COMPUTATIONAL FLUID DYNAMIC ANALYSIS OF CONVERGENT-DIVERGENT NOZZLE

P Rajamani Dept. of Mechanical Engineering, St. Martin's Engineering College, Hyderabad, India
K Sunitha Dept. of Mechanical Engineering, St. Martin's Engineering College, Hyderabad, India
e-mail: raji40jes@gmail.com, sunitha.skannaram@gmail.com
*Corresponding Author: raji40jes@gmail.comTel.: 9949752426

Abstract:

The convergent-divergent nozzle finds application in many areas of industry and technology. The present work focuses on the CFD simulation of a convergent-divergent nozzle undergoing compressible flow, as there exist a few works on this topic. The work performed using a standard geometry to capture the normal shock under different exit pressure conditions. Finite volume method is to be utilized for the numerical simulations using ANSYS FLUENT. The results are to be analyzed using pressure plot, Mach number plot, different contour plots for pressure, temperature, density, velocity, flow. The study is performed to contribute towards the fundamental knowledge and practical applications.

Keywords: CFD simulation, ANSYS FLUENT

INTRODUCTION

A nozzle (from nose, signifying 'little gush') is a container of changing cross- sectional territory (for the most part hatchet symmetric) going for expanding the speed of a surge, and controlling its bearing and shape. Nozzle stream dependably creates powers related to the adjustment in Stream Energy, as we can feel by handholding a hose and opening the tap. In the least complex instance of a rocket nozzle, relative movement is made by shooting mass from a chamber in reverse through the nozzle, with the response powers acting chiefly on the contrary chamber divider, with a little commitment from nozzle dividers. As imperative as the propeller is to shaft-motor impetuses, so it is the nozzle to stream drive, since it is in the nozzle that warm vitality (or some other sort of high-pressure vitality source) changes into active vitality of the fumes, and its related direct energy creating push. The stream in a nozzle is extremely fast (and accordingly adiabatic to a first estimate), and with next to no frictional loses (in light of the fact that the stream is almost one-dimensional, with a great pressure angle aside from if stun waves structure, and nozzles are moderately short), so the isentropic model up and down the nozzle is adequate for starter plan.

LITERATURE REVIEW

Early takes a shot at coaxial planes were persuaded for the most part by applications in combustion and air ship impetus. Forestall and Shapiro (1950) were the first to play out a trial examination on mass and energy exchange between the two floods of a co-streaming plane with exceptionally huge auxiliary streams in subsonic locale, and demonstrated that the speed proportion of the essential to optional stream is the central parameter deciding the state of the blending district, and proposed an exact connection for the length of the essential potential center.

DESIGN OF MODULES IN NX 12

NX, formerly known as "Unigraphics", is an advanced high-end CAD/CAM/CAE, which has been owned since 2007 by Siemens PLM Software. In 2000, Unigraphics purchased SDRC I- DEAS and began an effort to integrate aspects of both software packages into a single product which became Unigraphics NX or NX.

The most powerful, flexible, and innovative product development solution in the industry, NX for Design has the features, performance, and capabilities to help you get product to market faster than ever before for Design enables you to deliver products "right to market, first time" using more virtual product models and fewer, more costly, physical prototypes.

NX FOR MANUFACTURING

Drive efficient end-to-end part manufacturing operations and deliver high-precision parts through digitalization. Program CNC machine tools, control robotic cells, drive 3D printers and monitor quality using one software system. Digitally transform your part manufacturing business to gain productivity and increase profitability.

NX MODEL OF NOZZLE

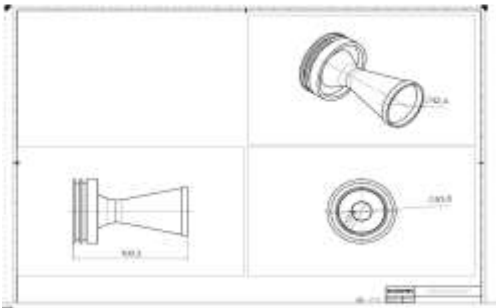

Fig.1: DRAFT SHEET 1

DIMENSIONS

MEASUREMENTS	VALUES
Length	100.3 mm
Inlet diameter	53 mm
Throat diameter	17 mm
Outlet diameter	42 mm

