RESEARCH ARTICLE　　　　　　　　　　　　　　　　　　　　OPEN ACCESS

Fluid Structure Interaction Based Investigation of Convergent-Divergent Nozzle

Siva V*, Prof . V. Sankar**

*(M.E, Department of Aeronautical Engineering, NEHRU INSTITUTE OF ENGG & TECH, Coimbatore-105)
** (Head, Aeronautical Engineering, NEHRU INSTITUTE OF ENGG & TECH, Coimbatore-105)

ABSTRACT

In present work, a convergent-divergent nozzle is designed aerodynamically. Structural loads of nozzle from fluid flow and temperature of fluid in it. Initially, CFD simulation is conducted with thermal considerations in range of nozzle NPR values. Besides, CFD result is used as load in structural analysis of nozzle. Nozzle is working in range of NPR in its service from designed NPR. Maximum load over nozzle structure may happen at any NPR value of flow. Problem associated with nozzle structure design is prediction of load at various nozzle pressure ratio ranges. It is difficult to find and solve this problem analytically. Experimentation will lead to investigation as costlier one. Therefore, we require computation method to find load at various NPR. Maximum load from CFD simulation is used to structural simulation for finalizing thickness of nozzle. Fluid-Structure Interaction (FSI) is the interaction of some movable or deformable structure with an internal or surrounding fluid flow. Fluid-structure interaction can be stable or oscillatory. Fluid-structure interaction problems and multi-physics problems in general are often too complex to solve analytically and so they have to be analyzed by means of experiments or numerical simulation

Key Words: FSI, Computational Fluid Dynamics, Structural Simulation.

I.　INTRODUCTION

Fluid–structure interaction (FSI) is the interaction of some movable or deformable structure with an internal or surrounding fluid flow. Fluid–structure interactions can be stable or oscillatory. In oscillatory interactions, the strain induced in the solid structure causes it to move such that the source of strain is reduced, and the structure returns to its former state only for the process to repeat. Fluid–structure interactions are a crucial consideration in the design of many engineering systems, e.g. aircraft and bridges. Failing to consider the effects of oscillatory interactions can be catastrophic, especially in structures comprising materials susceptible to fatigue. Tacoma Narrows Bridge (1940), the first Tacoma Narrows Bridge, is probably one of the most infamous examples of large-scale failure. Aircraft wings and turbine blades can break due to FSI oscillations. Fluid–structure interaction has to be taken into account for the analysis of aneurysms in large arteries and artificial heart valves. A reed actually produces sound because the system of equations governing its dynamics has oscillatory solutions. The dynamic of reed valves used in two strokes engines and compressors is governed by FSI. The act of "blowing a raspberry" is another such example

1.2 Simulation

The Newton–Raphson method or a different fixed-point iteration can be used to solve FSI problems.

Methods based on Newton–Raphson iteration are used in both the monolithic and the partitioned approach. These methods solve the nonlinear flow equations and the structural equations in the entire fluid and solid domain with the Newton–Raphson method. The system of linear equations within the Newton–Raphson iteration can be solved without knowledge of the Jacobin with a matrix-free iterative method, using a finite difference approximation of the Jacobean-vector product.

Whereas Newton–Raphson methods solve the flow and structural problem for the state in the entire fluid and solid domain, it is also possible to reformulate an FSI problem as a system with only the degrees of freedom in the interface's position as unknowns. This domain decomposition condenses the error of the FSI problem into a subspace related to the interface. The FSI problem can hence be written as either a root finding problem or a fixed point problem, with the interface's position as unknowns.

1.2.1 Convergent Nozzle

Convergent nozzles are used on many jet engines. If the nozzle pressure ratio is above the critical value (about 1.8:1) a convergent nozzle

will choke, resulting in some of the expansion to atmospheric pressure taking place downstream of the throat (i.e. smallest flow area), in the jet wake. Although jet momentum still produces much of the gross thrust, the imbalance between the throat static pressure and atmospheric pressure still generates some (pressure) thrust.

1.2.2 Divergent Nozzle
The supersonic speed of the air flowing into a scramjet allows the use of a simple divergent nozzle.

