:

The Boundary Conditions are given as,

Boundary Name	Condition
Inlet	P = 1.3 bar , T = 300 K
Nozzle	No Ship wall
Domain Inlet	Adiabatic Heat Transfer Wall
Domain Outlet	Total Opening Heat Energy , Pressure = 1 atm,
	Temperature=300K , Turbulence = SST
	Opening Turbulence , Relative Pressure = 1 atm
	Temperature = 300 K

CFX SOLVER:

Using the ANSYS CFX Solver the iterations are carried out to minimum iteration that the system can perform and the iteration find the Continuity, Momentum and Energy Equation for such iterations. When the Solution Converges to required Valve, the solver steps and solution is found and it is said as res file.

CFX POST – PROCESSING:

In the CFX – Post processing application parallel to XY line is created for the nozzle exit to study length. CSV file is exported from the post processing and the results are plotted in graphical format as PCL Comparison. By creating a line parallel to ZX plane in both directions, the result file is exported and Radial plots are plotted.

IV. PCL COMPARISON GRAPH:

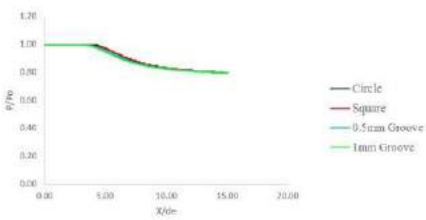

Fig 6.4 PCL Comparison of Nozzle at M = 0.6

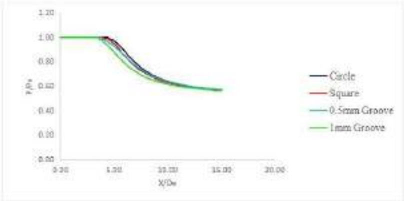

Fig 6.5 PCL Comparison of Nozzle at M = 0.8

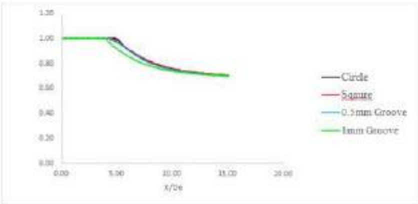

Fig 6.6 PCL Comparison of Nozzle at M = 1.0

By analysing the graphical results, we find that the potential core length is reduced in 1mm grooved nozzle when compared to the other study nozzles. Hence, the effectiveness in calculated to be 21% and hence it is found to produce better mixing characteristics among the study nozzles.

V. THRUST CALCULATION:

Nozzles	M=0.6	M=0.8	M=1.0
Circular Nozzle	2472.69 N	3977.32 N	6567.71 N
Square Nozzle	2331.06 N	3772.14 N	6224.13 N
Square Nozzle with 0.5mm groove	2443.08 N	4219.21 N	6600.29 N
Square Nozzle with 1mm groove	2593.13 N	4475.43 N	6982.23 N