DESIGN OF NOZZLE 1

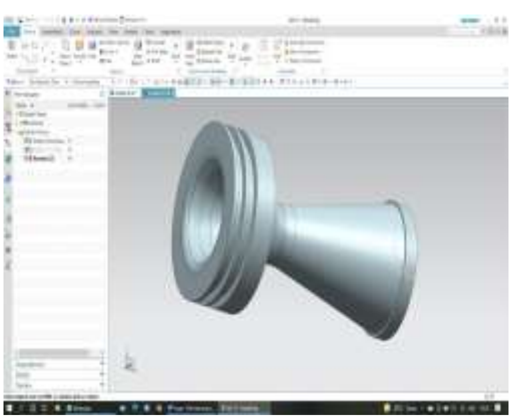

Fig.2: Internal flow of Nozzle 1

INTERNAL FLOW OF NOZZLE 1

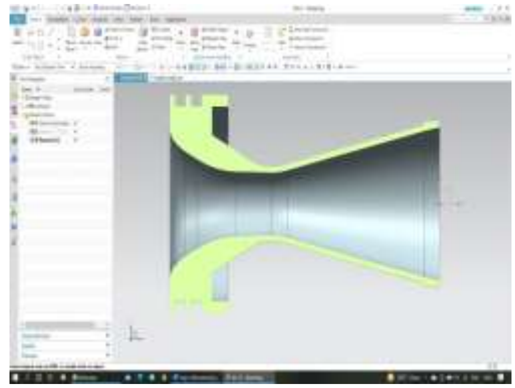

Fig.3: Internal flow of Nozzle 1

DIMENSIONS

MEASUREMENTS	VALUES
Length	100.3 mm
Inlet diameter	53 mm
Throat diameter	16 mm
Outlet diameter	42 mm

DRAFT SHEET OF NOZZLE 2

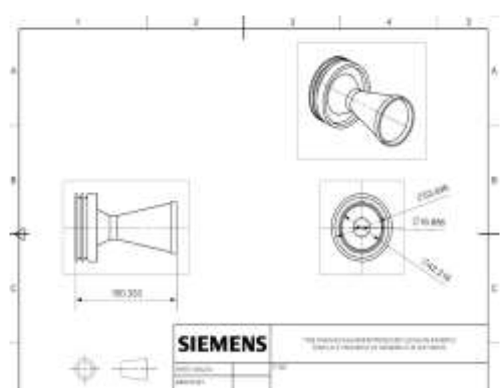

Fig.4: DRAFT SHEET 2

DESIGN OF NOZZLE 2

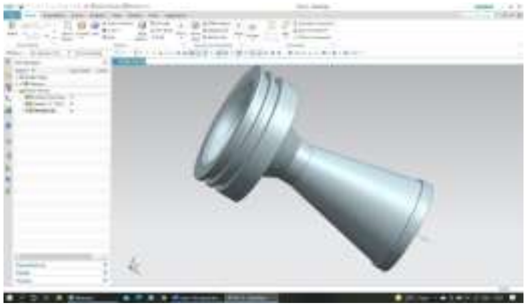

Fig.5: Design of nozzle

INTERNAL FLOW OF NOZZLE 2

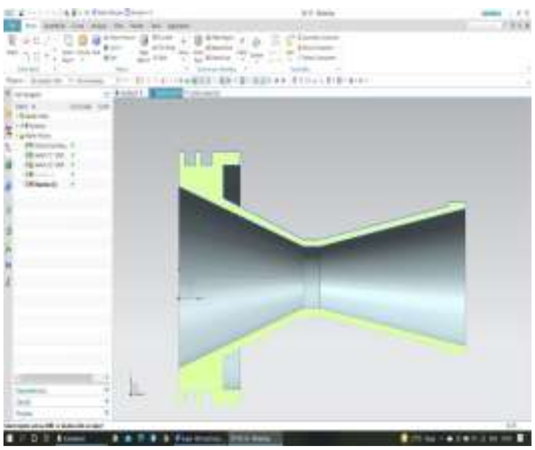

Fig.6: Internal flow of nozzle 2

INTRODUCTION TO ANSYS

ANSYS is business limited component examination programming with the ability to separate a broad assortment of different issues. ANSYS holds running under a collection of circumstances, including IRIX, Solaris, and Windows NT. Like any limited component programming, ANSYS unwinds speaking to differential conditions by breaking the issue into little components. The speaking to states of flexibility, fluid stream, warm trade, and electro-fascination would all have the capacity to be comprehended by the Finite component procedure in ANSYS. ANSYS can handle transient issues and moreover nonlinear issues. This report will focus on the basics of ANSYS using generally fundamental delineations.