1.2.3 Convergent-Divergent Nozzle
Engines capable of supersonic flight have convergent-divergent exhaust duct features to generate supersonic flow. Rocket engines the extreme case owes their distinctive shape to their very high area ratios of their nozzles. When the pressure ratio across a convergent nozzle exceeds a critical value the pressure of the exhaust exiting the engine exceeds the pressure of the surrounding air. This reduces the thrust producing efficiency of the nozzle by causing much of the expansion to take place downstream of the nozzle itself. Consequently, rocket engines and jet engines for supersonic flight incorporate a C-D nozzle which permits further expansion against the inside of the nozzle. However, unlike the fixed convergent-divergent nozzle used on a conventional rocket motor, those on turbojet engines must have heavy and expensive variable geometry to cope with the great variation in nozzle pressure ratio that occurs with speeds from subsonic to over Mn3. For a subsonic application of a fixed geometry C-D nozzle see section "Low ratio nozzle".

II. INDENTATIONS AND EQUATIONS
The design is calculated by using following formulas

$$T^* / T_1 = (2 / \gamma + 1) \qquad (1)$$
$$P^* P_1 = (2/\gamma + 1)^{\gamma/(\gamma-1)} \qquad (2)$$
$$V^* = (\gamma R T^*)^{1/2} \qquad (3)$$
$$V_{exit} = 44.72 * (Cp [T_1 - T_{exit}])^{0.5} \qquad (5)$$

$L_{div \ nozzle}$ (length of divergent nozzle)
$$= (d_{exit} - d^*) / 2 * \tan \alpha_d \qquad (6)$$
$L_{cov \ nozzle}$ (Length of convergent nozzle)
$$= (d_1 - d^*) / 2 * \tan \alpha_c \qquad (7)$$
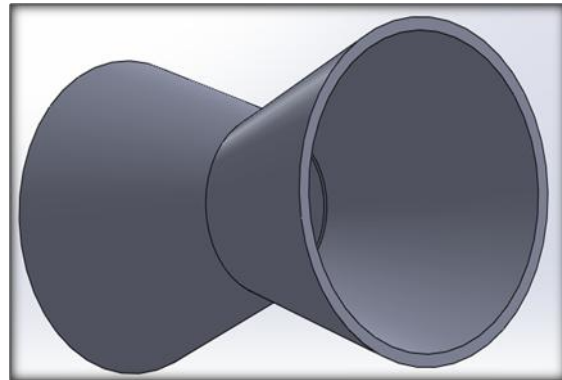
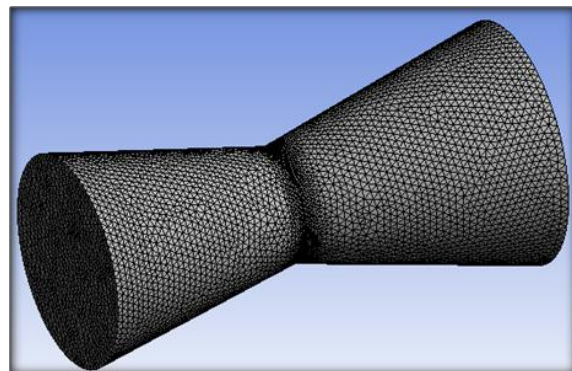### III. FIGURES AND TABLES

FIG 1. CAD MODEL

Fig 2. Meshing for CFD
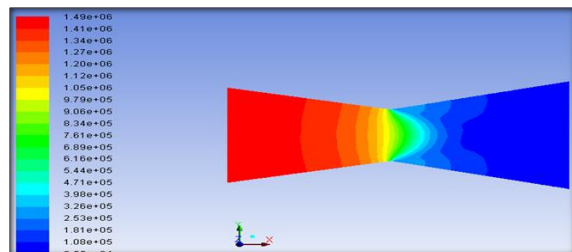Cells – 32604
Nodes-170326

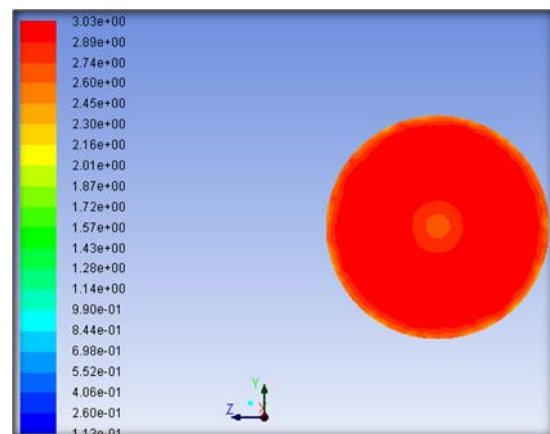*Fig 3. static pressure plot through length*
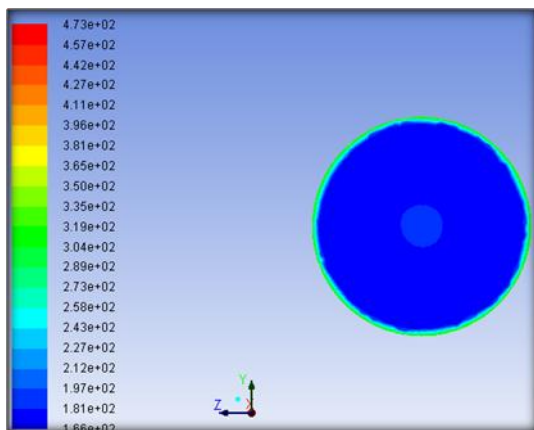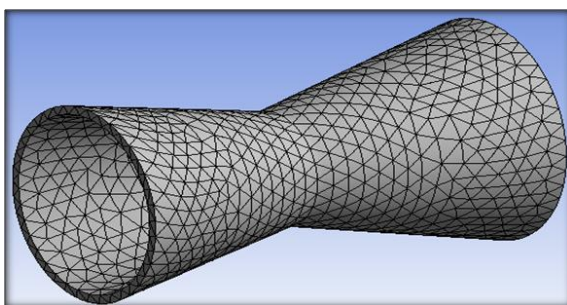

