[Kumaraswamy*, 2.(9): September, 2015]
ISSN 2349-6193
Impact Factor (PIF): 2.243

IJESMR

International Journal OF Engineering Sciences & Management Research

COMPUTATIONAL ANALYSIS OF SUPERSONIC JETS FROM RECTANGULAR NOZZLES

Kumaraswamy T[*1], Santhiya C[2], Dr. M. Satyanarayana Gupta[3]
[*1] Assistant Professor, Department of Aerospace, M.L.R. Institute of Technology, Hyderabad.
[2]Research Asst and Student, Department of Aerospace, M.L.R. Institute of Technology, Hyderabad.
[3]HOD and Professor, Aero Department, M.L.R. Institute of Technology, Hyderabad.

Keywords: Rectangular Nozzle, Supersonic jet flow, Non-uniformity flow.

ABSTRACT

A prominent field which is drawing much attention in recent times is the mixing characteristics of supersonic jet flows because of its wide ranging applications in Aero-engines. Unlike in subsonic flows, vanes or baffles could not be used for creating mixing in a supersonic flow. In the present study, the supersonic jet flow from a nozzle with rectangular cross section is analyzed using CFD Techniques. The primary cause of the non-uniform flow exiting the nozzle is due to cross-flow pressure gradients imposed on wall boundary layers. The cross-flow pressure gradients cause the boundary layer to roll up into counter-rotating vortex pairs on each of the four sides of the nozzle. These four vortex pairs produce significant non-uniformity in the nozzle exit flow. The nozzle is designed using CATIA design software and the computational analysis is done using ANSYS CFX software for different aspect ratios.

INTRODUCTION

Rectangular jets have been gaining importance owing to their desirable mixing, entrainment and acoustic characteristics compared to circular jets. They find applications in combustors and propulsive devices of high-speed systems. Rectangular jets in a row are extensively used for various industrial applications. In aerospace engineering, the mixing of jets is used for variety of engineering applications, ranging from thrust augmenting equipment to fuel mixing in combustion processes. The examples are thrust augmenting ejectors for VTOL / STOL aircraft.

In Scramjet engines combustion takes place at supersonic speed. For efficient combustion to happen the fuel should mix completely with the air. The main problem of scramjet fuel injection is that the airflow is quite fast, meaning that there is minimal time for the fuel to mix with the air and ignite to produce thrust. Enhancing the mixing, and thus reducing the combustor length, is an important aspect in designing scramjet engines. Hence jet exit at the fuel injector should reach the atmospheric pressure quickly. Thus the rectangular cross section fuel injector can be used to for efficient mixing. Also as the shock strength is less, the noise will also be less. The problem of noise suppression of turbojet engines has shown a need for turbulence data within the flow field of various types of nozzles. So, the rectangular jets are used to give improved propulsion and reduced noise level. The problem of noise suppression of turbojet engines has shown a need for turbulence data within the flow field of various types of nozzles. So, the rectangular jets are used to give improved propulsion and reduced noise level. Relatively complete information of the jet properties in the far field is available, but most of the mixing processes of technical interest take place in the near field. Although the number of investigations in this part of the flow has increased remarkably in the last few years, nevertheless no reliable theoretical prediction scheme is available today to describe the mixing of rectangular jets in the near field satisfactorily.

Recent interest in rectangular supersonic jets is motivated by the need to reduce plume length and acoustically excited structural loads in the exhaust systems of high performance aircrafts. Semi-periodic shock structures form in the jet plumes of high performance vehicles during low speed flight when the convergent divergent nozzles operate at off design conditions. These shock cells affect the jet velocity and temperature decay as well as the jet spread rate and its acoustic field. The present work investigates the jet from CD nozzles with rectangular exit geometry with different aspect ratios. The study focuses on the external regime of jet flows and determines how aspect ratio of a rectangular cross section influences the jet mixing and jet spread characteristics.

PROPERTIES OF SUPERSONIC RECTANGULAR JETS

Turbulent jets issuing from rectangular nozzles are free shear flows driven by the momentum introduced at the exit. As soon as the jet leaves the nozzle, the jet will start entraining the ambient fluid. The momentum will be exchanged with ambient fluid thus the jet will spread. Further downstream the mixing region will be wider penetrating the centerline after that the centerline velocity will decay and the turbulence intensities will grow. Large vortical structures are formed in the shear layer between the jet and its surrounding. These vortices carry out the irrotational ambient fluid and hence induce mixing. As soon as the vortices move downstream, they will grow, become larger and interact with each other. As these vortices will move further downstream, secondary instabilities will break them up.

The structure of non-axisymmetric incompressible rectangular jet shows that the flow-field may be subdivided into three main regions:

- The potential core region, where the center line velocity, Uc is constant,
- The two-dimensional transition region.
- The region extending to infinity, where the center line velocity decay is characteristics of axisymmetric jets.

Non-axisymmetric nozzles are generally well-known to exhibit the peculiar phenomenon ofaxis-switching. It is understood to be due to the effect of non-uniform vortices distribution along the edges of the non-axisymmetric cross section at the nozzle exit. It has been widely observed for nozzles with elliptic, square, triangular and rectangular cross-sections.

Plane or two dimensional jets (i.e. the jet entrains fluid only in axial and lateral directions), are produced from high aspect ratio nozzles. Their velocity fields are two dimensional. It was stated that aspect ratio must be larger than 50 to obtain two dimensional flow fields without any assistance from the sidewalls, whereas in rectangular or three dimensional jets, smaller aspect ratios are used. In these jets, the velocity fields will be axisymmetric after the two dimensional region and the length of the later region was found to be a function of the aspect ratio. The velocity flow fields of a rectangular jet are shown in figure 1.