The ANSYS program is a multi-reason program, suggesting that you can use it for a limited component investigation in for all intents and purposes any industry - vehicles, flying, railways, mechanical assembly, equipment, wearing stock, control time, control transmission, and biomechanics, to indicate just a couple. "Multi-reason" in like manner suggests the manner in which that the program can be used as a piece of all controls of structure – assistant, mechanical, electrical, electromagnetic, electronic, warm, fluid, and biomedical. The ANSYS program is moreover used as an enlightening gadget in schools and other academic foundations.

ANSYS GEOMETRY OF NOZZLE 1

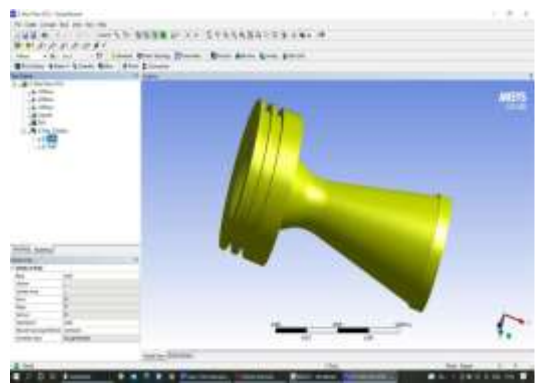

Fig.7: ANSYS Geometry of nozzle 1

INTERNAL FLOW SURFACE AREA OF NOZZLE 1

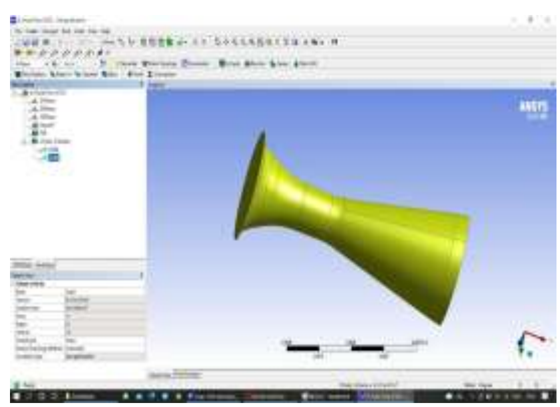

Fig.8: Internal flow surface area of nozzle 1

MESH ANALYSIS OF NOZZLE 1

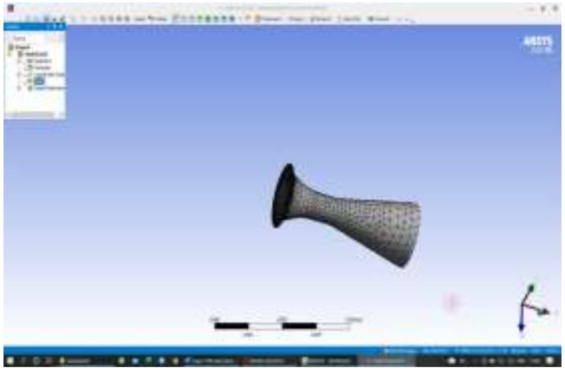

Fig.9: Mesh analysis of nozzle 1

Fig.10: Inlet of nozzle 1

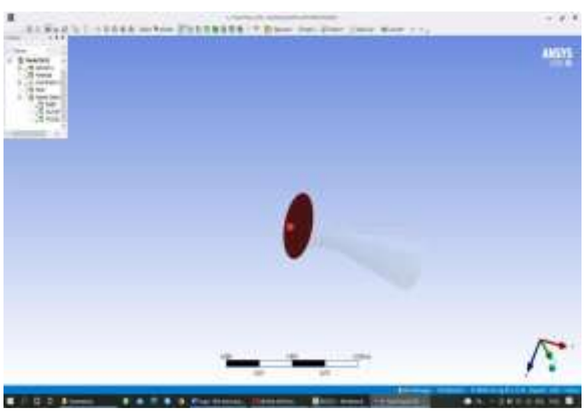

Fig.11: Outlet of nozzle 1

Fig.12: Nozzle wall 1

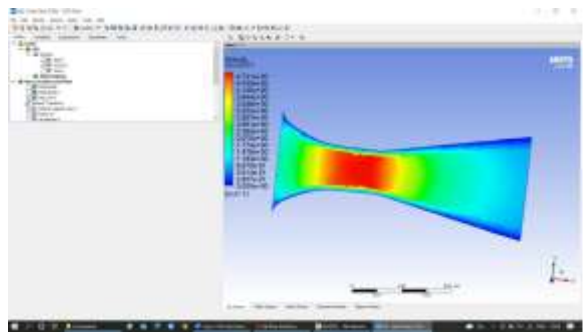

Fig.13: Velocity 1

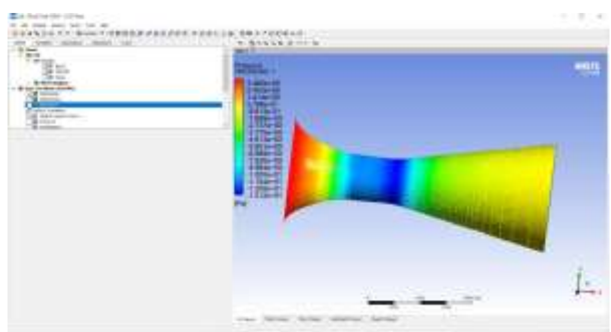

Fig.14: Pressure 1

ANSYS GEOMETRY OF NOZZLE 2

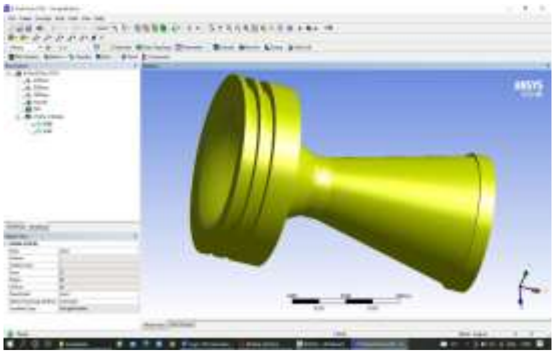

Fig.15: Geometry of nozzle 2

INTERNAL FLOW SURFACE AREA OF NOZZLE 2

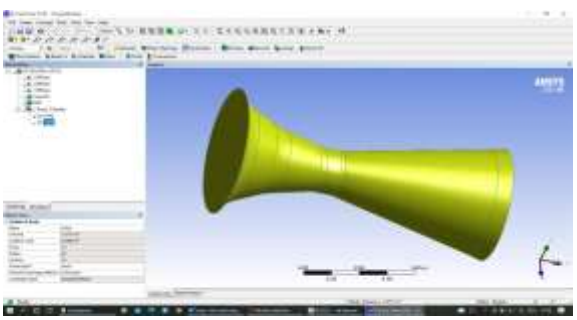