Table: Thrust Calculation

VI. RADIAL COMPARISON FOR NOZZLE :

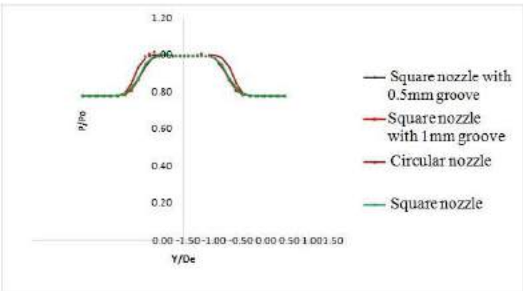

Fig 6.7 Radial Comparison of Nozzle at M = 0.6

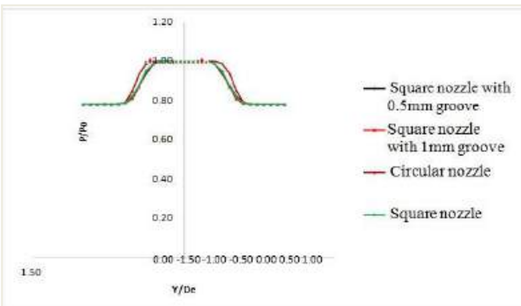

Fig 6.8 Radial Comparison of Nozzle at M = 0.8

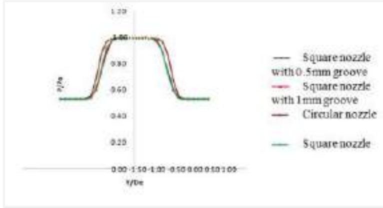

Fig 6.9 Radial Comparison of Nozzle at M = 1

VII. EFFECTIVENESS PLOT :

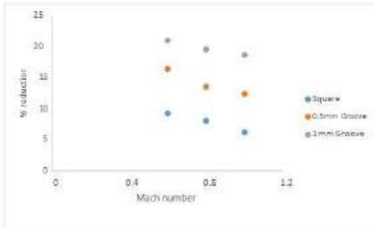

Fig 6.10 Effectiveness Plot

VIII. CONTOUR PLOT:

CIRCULAR NOZZLE:

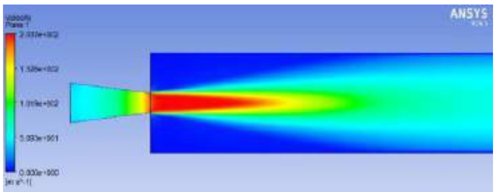

Fig 6.12 Circular Nozzle (M = 0.8)

The jet doesn't spread in the circular nozzle, the core length and the velocity decay is longer.

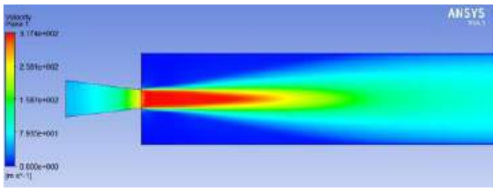

Fig 6.13 Circular Nozzle (M = 1)

The jet from the exit of the nozzle doesn't spread as of grooved nozzle since there is no edge to produce mixing which increases jet spread.

SQUARE NOZZLE:

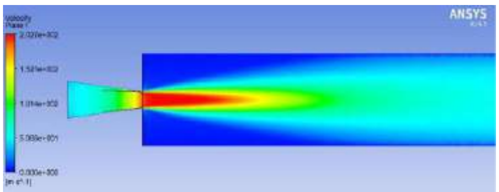

Fig 6.14 Square Nozzle (M = 0.6)

The jet mixing is greater when compared to circular nozzle, the velocity is reduced and the centre line velocity is less.

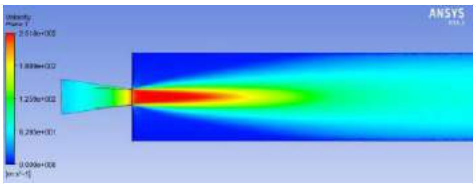

Fig 6.15 Square Nozzle (M = 0.8)

The centre line velocity increases as the match number increases and the jet spread is smaller when compared to nozzle with grooves.

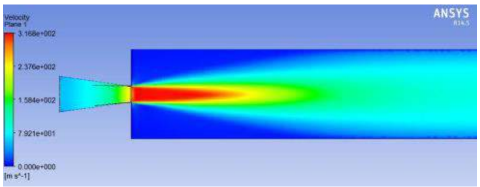

Fig 6.16 Square Nozzle (M = 1)

SQUARE NOZZLE WITH 0.5MM GROOVE:

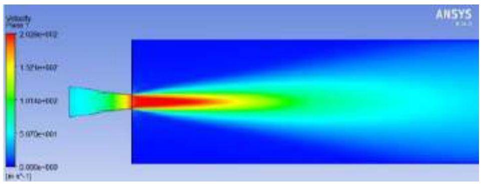

Fig 6.17 Square Nozzle with 0.5 mm Groove (M = 0.6)

The Jet spread is minimum in this flow, the centre line velocity is less and the velocity decays in a shorter length in M=0.6.

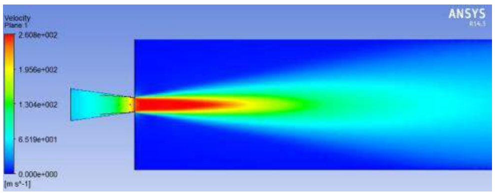

Fig 6.18 Square Nozzle with 0.5 mm Groove (M = 0.8)

The jet spread is comparatively less and the centre line velocity is more when compared to M=0.6.The velocity decay length is comparatively long when compared to M=0.6.

