REVIEW PAPER ON MODELLING AND ANALYSIS OF THRUST VECTOR CONTROL NOZZLE.

K. VINAY KUMAR REDDY[1], J. ABHIJEET DATTA[2], M.LAXMI TEJA[3], P.VAISHNAV[4]

ABSTRACT

This paper deals with the modelling and analysis of thrust vectoring nozzle. Thrust vectoring nozzle are widely used in military jet aircrafts, these nozzles are called has geometrical variable nozzles because of change in the nozzle area according to the application required. These nozzles have more advantages then the conventional nozzles, because they can create greater amount of lift for the aircraft with small deflection in the nozzle which is useful for the aircrafts manoeuvrability. Thrust vectoring nozzles are best suitable for short runway take off and for vertical take-off. We have designed a 3D model of the thrust vectoring nozzle of a military jet aircraft with appropriate dimensions using CATIA software. The nozzle performance parameters such has thrust, exit pressure and exit velocity were calculated. The designed nozzle was subjected to fluent and structural analysis. We subjected the nozzle to certain deflection for thrust analysis in ANSYS. And from the observations we found that when the deflection angle is increased the thrust developed by the nozzle increases.

KEYWORDS: Thrust vectoring nozzle, thrust, exit pressure, exit velocity, CATIA 3D software, ANSYS analysis soft

LITERATURE SURVEY

D. Daljit Majil et al.[1] (2013) conducted an design of Thrust vectoring Nozzle using CFD analysis for 2D nozzle design on which flow takes place at Subsonic speed and it is analysed to the variation in various deflection angle and Flow Parameters. Flow Parameters such as Pressure, velocity and mass flow rate are noted and Thrust generation will be calculated with the help of this Flow parameters. Same procedure is repeated for different deflection angle of nozzle. Main objective of this paper is to optimize the effectiveness of Deflection angle in the TVC Nozzle. Hence calculated numerical values are used to plot the graph between Deflection angle vs Thrust. Another Graph which graphically represents the proportionality between the difference in Thrust and Deflection angle of Nozzle. Hence effectiveness of Deflection angle is optimized to increase the application of Thrust Vectoring in Subsonic Aircrafts.

Abhilash Patil et al. [2] (2019) conducted designing and analysis on a Convergent-Divergent nozzle with various cases of vectoring and non-vectoring to attain supersonic flow and optimizing it to achieve maximum drop pressure, Mach number. The analysis is performed according to the Convergent-Divergent nozzle with various cases i.e. mechanically deflecting the divergent section downward by an amount 5°, 10°, 15° and 20° angles over an axis keeping the same boundary condition. Our objective is to investigate Pressure drop, Mach number for the various cases of vectoring and non-vectoring Convergent-Divergent nozzle using CFD. The baseline Convergent Divergent Nozzle i.e. non-vectoring nozzle analysis is carried out using various turbulence model (Enhanced kepsilon, Spalart–Allmaras, SST k-omega) to investigate the impact of turbulence models on Pressure drop and Mach number. From this study it reveals that, the baseline case i.e. Convergent-Divergent nozzle with non-vectoring using Spalart-Allmaras turbulence model with 5% Turbulence Intensity resulted into less pressure drop (12.83 %) as compared to k-epsilon (12.98%) and k-omega (14.10%) models. Attained maximum supersonic flow with exit Mach number using Spalart-Allmaras model is 2.07. Further, CFD analysis is conducted on vectoring cases of Convergent-Divergent nozzle i.e. downward deflection of divergent portion of nozzle (5°, 10°, 15° and 20° angles) using Spalart-Allmaras turbulence model. It has been found that Convergent-Divergent nozzle gives to increase Mach number. So, thrust vectoring applicable in Convergent-Divergent nozzle. Further deflecting with various angles thrust vectoring will be varying. In downward defection of thrust vectoring nozzle, 10° gives maximum increased in exit Mach number i.e.2.1251 which is 2.65% more than baseline case (non-vectoring nozzle). The appreciable amount of increased in the exit Mach number is seen in vectoring cases compared to non-vectoring case of Convergent-Divergent nozzle.

Arif et al.[3] (2018) his research aimed to integrate engine exhaust nozzle on a supersonic fighter aircraft and analyse its flow characteristics and variation in performance parameters due to its integration. Engine propulsion characteristics and parameters such as nozzle inlet temperature and total pressure have been analysed through an in-house validated engine analytical model developed by some of the authors of this study. In the first part of paper, exhaust plume structure has been analysed to study the flow behaviour (flow turbulence and flow distortion etc.) at nozzle exit. Later, nozzle performance parameters such as Exit Velocity, Nozzle Pressure Ratio (NPR), Engine Pressure Ratio (EPR), and Engine Temperature Ratio (ETR) have been calculated when exhaust nozzle is integrated with the aircraft. Finally, the results are compared and validated with analytical calculations to compare the performance of nozzle when it is in isolation and when it is integrated on aircraft. It is observed that nozzle flow has no significant effect on aircraft major surfaces such as fuselage, wing upper and lower surfaces, and nose section. However, there is a prominent effect of exhaust nozzle flow on horizontal stabilizers, vertical tail and rear fuselage area of the aircraft. An average difference of 18% in NPR, 12% in EPR, and 9% in ETR is observed between integrated nozzle and isolated nozzle which further signifies the importance of integrating exhaust nozzle in aircraft analysis. This proposed methodology will allow more accurate analysis of the effects of exhaust nozzle on the overall performance of aircraft. The methodology can further be used for proposing design changes in existing nozzle configurations.