Fig.16: Internal flow surface are of nozzle 2

VELOCITY 1 NOZZLE 2

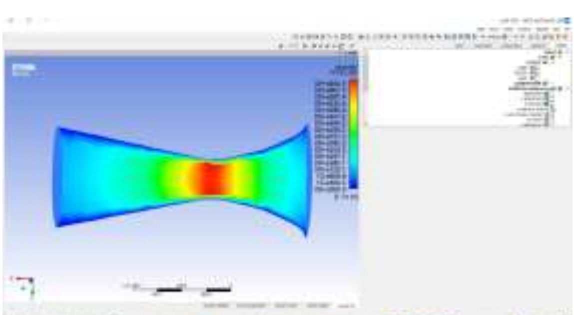

Fig.17: Velocity 1 at nozzle 2

INLET OF NOZZLE 2

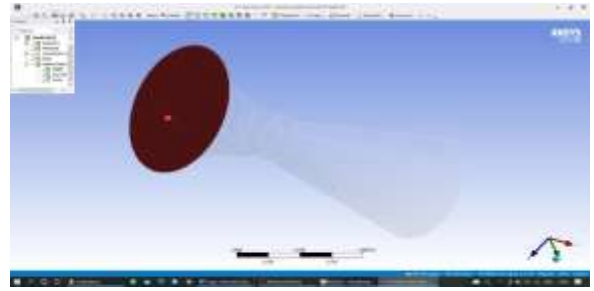

Fig.18: Inlet of nozzle2

PRESSURE OF NOZZLE 2

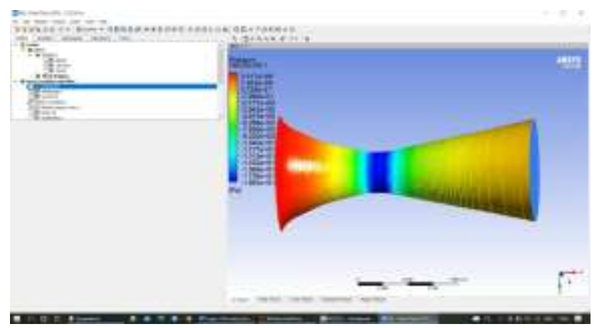

Fig.19: Pressure of nozzle 2

NOZZLE WALL 2

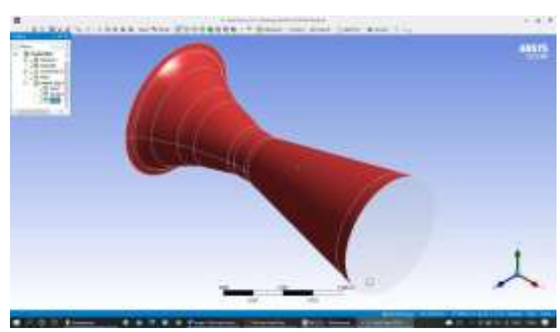

Fig.20: Nozzle wall 2

RESULTS AND DICUSSIONS

The below table shows the difference in pressure and velocity at different points

- Inlet pressure of nozzle is 3.9pa where outlet and pressure at throat is -
- 12.61 and -0.02 respectively.
- Velocity in nozzle 1 is varying from inlet to outlet through throat is 2, 4.8,
- 2.1 ms^-1 respectively as shown in above table.
- When it comes to nozzle 2 the inlet pressure is 1.3pa and outlet pressure is
- -0.08pa and the throat pressure is-18.26.
- The velocity in nozzle 2 is, at inlet it is 1 ms^-1, at outlet is 1.002 ms^- 1and at the throat is 1.84 ms^-1

S.NO		PRESSURE (pa)			VELOCITY(ms^-1)		
		P1 (inlet)	P2 (outlet)	P3 (throat)	V1 (inlet)	V2 (throat)	V3 (outlet)
1	NOZZLE1	3.9	-12.61	-0.02	2.00	4.8	2.1
2	NOZZLE2	1.3	-0.08	-18.26	1.00	1.84	1.022

CONCLUSIONS

The results obtained analytically are in accordance with theoretical relations used thus validating the results. It can be concluded that the optimization of the design resulted in better performance that is air is accelerated at must faster rate and reduced pressure in the CD Nozzle model than the conventional cylindrical model.