Fig 4. mach outlet

Fig 5. temperature outlet

Fig 6. meshing for structural somulation
Nodes – 9792
Elements – 4782

IV.CONCLUSION

Fluid-Structure Interaction based investigation of convergent-divergent nozzle is proposed. Aerodynamic design of nozzle is done and will be simulated with different NPR using commercial CFD software. Besides, structural simulation is done to obtain suitable thickness. Problem associated with nozzle structural design is prediction of load at various nozzle pressure ratio ranges. It is difficult to find and solve this problem analytically. Experimentation will lead to investigation as costlier one. Therefore, we require computation methods to find load at various NPR. Maximum load from CFD simulation is used to structural simulation for finalizing thickness of nozzle. Dimensions of nozzle are found using calculation except thickness. It needs structural consideration. Initially, load applied over body during operation is unknown which is given by fluid pressure. It is theoretically unpredictable throughout flow and also non-linear. Therefore, we are looking for CFD to predict it and generate load for structural analysis. Flow and temperature result from CFD is used for structural analysis in mechanical APDL solver. It has been explained clearly in previous chapter. Operating condition of nozzle is denoted by NPR Values. Shockwave not comes into play of this analysis. No shockwave comes into nozzle at wide range of NPR.

In CFD, to achieve different NPR values, inlet and outlet conditions are varied. Those are listed below. From these results, structural analysis is done for default thickness of 5mm initially because structural solver needs some solid object. Load of different NPRs are applied and individual structural analysis are done with 10mm thickness and it is observed which NPR load is giving maximum stress than others. Using load of that NPR and structural analysis done for different thickness. NPR value 1.51 gives maximum stress (CFD is done for NPR 0.257 to 1.89). Material is titanium alloy. For load of 1.51 NPR value, structural analysis of nozzle is done with FOS of 3 by varying nozzle thickness. Finally, 309 MPa is induced for 12.5mm thickness. Therefore, this is thickness of nozzle can be used.

REFERENCES

[1]. Aginsky Z. et. al (2011) '*Numerical analysis of nonlinear acoustic fluid-structure interaction of a two dimensional plate in an in-viscid compressible fluid*', ENOC, pg: 24-29.

[2]. Ali S. et. al (2013) '*Numerical analysis and simulation of the interaction between a Von Kármán vortex street and elastic flaps*', Congress François de Technique.

[3]. Birken P. et. al (2014) '*Thermomechanical Coupling In Fluids, Structures and Fluid-Structure-Interaction*', 6th. European Conference on Computational Fluid Dynamics.

[4]. Driss Z. et. al (2007) '*Computer Simulations of Fluid-Structure Interaction Generated by a Flat-Blade Paddle in a Vessel Tank*', International Review of Mechanical Engineering, *Vol. 1, no. 6*, pg: 608-617.

[5]. Goud S.K. et. al (2014) ' *Analysis of Fluid-Structure Interaction on an Aircraft Wing*', International Journal of Engineering and Innovative Technology, Volume 3, Issue 9, pg: 146-152.

[6]. Guruswamy G.P. et. al (1988) '*Integrated Approach for Active Coupling of Structures and Fluids*', AIAA Journal Vol. 27, No. 6, pg: 788-794.

[7]. Hai E.M.S.B. et. al (2013) '*Numerical Simulation of Fluid Structure Interaction (FSI) on a double wedge Airfoil based on Arbitrary Lagrangian-Eulerian Frameworks*', The workshop on Numerical Modelling of Grains/Fluid Flows, pg:1-6.