ACTA TECHNICA CORVINIENSIS – Bulletin of Engineering
Tome VI (Year 2013) – FASCICULE 3 [July–September]
ISSN 2067–3809

[1.] G. SATYANARAYANA, [2.] Ch. VARUN, [3.] S.S. NAIDU

CFD ANALYSIS OF CONVERGENT-DIVERGENT NOZZLE

[1-3.] DEPARTMENT OF MECHANICAL ENGINEERING, MVGRCE, VIZIANAGARAM, 535005, (A.P.), INDIA

ABSTRACT: CFD is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. In this thesis, CFD analysis of flow within Convergent-Divergent supersonic nozzle of different cross sections rectangular, square and circular has been performed. The analysis has been performed according to the shape of the supersonic nozzle and keeping the same input conditions. Our objective is to investigate the best suited nozzle which gives high exit velocity among the different cross sections considered. The application of these nozzles is mainly in torpedos. The work is carried out in two stages: 1.Modeling and analysis of flow for supersonic nozzles of different cross sections.2.Prediction of best suited nozzle among the nozzles considered. In this, initially modeling of the nozzles has been done in CATIA and later on mesh generation and analysis have been carried out in ANSYS FLUENT 12.0 and various contours like velocity, pressure, temperature have been taken and their variation according to different nozzles has been studied. Compared to square and circular nozzles, rectangular nozzle gives an increased velocity of about 23.93% and 24.47% respectively and an increased pressure drop of 22.93% and 23.97% respectively and an increased temperature drop of 42.56% and 43.68% respectively. It is found that fluid properties like velocity, pressure and temperature are largely dependent on the cross section of the nozzle which affects the flow within the nozzle and the extent of flow expansion.
KEYWORDS: convergent divergent nozzle

INTRODUCTION

Advances in computing technology, software and hardware have revolutionized the design process of engineering vehicles such as aircrafts, automobiles and ships. Many commercial software packages are being used in the design as well as analysis processes which not only save the lead time and costs of new designs, but also are used to study systems where controlled experiments are difficult or impossible to perform. In the area of fluid dynamics, there are many commercial Computational Fluid Dynamics (CFD) packages available for modeling flow in or around objects. Computational Fluid Dynamics (CFD) has been constantly developed over the past few decades and now both commercial and research codes can provide more and more robust and accurate results. Combined with the use of wind tunnel test data, CFD can be used in the design process to drive geometry changed instead of being used mainly as a design validation tool. Computational Fluid Dynamics (CFD) has become an integral part of the engineering design and analysis environment of many companies that require the availability to predict performance of new designs or processes before they are ever manufactured or implemented.

One of the most critical requirements for any CFD tool used for thermal applications is the ability to simulate flows along nozzles, turbines. Such challenging features has pressure gradients, shocks, velocity distribution, eddy location, stream line curvature, and stream wise vortices pose a challenge for computing. The small margins of improvement that are usually targeted in nozzle and turbines design today require precise tools capable of discerning small differences between alternate designs. Custom modeling tools that are based as simplified numerical methods and assumptions cannot provide accuracy that can be obtained with CFD, which offers mainly inherent advantages for ex: it offers quick and cheap solution and comparison to experimental solution and more accurate in comparison to empirical methods used in design. Accurate simulation of flows through the nozzle is important for prediction of velocity pattern and pressure pattern.

The current study aims analysis of flow through the nozzle and prediction of optimal axial clearance. Solution of flow along the nozzle involves only one phase of gas. Results are verified with the experimental data. As a part of project work nozzle studies carried out and with using same nozzle, axial gap / (Clearance determination) is analyzed. The results are in good agreement with the experimental ones.

MODELING OF THE COMPONENTS

Modeling the Supersonic Nozzle:

The coordinates are provided in Table 1, Table 2 for development of the 3D model of the supersonic *nozzle. For design purpose, the nozzle can be seen as an assembly of 3 separate sections operating in series a converging section, a throat and finally the diverging section. Different models of nozzles can be observed in Figure 1 to Figure 5.*

Table 1: coordinates for end points of nozzle profile

Sl. No.	X (mm)	Y (mm)
1	4.000	0.000
2	5.000	1.000
3	5.000	2.854
4	6.810	9.191
5	14.639	17.129
6	25.915	23.387
7	37.497	27.252
8	54.685	32.100
9	78.997	32.100
10	63.282	28.688
11	38.482	23.303
12	29.197	19.638
13	23.628	15.216
14	21.553	11.526
15	20.525	5.000
16	20.525	1.000
17	21.525	1.000