From the above table we can conclude that when we decrease the throat diameter the pressure and velocity will be changed. We can also observe that there will be more decreases in pressure than the decrease in velocity

This type of nozzles has wide range of applications in aerospace industry like in ramjet engine, rocket engine etc.

REFERENCES

1.Md Iliyaas A, Karuppasamy K (2016) Computational analysis of CD nozzle for solid propellant rocket. Int J Innov Works Eng Technol (IJIWET)

2. Sudhakar BVVN, Sekhar BPC, Mohan PN, Md Ahmad T (2016) Modeling simulation of convergent-divergent nozzle using computational fluid dynamics. Int Res J Eng Technol (IRJET) .

3. H. Zhou, F. Xu, X. Zhang, and Z. Jia, “Effect of thickness on oxidation resistance of coatings on 490N apogee engine,” Aerospace Materials & Technology, vol. 41, no. 5, pp. 66–68, 2011

4.Deepak D 2011 Numerical Analysis of Flow through Abrasive Water Suspension Jet: The effect of Inlet Pressure on Wall Shear and Jet Exit Kinetic Energy Proceedings of World Congress of Engg III WCE

5.Deepak D, Anjaiah D, Vasudeva Karanth K and Yagnesh Sharma N 2012 Effect of inlet operating pressure and volume fraction on skin friction and exit kinetic Energy (Manipal, India: Manipal University)