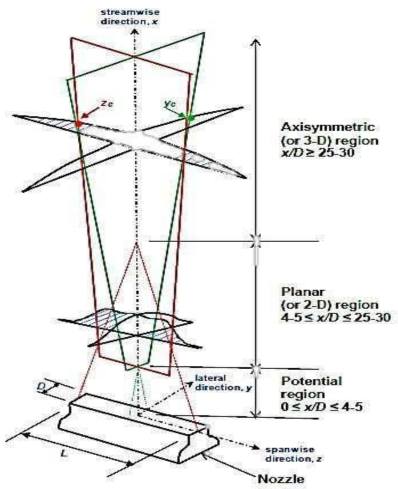


Fig. 1. Demonstration of a subsonic jet

AXIS SWITCHING

Axis switching is a phenomenon in which the cross-section of an asymmetric jet evolves in such a manner that, after a certain distance from the nozzle, the major and the minor axes are interchanged. The spreading rates of the jet column issuing from non-axisymmetric nozzles are not equal in all directions. This result in the eventual development of the jet being aligned such that, at some point downstream of the nozzle, it appears that the jet column has through a certain angle while spreading. It is usually termed as axis-switching since the orientation of the axes as seen along the nozzle exit cross-section appears to turn through an angle as it evolves downstream. Axis-switching is thus defined as “the phenomenon in which the cross-section of an asymmetric jet evolves in such a manner that, after a certain distance from the nozzle, the major and minor axes are interchanged”. An example for the axis- switching sequence depicted through the deformations of non-axisymmetric vortex rings is shown in Figure 2.

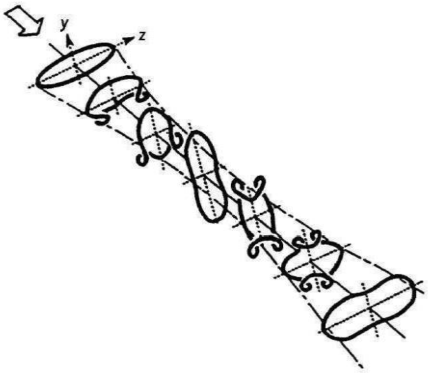


Fig. 2 an example for the axis-switching sequence depicted through the deformations of non-axisymmetric vortex rings

FACTORS AFFECTING RECTANGULAR SUPERSONIC JETS

REYNOLDS’S NUMBER

Experimental investigations have shown that the various characteristics of a jet (e.g. growth and decay rates, mass and momentum flow rates, turbulent flow fields) depend on Reynolds number. For jet issuing from a rectangular nozzle, the jet exit Reynolds number, Re is defined using the nozzle opening width D (minor dimension of the nozzle, Figure 1), centreline mean velocity at the exit, U0 and kinematic viscosity, v of the tested fluid using the expression

D = U0D/v

ASPECT RATIO

The aspect ratio, AR of a jet issuing from a rectangular nozzle is the ratio of the long side of the nozzle, L to the short side of the nozzle, D

AR=L/D

Aspect ratio plays a very important role in determining the characteristics of a jet issuing from rectangular nozzle. Therefore, much effort was given to study the influence of this parameter.

OBJECTIVE OF PRESENT STUDY

As mentioned earlier geometry of nozzle, Reynolds’s number, aspect ratio, etc. have a great influence in jet issuing from a nozzle. Apart from circular jets recent studies have been concentrated more on non-circular jets. Various studies on elliptical, triangular, hexagonal, cruciform, square and rectangular jets have been undergone since time. In all the cases it was found non-circular nozzles to be beneficial in any of the respect than circular one. The detailed research in the non-circular jets has been performed in the past two decades, because largely due to their potential to entrain ambient fluid more effectively than comparable circular jets. Variations in rectangular exits are now experimented on nozzles. Four aspect ratios have been taken into consideration and nozzles for each aspect ratio have been designed using CATIA V5. The same have been analyzed using Ansys CFX. The total pressure decay along the centerline of jet will be more concentrated since mixing is enhanced more with rapid total pressure decay along centerline. In addition to that the potential core length for each nozzle will be evaluated. Analysis will be carried out in two different conditions; under expanded and over expanded conditions. This project is aimed at investigating the influence of aspect ratio in rectangular jets for the respects mentioned above.

SCOPE OF THE PROJECT

The rectangular nozzle will probably soon be accepted as one of the most commonly preferred component of a propulsion system for both aircraft and spacecraft. Besides, its ease of manufacture and maintenance makes it an economical choice compared to the circular nozzle of similar capabilities. The increased current need to understand the properties of the rectangular nozzle is primarily due to the wide range of applications that are now being realized in the aerospace design sector. All the factors affecting the rectangular free jet mentioned above have been studied with varying degree of success over the last three-four decades. The exit aspect ratio of the nozzle is known to affect axis-switching, which is one of the properties of rectangular jets. The turbulence intensity also affects the spread of the jet; the higher the turbulence intensity, the greater the spreading of the jet. For supersonic jets, the temperature and scale of the nozzle affects screech production which has a further effect on jet development. Amongst these, the least understood is probably the dependence of the jet development on the internal geometry of the nozzle. The focus of this study is thus to understand and investigate further the effects of the aspect ratio of rectangular nozzle geometry that seem to have a significant influence on the development of the jet downstream of the exit. This effect has not yet been qualitatively analyzed nor quantified regarding its impact on the supersonic jet development.

LITERATURE SURVEY

Ponnambalam Manivannan and Banbla Tharaka Narendra Sridhar, 1993 "Characteristic Study of Non-Circular Incompressible Free Jet" states the non-circular jets have shorter potential core length compared to the circular jet. It is due to the effect of the nozzle shapes having corners and flat sides. Within the non-circular shapes, the cruciform jet core length was shorter because the numbers of corners are more than the hexagonal shape. Coherent structures could be generated at the flat sections of nozzle. The characteristic decay region, herein the axis velocity decay is dependent upon orifice configuration, and velocity profiles in the plane of the minor axis are found to be similar whereas those in the plane of the major axis are non-similar.

Edward J. Rice, Ganesh Raman, 1993 "Supersonic Jets From Beveled Rectangular Nozzle Related Work" states Beveled geometries provided screech noise reduction for under-expanded jets and an upstream mixing noise directivity shift for over-expanded jets. Two approaches to improving the performance of shear flow control device:

i) The first is to increase the mixing rate of the jets to move the jet noise source back toward the nozzle lip and thus provide a longer propagation length for an acoustic lining to reduce the internal mixing noise
ii) The second is to cause the directivity of the internally generated mixing noise to be more normal to the acoustic treatment surface which would make the suppressor much more effective.