Table 2: coordinates for arc centers

CENTERS	X (mm)	Y (mm)	RADIUS (mm)
C1	4.000	1.000	1.000
C2	17.000	2.854	12.000
C3	31.531	-7.360	29.750
C4	48.045	-29.780	57.589
C5	56.123	-47.842	77.370
C6	-47.118	-12.171	36.510
C7	39.334	1.158	21.079
C8	30.695	8.812	9.536
C9	50.985	2.284	38.556
C10	21.525	1.000	1.000

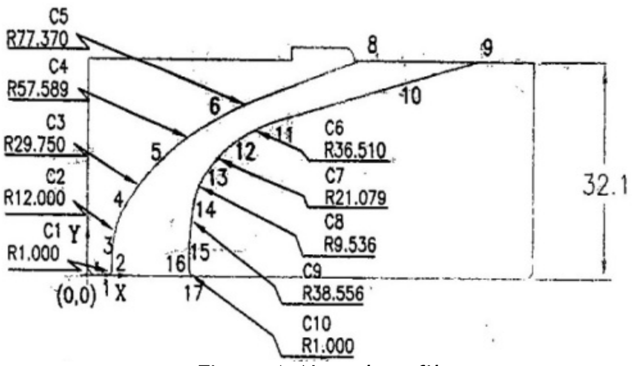

Figure 1: Nozzel profile

Figure 2: Geometry of rectangular bent nozzle

Figure 3: Geometry of rectangular straight nozzle

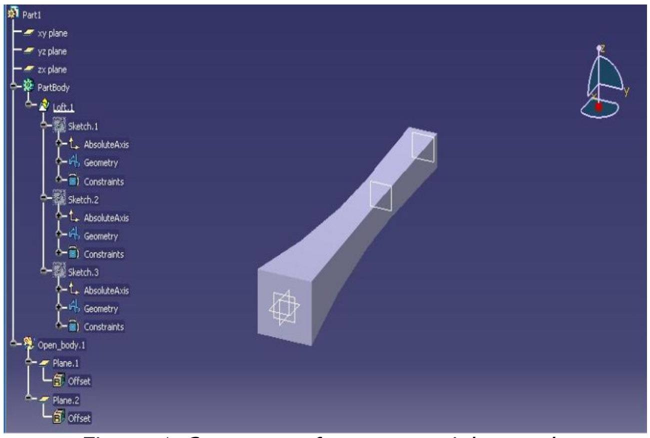

Figure 4: Geometry of square straight nozzle

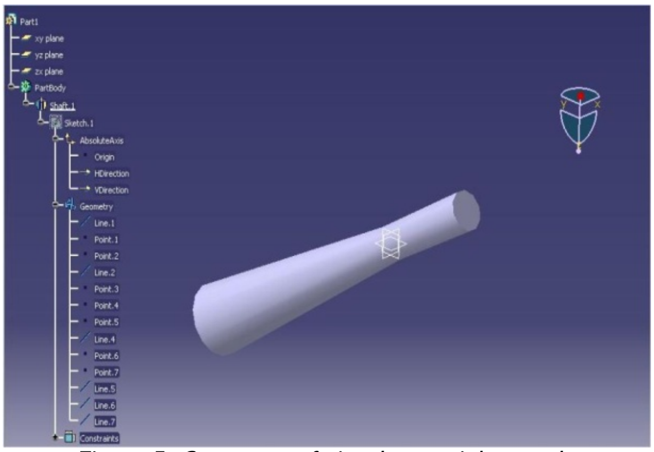

Figure 5: Geometry of circular straight nozzle

Conversion of nozzle geometry:

Taking the rectangular bent nozzle as reference and keeping the inlet, throat and exit areas and the axis length constant, three dimensional rectangular, square and circular straight nozzles geometries were generated.

Analysis of C-D nozzle:

The analysis and meshing is carried out in ANSYS FLEUNT software importing the conditions and the boundary values for the problem statement.

Meshing:

ANSYS FLUENT software is opened and the design file of C-D nozzle saved in CATIA is imported. And then the meshing of nozzle is done with inflation control is set to "Program Controlled" and the INLET, OUTLET and WALL named selections are defined and the meshing is updated.

Following this the scaling is done. Scale is set to mm. Grid created was changed to mm.