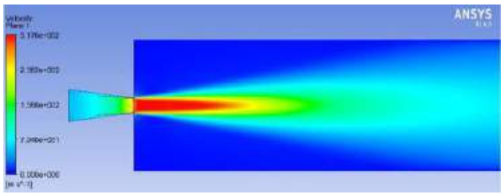

Fig 6.19 Square Nozzle with 0.5 mm Groove (M = 1)

The centre line velocity is increased when compared to M=0.6 and M=0.8. The jet does not spread as much in other match numbers.

SQUARE NOZZLE WITH 1MM GROOVE:

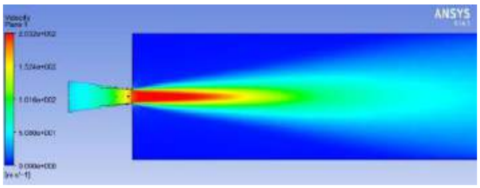

Fig 6.20 Square Nozzle with 1 mm Groove (M = 0.6)

The Jet spread is minimum in this flow, the centre line velocity is less and the velocity decays in a shorter length in M=0.6. The velocity decay is larger when compared to square nozzle with 0.5mm groove.

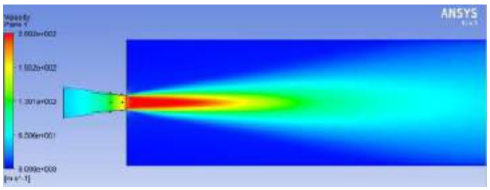

Fig 6.21 Square Nozzle with 1 mm Groove (M = 0.8)

The jet spread is comparatively less and the centre line velocity is more when compared to M=0.6.The velocity decay length is comparatively long when compared to M=0.6.The velocity at the nozzle exit is less when compared with other nozzles.

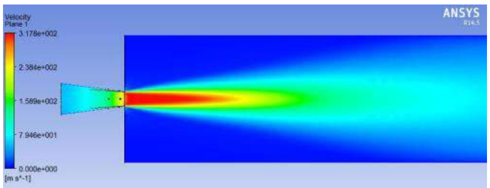

Fig 6.22 Square Nozzle with 1 mm Groove (M = 1)

The centre line velocity is increased when compared to M=0.6 and M=0.8. the jet does not spread as much in other match numbers.

IX. Conclusion:

By Analysing the Result from the contour plots, the jet spread is maximum in the 1mm Grooved Nozzle, It also has the elastic reduction in its potential core length, when compared to other study nozzles.it is much effective in the velocity m=0.6, the core velocity or centre core is much reduced in the 1mm grooved nozzle and hence we can get fine mixing characteristics and hence the efficiency of the mixing is higher.

X. Reference :

1.Naveenkumar S and Radhakrishnan E. (2002) 'Sonic Jet Control with Tabs, International Journal of Turbo and Jet Engines, Vol. 19, Nos. 1-2, pp. 107-118.

2. Norum T.D. (1983) 'Screech Suppression in Jets', AIAA Journal, Vol. 21, pp. 235-240.

3.Pannu S.S. and Johannesen N.H. (1976) 'The Structure of the Jets from Notched Nozzles',Journal of Fluid Mechanics,Vol. 74, Part 3, pp. 515-528.

4.Papamoschou D. (1990) ‘Commnication Paths in the Compressible Shear Layers’, AIAA Paper 90-0155.

5.Papamoschou D. (1991) ‘Structure of the Compressible Turbulent Shear Layers’, AIAA Journal, Vol. 29, No. 5, pp. 680-681.

6.Papamoschou D. and Debiasi M. (1999) ‘Noise Measurements inJets Treated with the Mach Wave Elimination Method’, AIAA Journal,Vol. 37, No. 2, pp. 154-160.

7.Papamoschu D. and Roshko A. (1988) ‘The Compressible Turbulent Shear Layer: An Experimental Study’, Journal of Fluid Mechanics, Vol. 197, pp. 453-477.

8. Parviz Behrouzi and James J.M. (2006) ‘Effect of Tab Parameters on Near-Field Jet Plume Development’, Journal of Propulsion and Power,Vol. 22, No. 3, pp. 576-585.

9. Povinelli L.A. and Ehlers R.C. (1972) ‘Swirling Base Injection for Combustion Ramjets’, AIAA Journal, Vol. 10, No. 9,pp. 1243-1244.

10. Powell A. (1953a) ‘On the Mechanism of Choked Jet Noise’, Proceedings of Physical Society of London, Vol. 66, pp. 1039-1056.