A. Mohamed, A. Hamed, T. Lehnig, 2010 "Supersonic Rectangular Over-Expanded Jets Of Single And Two-Phase Flows" detailed experimental investigation to study supersonic jets from convergent divergent nozzles with rectangular cross section The rectangular supersonic jet spread rate is greater along the minor axis and increases with the nozzle pressure ratio. In two-phase rectangular jets of gas and dispersed solid particles the shock strength was found to attenuate with increased particle loading. The shock strength as well as the total number of shock cells in the jet plume decrease as the nozzle pressure ratio decreases. The visual jet spread rate along the minor axis is seen to decrease as the nozzle pressure ratio decreases. The visual jet width in the major plane does not exhibit noticeable spread, nor does it change with the nozzle pressure ratio. The particle loading increases, the shock strength as well as the number of shock cells decreases.

DESIGN OF A SUPERSONIC RECTANGUALR NOZZLE

DESIGN PARAMETERS & DESIGN SPECIFICATIONS

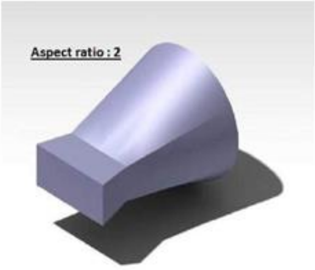
For designing the nozzle, the parameters like exit area is assumed and fixed as per the literature survey conducted. Mach no is also fixed as 1.8.
Mach no = 1.8
Nozzle exit area = 108 mm2
By using the area- Mach number relation, the other parameters and dimensions are found out as given below:

$$\left(\frac{A}{A^*}\right)^2=\frac{1}{M^2}\left(\frac{2}{\gamma+1}+\frac{\gamma-1}{\gamma+1}M^2\right)^{\frac{\gamma+1}{\gamma-1}}$$

Throttle area = 75 mm^2
Inlet area = 308.13 mm^2
Inlet to throat length =20 mm

Throat to exit length = 6.5 mm

DESIGN OF SUPERSONIC RECTANGULAR NOZZLE WITH DIFFERENT ASPECT RATIO



AR = 2

Inlet area = 308.13mm^2

Throat Dimensions:

Throat area = 75mm^2

Length =12.2474mm

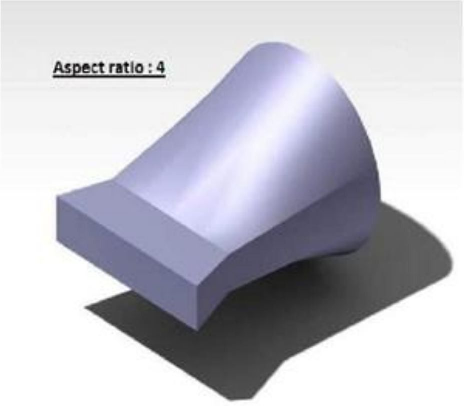
Breadth=6.1237mm

Exit dimensions:

Exit area = 108mm^2

Length = 14.6968mm

Breadth = 7.3484mm



AR = 4

Inlet area = 308.13mm2

Throat Dimensions:

Throat area = 75mm2

Length =17.32mm

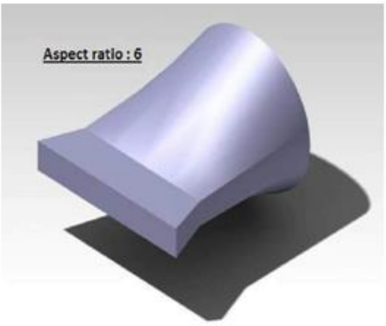
Breadth=4.33mm

Exit dimensions:

Exit area = 108mm2

Length = 20.7846mm

Breadth=5.1962mm



AR = 6

Inlet area = 308.13mm2

Throat Dimensions:

Throat area = 75mm2

Length =21.213mm

Breadth=3.536mm

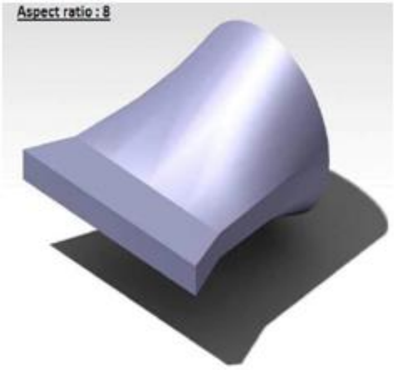
Exit dimensions:

Exit area = 108mm2

Length = 25.46mm

Breadth=6.4545mm

Breadth = 4.24mm



AR = 8

Inlet area = 308.13mm

Throat Dimensions:

Throat area = 75mm2

Length=24.5mm

Breadth=3.062mm

Exit dimensions:

Exit area = 108mm2

Length= 29.392mm

Breadth=3.674mm

Fig. 3. Isometric view of (a) AR 2 (b) AR 4 (c) AR 6 (d) AR 8 Nozzle with design parameters

COMPUTATIONAL ANALYSIS OF RECTANGULAR SUPERSONIC NOZZLE JETS

ANSYS CFX software is a high-performance, general purpose fluid dynamics program that has been applied to solve wide-ranging fluid flow problems for over 20 years. ANSYS CFX is more than just a powerful CFD code. Integration into the ANSYS Workbench platform provides superior bi-directional connections to all major CAD systems, powerful geometry modification and creation tools with ANSYS Design Modeler, advanced meshing technologies in ANSYS Meshing, and easy drag-and-drop transfer of data and results to share between applications. For example, a fluid flow solution can be used in the definition of a boundary load of a subsequent structural mechanics simulation. A native two-way connection to ANSYS structural mechanics products allows capture of even the most complex fluid–structure interaction (FSI) problems in the same easy-to-use environment, saving the need to purchase, administer or run third-party coupling software. For more than 20 years, companies around the world have trusted ANSYS CFX technology to provide reliable and powerful computational fluid dynamics (CFD) solutions. ANSYS CFX combines advanced solver technology with a modern user interface and an adaptive architecture to make CFD accessible to both designers with general engineering knowledge and fluid dynamics specialists requiring in-depth model control and options. It is used in a vast array of industries to provide detailed insight into equipment and processes that increase efficiency, improve product longevity and optimize processes. The CFD analysis is based on the basic aerodynamic equations like energy equation, momentum equation and continuity equation.