SOLVER AND MATERIAL SELECTION AND OPERATING CONDITION DEFINING

The solver is defined first. Solver is taken as Couple based and formulation as implicit, space 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 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*
- *C_p (Specific heat capacity) = 2034.6 J/Kg.K*
- *Thermal Conductivity = 0.0706 W/m.K*
- *Viscosity = 6.07 e-5 Kg/m.s*
- *Molecular Weight = 23.05 Kg/K.mol*

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

BOUNDARY CONDITIONS

Nozzle Inlet:

Pressure inlet was taken as inlet for nozzle. The value of pressure is 8101325 Pascal. Initial gauge pressure was taken as 7898681 Pascal. Temperature was taken as 1583 K.

Nozzle Outlet:

The nozzle outlet is set as pressure outlet with a value of 13e5.

Controls Setup:

The solution controls are set as listed below:

- *Turbulence Kinetic Energy = 0.8*
- *Turbulence Dissipation rate = 0.8*
- *Turbulence Viscosity = 1*

The under relaxation factor was given as given.

Discretization Equation is selected as given:

- *Flow (Second order up wind)*
- *Turbulence Kinetic Energy (1^{st} order upwind)*
- *Turbulence Dissipations rate (1^{st} order upwind)*

Initialization:

Solution initialization is done. Initial value of velocity is taken as 186.3 m/s.

Temperature is taken as 1583 K.

Residual monitoring is done and convergence criteria are set up. The convergence criteria of various parameters are listed below.

Continuity	-	*0.001*
X Velocity	-	*0.001*
Y Velocity	-	*0.001*
Z Velocity	-	*0.001*

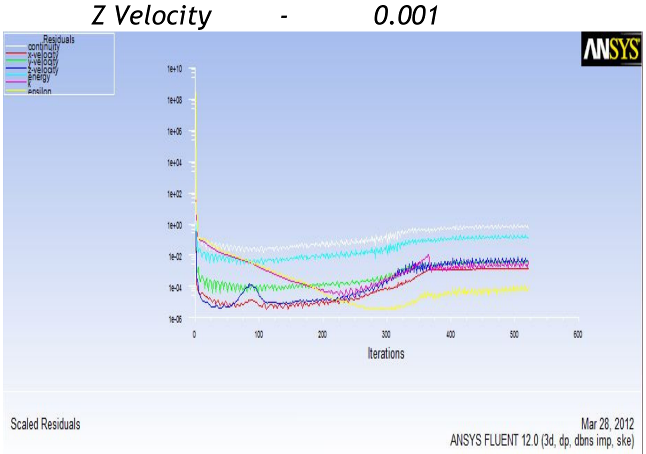

Figure 6: Convergence history for rectangular bent nozzle

The number of iterations is set up and iterations start.

The iterations continue till the convergence is reached and convergence history as shown in Figure 6 to Figure 9.

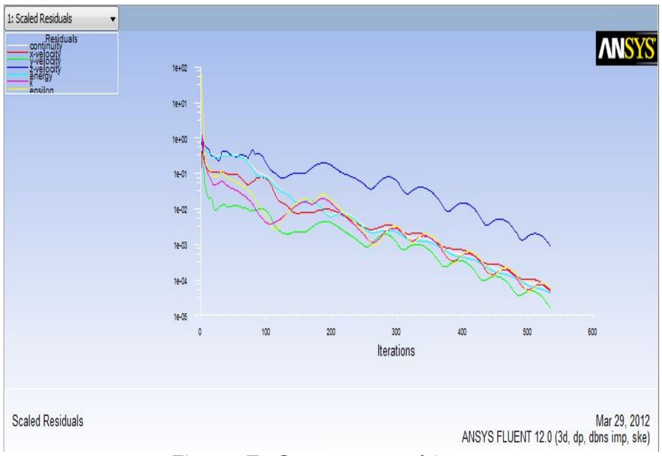

Figure 7: Convergence history for rectangular straight nozzle

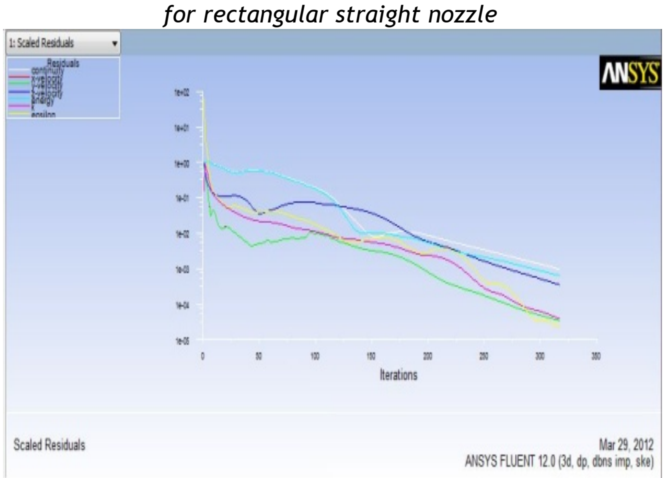

Figure 8: Convergence history for square straight nozzle

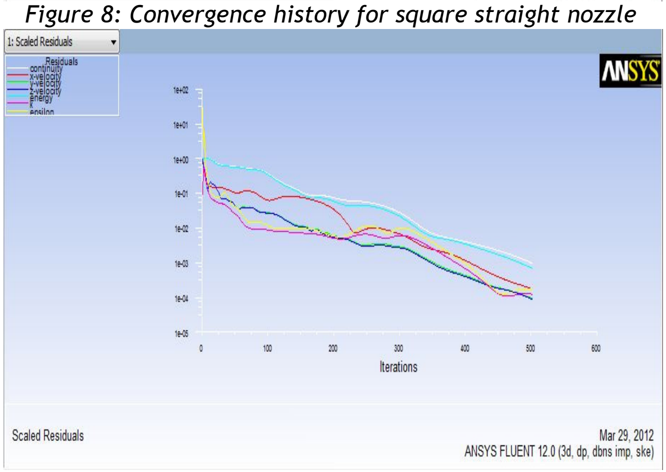

Figure 9: Convergence history for circular straight nozzle

THEORETICAL CALCULATIONS OF C-D NOZZLE

Properties of Gases:

- *GAMA of gases (γ)=1.27*
- *Thermal Conductivity [K]=0.0706 W/m.K*
- *Molecular Weight=23.05 Kg/K.mol*
- *Specific heat of gases (C_p)=2034.6 J/Kg.K*
- *Density treated as Ideal gas*
- *Gas constant (R) =432*
- *Viscosity=6.07e-5 Kg/m.s*

Boundary (Inlet/Exit) Conditions:

- *NPR (Nozzle Pressure Ratio) =6.16*
- *Inlet pressure= 80 bar*

- *Exit pressure= 12.987 bar*
- *Inlet temperature=1583 K*

Geometrical Parameters:
Throat area =2.85e-5 m^2
Given Inlet Mach number= 0.2
Mach number definition: $M=V/C$ 1(a)
C is acoustic velocity

$$C=\sqrt{\gamma RT} \qquad 1(b)$$

$$C=\sqrt{(1.27)(432)(1583)}=931.931$$

Inlet Velocity is $\frac{V}{931.931}=0.2$

$$V = 186.3 \text{ m/sec}$$

Throat Pressure $\frac{P_T}{P_I}=\left(\frac{2}{n+1}\right)n/(n-1)$ 1(c)

$P_T = 44.096$ bar

Velocity at throat: $P=\rho RT$ 1(d)

$$80\times10^5=\rho\,(432)\,(1583)$$

$$\rho = 11.698 \text{ kg/m}^3$$

$$V_T=\sqrt{\frac{2n}{n-1}P_I V\left(1-\left(\frac{P_T}{P_I}\right)^{\frac{n-1}{n}}\right)}= \qquad 1(e)$$

$$=\sqrt{\frac{2\times1.27}{0.27}\cdot\frac{80\times10^5}{11.698}\left(1-\left(\frac{44.096}{80}\right)^{\frac{0.27}{1.27}}\right)}=874.78m/sec$$

Velocity at exit:

$$V_E=\sqrt{\frac{2n}{n-1}P_I V_I\left(1-\left(\frac{P_E}{P_I}\right)^{\frac{n-1}{n}}\right)}= \qquad 1(f)$$

$$=\sqrt{\frac{2\times1.27}{0.27}\cdot\frac{80\times10^5}{11.698}\left(1-\left(\frac{1}{6.16}\right)^{\frac{0.27}{1.27}}\right)}=1436\ m/sec$$

So by the above results Exit Mach number is

$$M=\frac{V_e}{C}=\frac{1436}{931.931}=1.54$$

Known relation, Mach number related with other parameters,

$$\frac{A_E}{A_T}=\frac{1}{M_{Exit}}\left[\frac{1+\frac{(\gamma-1)}{2}M_{Exit}^{\ 2}}{\frac{(\gamma+1)}{2}}\right]^{\frac{(\gamma+1)}{2(\gamma-1)}} \qquad 1\ (g)$$

$$\frac{A_E}{A_T}=\frac{1}{1.54}\left[\frac{1+\frac{0.27}{2}(1.54)^2}{\frac{2.27}{2}}\right]^{\frac{1.27}{2(0.27)}}$$

$$\frac{A_E}{A_T}=1.255$$

So $A_{EXIT}=0.3495\times10^{-4}\,m^2$; $A_{THROAT}=0.285\times10^{-4}\,m^2$

RESULTS AND DISCUSSION

Results for nozzle:

Nozzle profile which is examined is considered in 3D. Flow through the nozzle for given input condition with velocity as 183.6 m/s and maximum output was observed as 1436 m/s such that Mach number is increased from 0.2 to 1.54 in which nozzle is acting as supersonic nozzle. The velocity contours of nozzle are plotted in Figure 10 to Figure 12. The pressure contours of nozzle is plotted in Figure 14 to Figure 17 and temperature contours of nozzle is plotted in Figure 18 to Figure 21. The velocity, temperature, Mach number and pressure variation along the nozzle is compared with theoretical calculation and with experimental too.

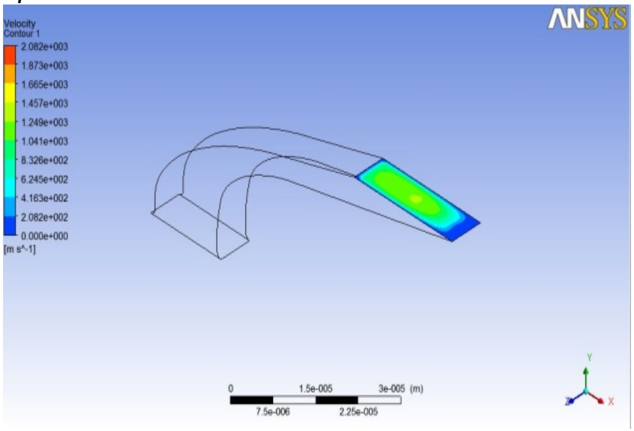

Figure 10: Velocity contour of rectangular bent nozzle

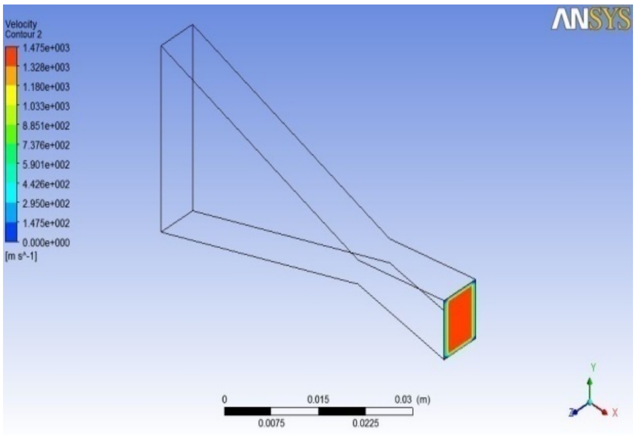

Figure 11: Velocity contour of rectangular straight nozzle

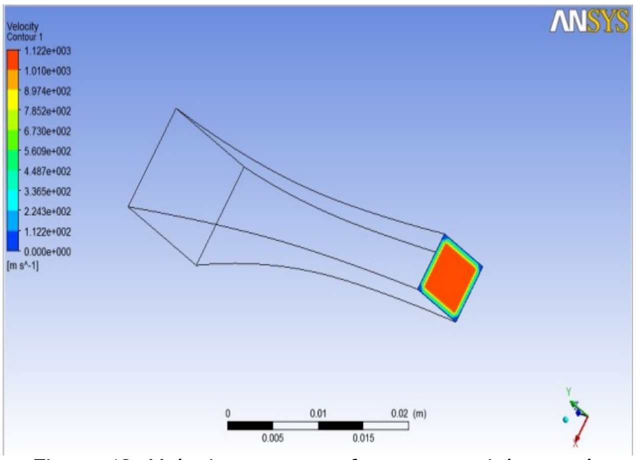

Figure 12: Velocity contour of square straight nozzle

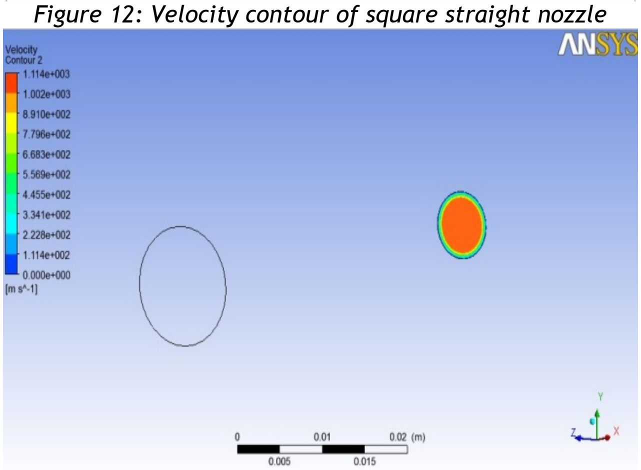

Figure 13: Velocity contour of circular straight nozzle

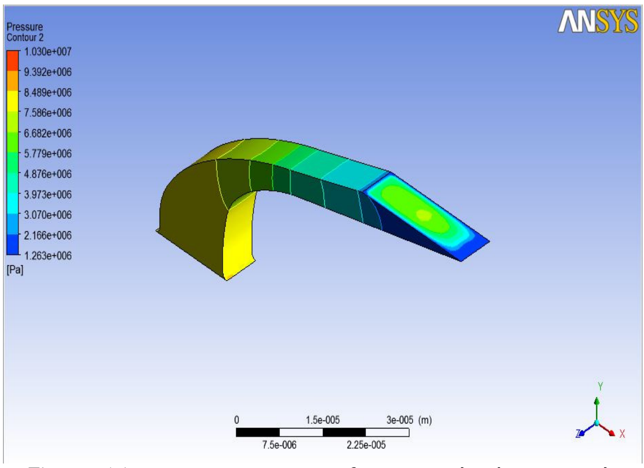

Figure 14: pressure contour of rectangular bent nozzle

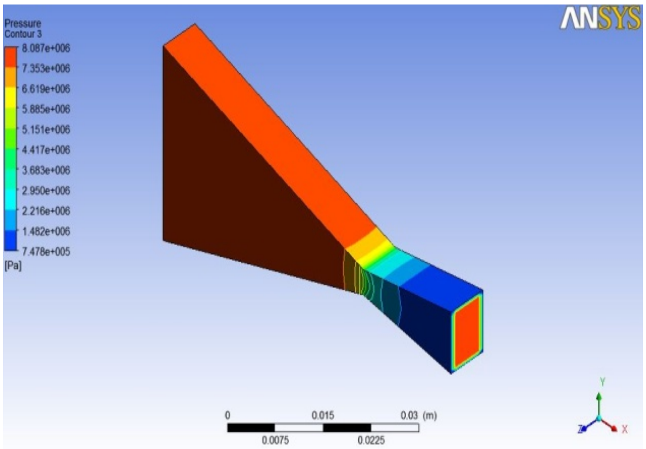

Figure 15: pressure contour of rectangular straight nozzle

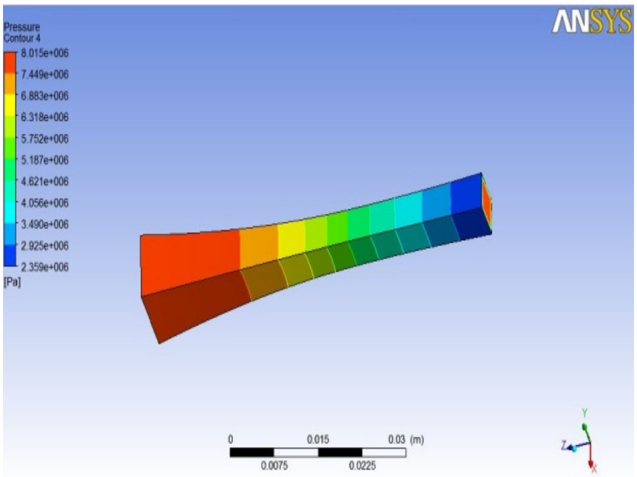