ANSYS CFX software is fully integrated in to the ANSYS Workbench environment, the framework for the full suite of engineering simulation solutions from ANSYS. Its adaptive architecture enables users to easily set up anything from standard fluid flow analyses to complex interacting systems with simple drag-and-drop operations. Users can easily assess performance at multiple design points or compare several alternative designs. Within the ANSYS Workbench environment, applications from multiple simulation disciplines can access tools common to all, such as geometry and meshing tools. The step by step procedure undertaken in CFX is explained and demonstrated in detail below.

GEOMETRY & MESHING

ANSYS Design Modeler software is specifically designed for the creation and preparation of geometry for simulation. It’s easy-to-use, fully parametric environment with direct, bidirectional links to all leading CAD packages acts as the geometry portal for all ANSYS products to provide a consistent geometry source for all engineering simulations.

For the analysis purpose a cylindrical domain is to be constructed at the exit of the nozzle with length and diameter approximately 50 and 20 times that of the equivalent exit diameter. The domain is the expected region of exit flow influence. The domain must be large enough to accommodate the complete exit flow from nozzle. Here we have almost taken 100mm dia and 500mm length for the domain. The domain was first designed using CATIA and saved the file in .iges format. It was then simply imported to the ANSYS workbench. After this, the point of influence was defined and it was given for almost 200 points on the surface of the domain. The blue arrows denote the pressure points on the surface of the domain and black arrows depict the inlet pressure points.

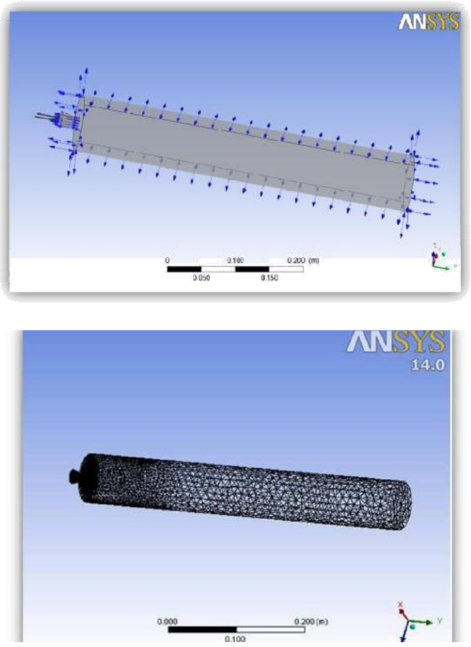


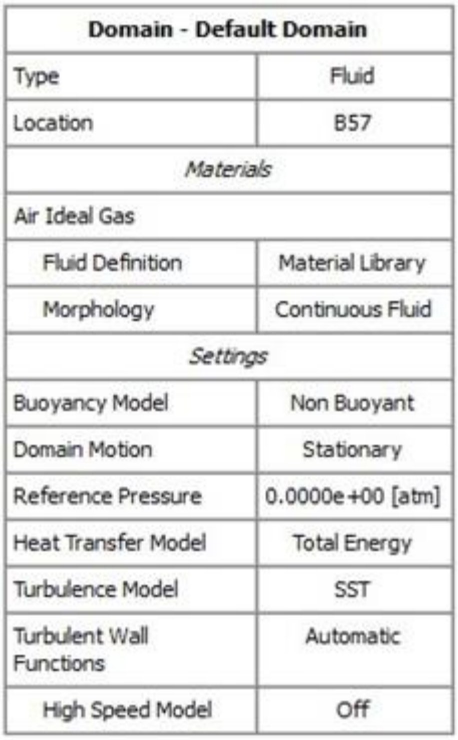
Fig. 4. (a) Boundary conditions defined for pre-meshing (b) Meshed geometry

For acquiring accurate CFD results, superior meshing technology is required. ANSYS Meshing provides a multitude of meshing technologies in a single application to allow users to select the best option on a part by part basis. This ANSYS CFX includes unlimited mesh editing capabilities as well as structure hexahedral meshing. In order to know whether the data entered was feasible or not, a simple coarse meshing was initially done. Once the solution was found to converge, then we move on to fine meshing. For this purpose we chose global co-ordinate system and about five co-ordinate systems was selected along center line to accomplish center line meshing. Each sphere of 15mm radius equally spaced with centers at 22.5mm apart was formed. Also the default values were changed according to our requirements. The type of mesh was selected as sphere of influence, also for each co-ordinate system, body size (whole body considered as a single unit) meshing was defined with elemental sizing 0.006 mm. After it the whole procedure was saved and we moved on to the next stage of analysis.