Figure 16: pressure contour of square straight nozzle

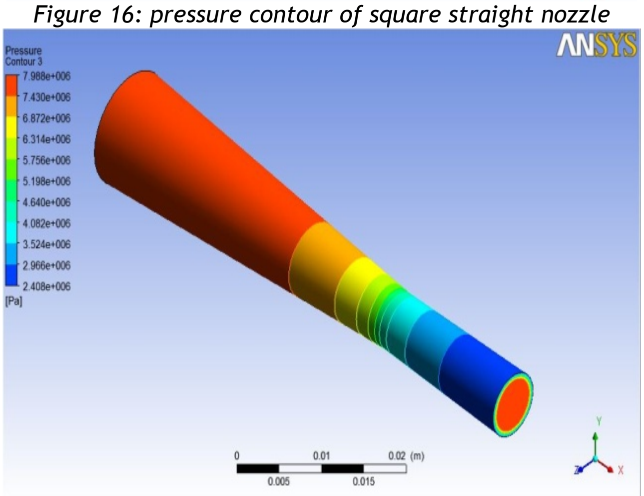

Figure 17: pressure contour of circular straight nozzle

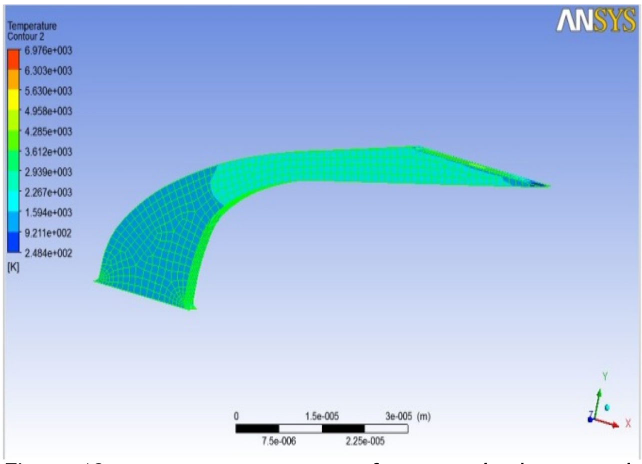

Figure 18: temperature contour of rectangular bent nozzle

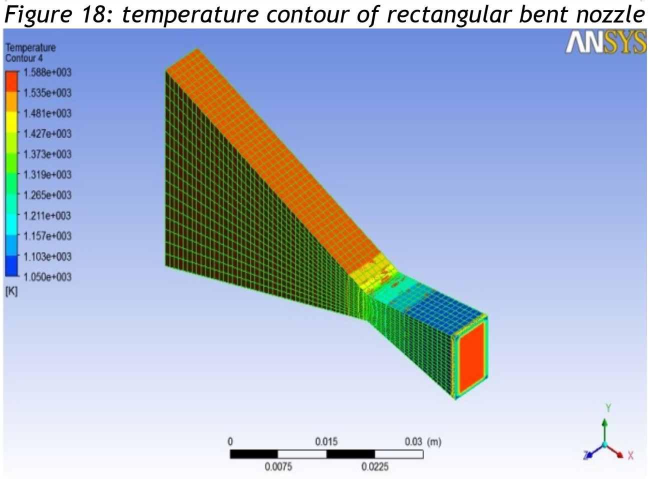

Figure 3.19: temperature contour
of rectangular straight nozzle

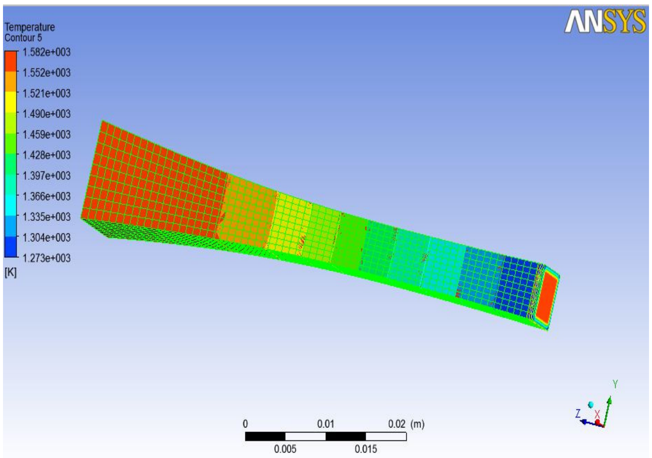

Figure 3.20: temperature contour of square straight nozzle

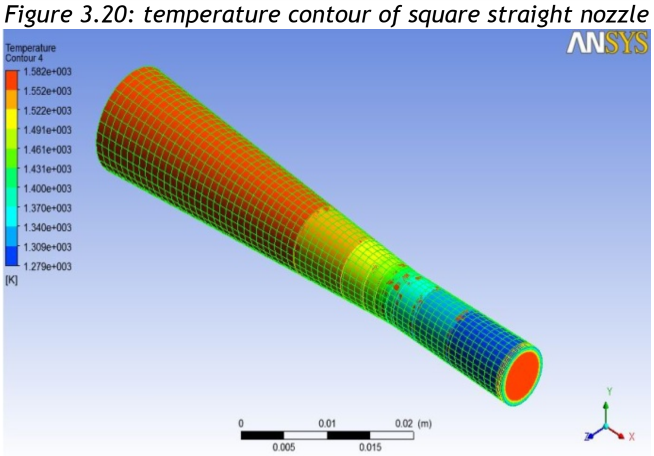

Figure 3.21: temperature contour
of circular straight nozzle

These three results are good in agreement with each other.