BOUNDARY CONDITION

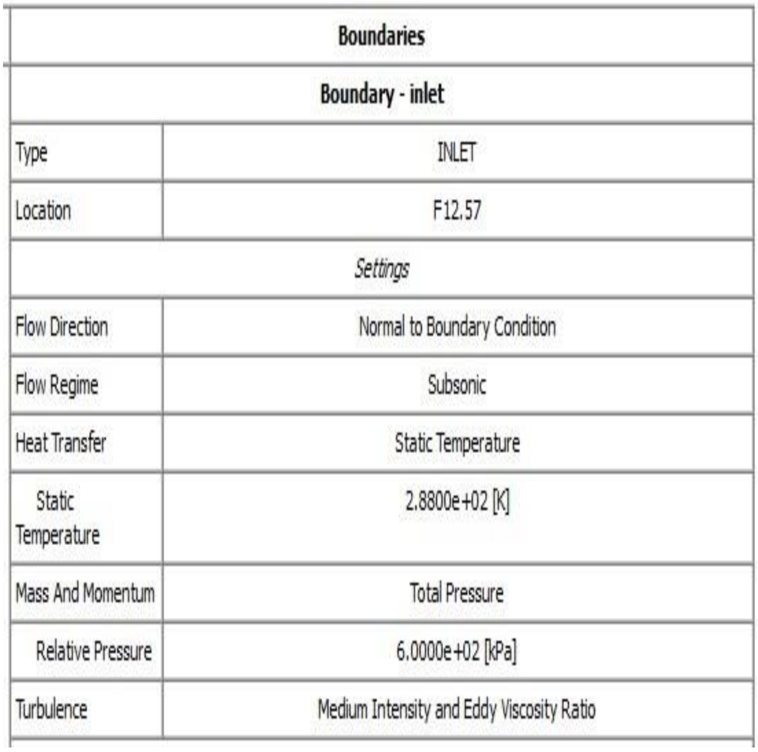
The ANSYS CFX physics pre-processor is a modern and intuitive interface for the setup of CFD analyses. In addition to a general mode of operation, predefined wizards are available to guide users through the setup of common fluid flow simulations. A powerful expression language gives users the ability to customize their problem definition in numerous ways, such as with complex boundary conditions, proprietary material models or additional transport equations. The adaptive architecture of CFX-Pre even allows users to create their own custom GUI panels to standardize input for selected applications, and thereby ensure adherence to established best practices. In this context as mentioned earlier the boundary conditions are defined. The material was selected as ideal gas and the initial pressure conditions were assigned to 0 atm. The basic domain physics chart is given aside. The boundary conditions were all changed to our accordance and this is briefed in the below given tables.

IJESMR
International Journal OF Engineering Sciences & Management Research



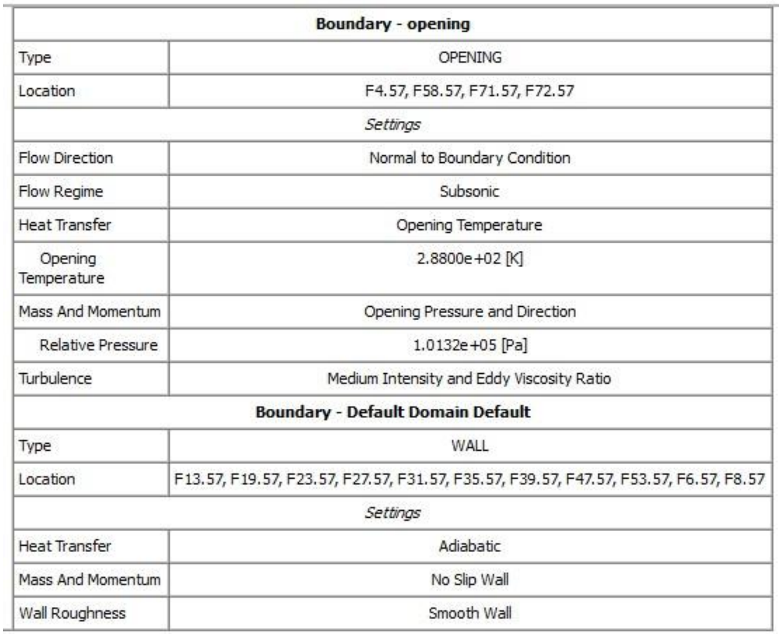
Domain - Default Domain	
Type	Fluid
Location	B57
Materials	
Air Ideal Gas	
Fluid Definition	Material Library
Morphology	Continuous Fluid
Settings	
Buoyancy Model	Non Buoyant
Domain Motion	Stationary
Reference Pressure	0.0000e+00 [atm]
Heat Transfer Model	Total Energy
Turbulence Model	SST
Turbulent Wall Functions	Automatic
High Speed Model	Off

Table 1: Domain physics for CFX



Boundaries	
Boundary - inlet	
Type	INLET
Location	F12.57
Settings	
Flow Direction	Normal to Boundary Condition
Flow Regime	Subsonic
Heat Transfer	Static Temperature
Static Temperature	2.8800e+02 [K]
Mass And Momentum	Total Pressure
Relative Pressure	6.0000e+02 [kPa]
Turbulence	Medium Intensity and Eddy Viscosity Ratio

Table 2: Inlet Boundary conditions



Boundary - opening	
Type	OPENING
Location	F4.57, F58.57, F71.57, F72.57
Settings	
Flow Direction	Normal to Boundary Condition
Flow Regime	Subsonic
Heat Transfer	Opening Temperature
Opening Temperature	2.8800e+02 [K]
Mass And Momentum	Opening Pressure and Direction
Relative Pressure	1.0132e+05 [Pa]
Turbulence	Medium Intensity and Eddy Viscosity Ratio
Boundary - Default Domain Default	
Type	WALL
Location	F13.57, F19.57, F23.57, F27.57, F31.57, F35.57, F39.57, F47.57, F53.57, F6.57, F8.57
Settings	
Heat Transfer	Adiabatic
Mass And Momentum	No Slip Wall
Wall Roughness	Smooth Wall

Table 3: Outlet and Domain Boundary Conditions

SOLUTION

At the heart of ANSYS CFX software is its advanced solver technology using coupled algebraic multigrid, the key to achieving reliable and accurate solutions quickly and robustly. Its engineered scalability ensures a linear increase in CPU time with problem size and parallel performance that is second to none. Users can follow convergence progress and dynamically monitor numerical and physical solution quantities. Solver parameters, boundary conditions and other parameters can be adjusted on the fly, without stopping the solver. The ANSYS CFX solver uses second order numeric by default, ensuring users always gets the most accurate predictions possible. For almost 1000 iterations we give the solver to solve the context, but the result is extracted most probably when the solution converges. The solution was taken on four grounds and based on this the result was extracted.

RESULTS AND DISCUSSION

As discussed earlier turbulent jets issuing from rectangular nozzles are free shear flows driven by the momentum introduced at the exit. As soon as the jet leaves the nozzle, the jet will start entraining the ambient fluid. The momentum will be exchanged with ambient fluid thus the jet will spread. Further downstream the mixing region will be wider penetrating the centerline after that the centerline velocity will decay and the turbulence intensities will grow. The jet mixing and spreading rates can be thus accessed via the centerline total pressure decay and the length of potential core.

Here the four properties along the centerline are taken into consideration. They are the center line total pressure decay, center line static pressure, center line density gradient and the potential core lengths. Each property considered is separately discussed for each aspect ratio.

Also the Circular Nozzle has been compared with the Rectangular Nozzle of AR=2 with center line total pressure decay, density gradient, static pressure ratio and potential core lengths.

COMPARISION OF RECTANGULAR NOZZLE OF VARIOUS ASPECT RATIOS (AR=2, 4, 6 & 8)

CENTERLINE TOTAL PRESSURE DECAY AND DENSITY GRADIENT OF SUPERSONIC RECTANGULAR NOZZLE OF VARIOUS AR’s:

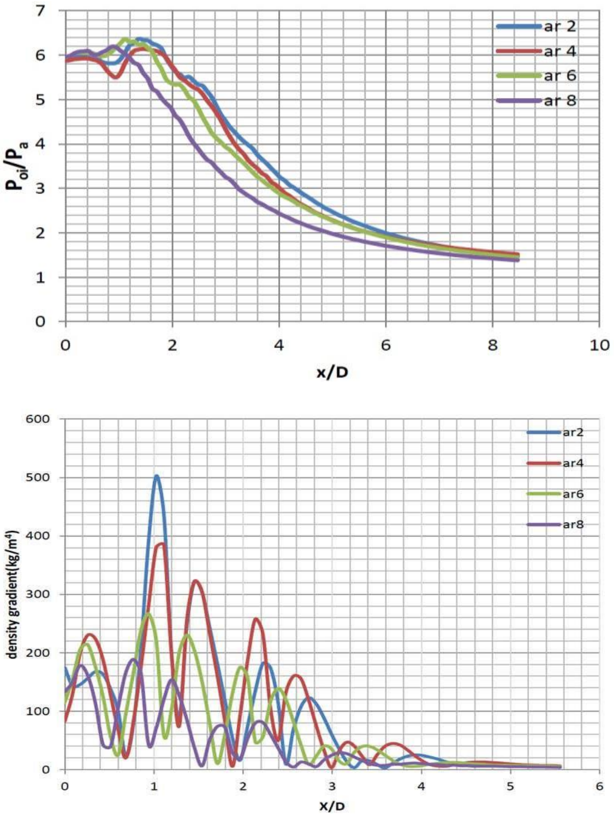


Fig. 5. (a) Comparison for centerline total pressure decay for rectangular jet with different aspect ratios (Poi=600KPa, M=1.8) (b) Comparison of density gradient for four aspect ratios

Total pressure decay characteristics have been discussed with a plot of pressure ratio and non-dimensional distance from the nozzle exit. For examining we are specifying some points and defining the pressure property for the same. On examining the plot for a particular pressure ratio, say for 4; for aspect ratio 8 the value have been attained more rapidly and for aspect ratios 6, 4,2, it is 2.82, 3.07, 3.24. Hence from this it can be noted that the center line total pressure ratios are lagging the former at a rate of 21.42%, 28.568%, and 39.01% respectively.

The density gradient has been evaluated by plotting density against non-dimensional distance from the exit. Here the peak values are considered first. For aspect ratio 2 the maximum value obtained is 502.733. For aspect ratios 4, 6, 8 the peak values are 280.01, 266.66 and 188.02 respectively which is almost 75.73%, 53.05% and 37.5% respectively of the former value. From the plot it is clear that, only at the very near field of the jet exit a density gradient has occurred. As the distance from exit increases, the density gradient for the four aspect ratios becomes almost constant.

The x/d value at which the density gradient becomes constant is 2.56, 2.73, 2.99, and 3.33 respectively for aspect ratio 8, 6, 4 and 2 respectively. The same can be visualized in the figure depicted above in which the density gradient variation for aspect ratio 2, 4, 6 and 8 are given respectively.

STATIC PRESSURE AND TOTAL PRESSURE OF SUPERSONIC RECTANGULAR NOZZLE OF VARIOUS AR's:

For examining the static pressure variations a plot with pressure ratio and distance is taken. In a near distance range of the jet exit, aspect ratio 2 has achieved the atmospheric pressure first at about x/D =1.28. Much nearer to it aspect ratios 4, 6 and 8 have achieved atmospheric pressure at, x/D=1.45, 1.62, 1.7. It can be observed that the rate of static pressure increment is 1.13%, 1.26% and 1.33% respectively for aspect ratio 4, 6, 8.

On observing the below figure it is clear that for aspect ratio 2 the potential core length is the greatest. Potential core length decreases as aspect ratio increases, and it decreases with a value of about 11.11% for each aspect ratio. For aspect ratios 4, 6, 8 the potential core length decreases in the rate of 11.11%, 22.22%, 33.33%

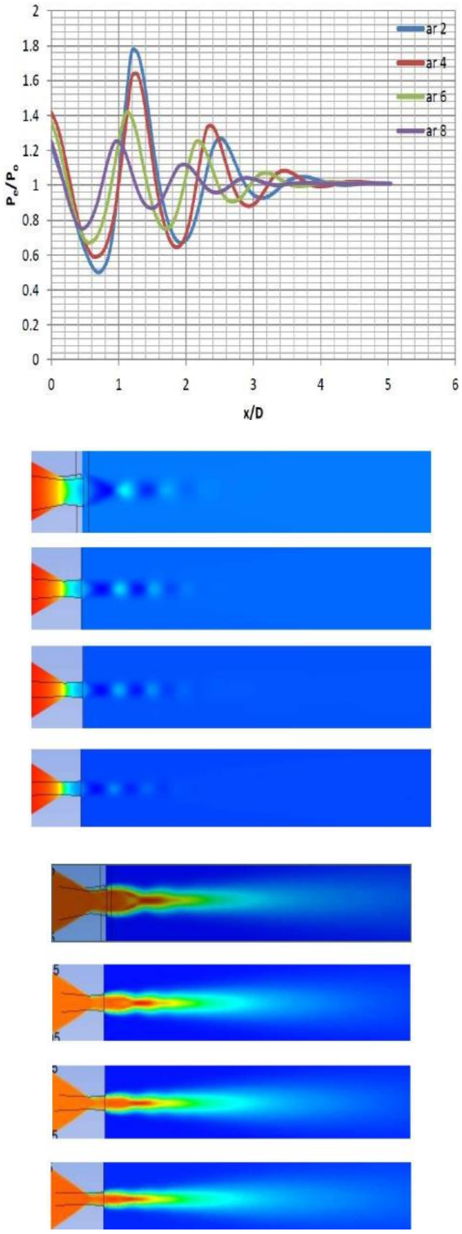


Fig. 6. (a) Comparison of static pressure for various aspect ratios (b) Variation in static pressure for aspect ratio 2, 4, 6, 8 respectively (c) Total pressure distribution for various aspect ratios