Velocity contours:

By observing the velocity contours for rectangular bent in shape nozzle. Exit velocity and throat velocity and same as the theoretical calculations, and maximum incurred velocity is in this nozzle is ranging from the input value to 2082 m/s. For the same inlet and outlet condition and keeping the inlet, exit, throat area as like bent in shape rectangular nozzle is allowed to pass through straight in rectangular shape nozzle the maximum velocity incurred in this case is from inlet valve to 1475 m/s the same flow is allowed for straight in square shape nozzle the maximum velocity incurred is of range 1122 m/s. The same flow is allowed for straight circular in shape nozzle the maximum velocity incurred is of range 1114 m/s to minimum the machining cost of rectangular bent in shape nozzle, we can use straight in rectangular shape nozzle.

Pressure contours:

By pressure contours observation we conclude that maximum pressure incurred is of range 103 bar for bent rectangular shape. In straight rectangular nozzle maximum pressure will reach till the throat and drives to increase the exit velocity and in this pressure contours the maximum pressure incurred is of 80 bar. In straight square nozzle the pressure will not reach maximum length and velocity obtained is of range 1122 m/s at exit.

In circular straight shape nozzle maximum incurred pressure is of 79 bar and incurred maximum velocity is of 1114 m/s velocity at exit.

Temperature contours:

Temperature contours obtained for rectangular bent in shape is at range of 6976 K which is a very high value compared to the rectangular straight in shape nozzle. For exit velocity of 1475 m/s the maximum temperature value in the contours is 1588 K and impact of temperature reach till the throat is very high when compared to circular and square in shape nozzles.

CONCLUSIONS

From the investigation we have done, the following conclusions have been drawn.

- *CFD analysis has been done on Convergent-Divergent nozzles of different cross sections like rectangular, square and circular.*
- *It has been found that rectangular nozzle gives a velocity of 1475 m/s where as square nozzle gives a velocity of 1122 m/s and circular nozzle gives a velocity of 1114 m/s.*
- *Thus, rectangular nozzle gives an increased velocity of about 23.93% compared to square nozzle and about 24.47% compared to circular nozzle.*
- *Velocity increases when pressure drops. It has been found that rectangular nozzle gives a pressure drop of 73.392 bar where as square nozzle gives a pressure drop of 56.56 bar and circular nozzle gives a pressure drop of 55.8 bar.*
- *Thus, rectangular nozzle gives an increased pressure drop of about 22.93% compared to square nozzle and about 23.97% compared to circular nozzle.*
- *It has been found that rectangular nozzle gives a temperature drop of 538 K where as square nozzle gives a temperature drop of 309 K and circular nozzle gives a temperature drop of 303 K.*
- *Thus, rectangular nozzle gives an increased temperature drop of about 42.56% compared to square nozzle and about 43.68% compared to circular nozzle.*
- *Hence, it has been found from the results that rectangular nozzle gives high exit velocity, high pressure drop and high temperature drop along the nozzle compared to square and circular nozzles.*
- *Computational results are in good agreement with the experimental results of N.S.T.L. and also with the theoretical results published in the literature.*

Nomenclature

CO	*carbon monoxide*
HC	*hydro carbon*
NO_x	*oxides of nitrogen*
CO_2	*carbon dioxide*
HSU	*Hatridge smoke unit*
BTDC	*before top dead centre*
Y	*total percentages of uncertainty*

REFERENCES

[1] *A.A.Khan and T.R.Shem bharkar, “Viscous flow analysis in a Convergent-Divergent nozzle”. Proceedings of the international conferece on Aero Space Science and Technology, Bangalore, India, June 26-28, 2008.*

[2] *H.K.Versteeg and W.Malala Sekhara, “An introduction to Computational fluid Dynamics”, British Library cataloguing pub, 4th edition, 1996.*

[3] *David C.Wil Cox, “Turbulence modeling for CFD” Second Edition 1998.*

[4] *S.Majumdar and B.N.Rajani, “Grid generation for Arbitrary 3-D configuration using a Differential Algebraic Hybrid Method, CTFD Division, NAL, Bangalore, April 1995.*

[5] *Layton, W.Sahin and Volker.J, “A problem solving approach using Les for a backward facing-step” 2002.*

[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.*

ACTA TECHNICA CORVINIENSIS – Bulletin of Engineering

ISSN: 2067-3809 [CD-Rom, online]