COMPARISION OF RECTANGULAR NOZZLE OF AR 2 TO THE CERCULAR NOZZLE

CENTERLINE TOTAL PRESSURE DECAY & STATIC PRESSURE VARIATION

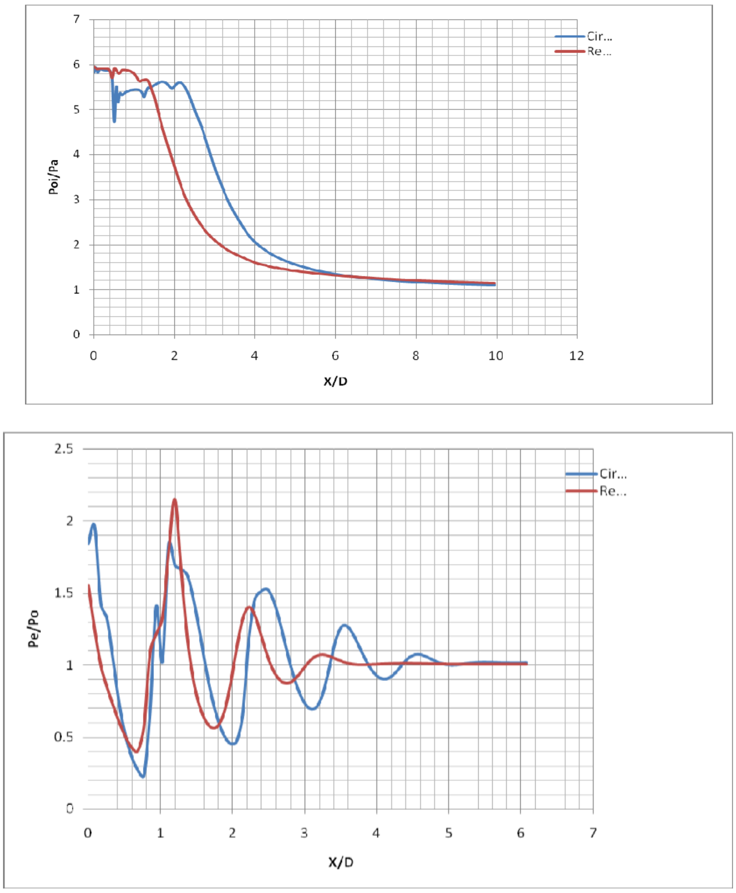


Fig. 7. (a) Comparison for center line total pressure decay for rectangular jet with of Aspect Ratio 2 with Circular Nozzle (Poi=600KPa, M=1.8)
(b) Comparison of static pressure for Rectangle Nozzle of Aspect Ratio 2 with Circular Nozzle

Total pressure decay characteristics have been discussed with a plot of pressure ratio and non-dimensional distance from the nozzle exit. For examining we are specifying some points and defining the pressure property for the same. On examining the plot for a particular pressure ratio, say for 4; for rectangular nozzle the pressure ratio have been attained more rapidly and for circular it occurs at later. Hence from this it can be noted that the center line total pressure ratios of circular nozzle is lagging behind the rectangular nozzle.

For examining the static pressure variations a plot with pressure ratio and distance is taken. In a near distance range of the jet exit, aspect ratio 2 have achieved the atmospheric pressure earlier than the circular nozzle.

DENSITY GRADIENT OF CIRCULAR NOZZLE Vs RECTANGULAR NOZZLE OF AR=2

The density gradient has been evaluated by plotting density against non-dimensional distance from the exit. Here the peak values are considered first. For aspect ratio 2 the maximum value obtained is 103.689. For circular nozzle the peak values is 386.656. From the plot it is clear that, only at the very near field of the jet exit a density gradient has occurred. As the distance from exit increases, the density gradient for the circular nozzle becomes almost constant before rectangular nozzle.

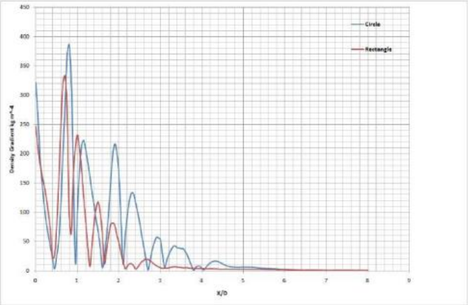


Fig. 8. Comparison of density gradient for Rectangle Nozzle of Aspect Ratio 2 with CircularNozzle

POTENTIAL CORE LENGTH OF CIRCULAR NOZZLE Vs RECTANGULAR NOZZLE OF AR=2

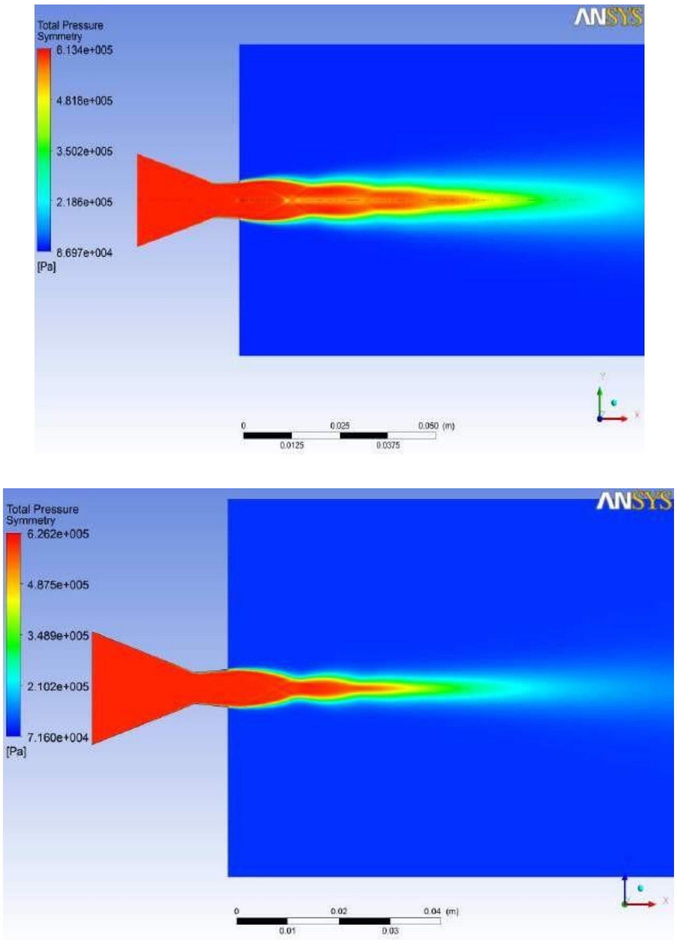


Fig. 9. (a) Total pressure distribution for Circular Nozzle (b) Total pressure distribution for Rectangular Nozzle of Aspect Ratio 2

On observing the above figs 9(a) and 9(b), it is clear that for circular nozzle the potential core length is the greatest. Potential core length is less for the Rectangular nozzle

CONCLUSION

Mixing and jet spread are the two major concerns of supersonic flow. As in subsonic flow, turbulence cannot be created in a supersonic flow by means of bluff bodies or anything else. It would have adverse effect on the design at supersonic conditions. So we prefer to have a design which can geometrically induce turbulence. Turbulence can be found anywhere at the exit. For simplicity we are considering the property along the centerline. Physically turbulence means total pressure decay; here it is centerline total pressure decay. So as rapid as the center line total pressure decay occurs, the more efficient the design will be.

On comparing the total pressure for various aspect ratios it is observed that potential core length is largest for Aspect Ratio AR=2 and least for AR=8. As potential core length decreases, rate of total pressure decay increases. So the rate of total pressure decay is rapid for AR=8 and decreases in the order AR=6, AR=4, and AR=2.

While comparing the static pressure for different aspect ratios, AR=2 has the maximum peak value and this peak value decreases with increase in aspect ratio. So the atmospheric pressure is attained by AR=8 at the earliest. The static pressure increases due to the shock strength increase, so AR=2 has the highest shock strength.

Density gradient comparison plots indicates that the shock strength is maximum for AR=2 and minimum for AR=8. The peak value of maximum density gradient is attained by AR=8. As the aspect ratio of rectangular nozzle increases, the static pressure and the rate of total pressure decay increases resulting in decreasing shock strength which in turn increases jet mixing and reduce noise

REFERENCES

1. Richard L. Gaffney, Jr., John J. Korte “Analysis and Design of Rectangular Cross Section nozzles for Scram jet Engine Testing” 42nd AIAA Aerospace Sciences Meeting and Exhibit. 2004.
2. Strykowski, P.J., and Niccum, D.L., “The Stability of Countercurrent Mixing Layers in Circular Jets,” Journal of Fluid Mechanics, Vol. 227, 1991, pp. 309- 343.
3. Seiner J. M., Dash S. M., and Kenzakowski D.C., “Historical survey on enhanced mixing in scramjet engines,” AIAA Journal of Propulsion and Power, Vol. 17, pp 1273-1286, 2001.
4. K.M.Pandey, T.Sivasakthivel: Recent Advances in Scramjet Fuel Injection – A Review. International Journal of Chemical Engineering and Applications, Vol. 1, No. 4, 2010
5. Seung-Jae Hwang: Numerical Simulation of Enhanced Mixing in Scramjet Combustor Using Ramp, Tabs and Suction Collar. Thesis, University of Kansas School of Engineering 2011.
6. Ashraf Elsaid Mohamed Ibrahim, “An Experimental Investigation of Supersonic Rectangular Over-Expanded Nozzle of Single and Two-Phase Flows”, 2008, University of Cincinnati, pp. 2008 – 2109
7. By K. B. M. Q. ZAMAN, “Axis switching and spreading of an asymmetric jet: the role of coherent structure dynamics”, 1994.
8. Ephraim J. Gutmark, Irene M. Ibrahim, and Shanmugam Murugappan, “Circular and noncircular subsonic jets in cross flow”, American Institute of Physics, Physics in Fluids, 2008.
9. K.B.M.Q. Zaman, “Axis Switching and Spreading of an Asymmetric Jet- Role of Vorticity Dynamics”, NASA Technical Memorandum, 33rd Aerospace Sciences Meeting and Exhibit, 1995.
10. Valerio Viti, Reece Neel, and Joseph A. Schetz, “Detailed flow physics of the supersonic jet interaction flow field” American Institute of Physics, Physics in Fluids, 2008
11. A.A. Mostafa, M.M. Khalifa , E.A. Shabana, “Experimental and numerical investigation of multiple rectangular jets” Journey of Experimental Thermal and Fluid Science, 2000, Vol.
12. T.S. Cheng, K.S. Lee, “Numerical simulations of under expanded supersonic jet and free shear layer using WENO schemes” International Journal of Heat and Fluid Flow 26, 2005, pp. 755-770
13. C. M. Or1, K. M. Lam and P. Liu, “Potential Core Lengths of Round Jets in Stagnant and Moving Environments” Journal of Hydro-Environment Research 2nd Revision: December 2010
14. Edward J. Rice and Ganesh Raman; “Supersonic Jets From Beveled Rectangular Nozzles”, Symposium on Flow Acoustics Interaction and Fluid Control, 1993
15. Rathakrishnan. E, “Corrugated Tabs for Supersonic Jet Control”; 10th International Conference On Fluid Control, Measurements And Visualization, 2009