A. Gogoi et al.[4] did CFD analysis of the thrust vectoring system along with a transition duct shows significant jet deflection. Preliminary design of the thrust vectoring system components are carried out based on the CFD analysis and a kinematic analysis of the system shows the feasibility of the designed kinematic chain. Yaw thrust vectoring mechanism comprising of a high aspect ratio, rectangular nozzle and five numbers of deflectable vanes is designed for a tail-less aircraft. The vanes were designed for maximum deflection of 10 degrees in the yaw plane. Kinematics Analysis demonstrated the feasibility of jet vane deflection. CFD Analysis of yaw thrust vectoring system is carried out and the corresponding jet deflection is obtained for 5 degree and 10 degree of vane deflection.

Daniel Ikaza [5] (2000) this paper describes the technical features of the Thrust Vectoring Nozzle (TVN) developed by ITP and its advantages for modern military aircraft. It is presented in conjunction with two other papers by DASA (Thrust Vectoring for Advanced Fighter Aircraft High Angle of Attack Intake Investigations) and MTU-Minchin (Integrated Thrust Vector Jet Engine Control) respectively. The ITP concept consists of a patented design featuring the so-called "Three-Ring-System", which allows all nozzle functions (Throat Area, Exit Area, Pitch Vectoring and Yaw Vectoring) to be performed with a minimum number of actuators, which, in turn, leads to optimized mass and overall engine efficiency.

Dr. Norbert C. Bissinger et al.[6] (2000) in this paper the results of investigations will be presented were dealt with the determination of intake flows at high and very high angles of attack. The overall approach comprised of experimental and numerical (CFD) investigations will be detailed. Results and comparisons between flows at small and large angles of attack were used to try to clarify the origin of possible disturbances in the intake flow. Intake bleed and the flow losses at the intake entrance are identified as major components contributing to the quality of the intake flow at the engine face usually is designed for. Therefore it must be ensured that at these high angles of attack the flow inside the intake will not be deteriorated so much that serious intake/engine compatibility problems would occur. These problems could lead not only to thrust losses at very critical flight situations but could also result in damages of the engine and/or aircraft due to the very high pressure loads connected withmal functioning (surge) of the engine.

M. N. Tomac et al.[7] (2016) conducted various Jet vectoring performances of ten different designs with various depths and geometrical outlines were quantified through constant temperature anemometry measurements for a Reynolds number range from 10,000 to 30,000 by using passive and active flow control methods at cold flow. The reference design was based on NASA's double throat nozzle concept and a self-injection double throat nozzle design that uses similar flow control concept as the reference design, were also tested for performance comparison. Furthermore, jet vectoring performance of a single throat design, utilizing Coanda effect for jet vectoring, was also quantified. Results indicated jet vectoring angles starting from 2° up to 47° for a control jet flow rate range from 1% up to 10% with respect to the primary jet flow rate in the investigated Re range. Maximum jet vectoring angle was achieved with a single throat design which incorporates small step geometry before the

Coanda surface for more effective flow attachment and these results were compared with the vectoring performance of the double throat nozzle designs.

Shaik Khaja Hussain et al.[8] (2016) there objective of the present work was to simulate and understand Supersonic flows with single jet Flow at various Mach numbers. The purpose was to precisely understand the fluid dynamics and variation of flow properties such as velocity, pressure and turbulence in supersonic flow regime for various Mach numbers pressure ratio and dimensionless spacing single jet nozzle. The jet nozzle can take three different materials like Titanium, Tungsten and Nickel. It can observe the material is to be in the supersonic flow in the nozzle. The Mach number at the nozzle exit is observed to be less in comparison with designed value. This is due to the viscosity and turbulence in fluid near the wall of the duct. The Mach number decreases due to shock wave and reversible flow. A single jet nozzle is designed in solid works software and CFD analysis is carried out in solid works flow simulation.

D. Daljit Majil [9](2016) this paper dealt with optimization of various nozzle parameters (C-D Nozzle) such as Radius of Throat; Divergent angle.2-D model of nozzle is designed in ANSYS 14.5 which is then analysed with fluent software for exit velocity for various nozzle parameters at the different values. Taguchi design technique is used for optimization of nozzle parameters to get optimum nozzle parameters of jet vane Thrust vectoring nozzle is analysed for two different operating conditions i.e. one is at standard atmospheric condition and another one is for outer space condition. Jet Vane TVC analysis with the help of CFD and Parameters optimization obtained from the tag chi analysis technique. The following observations were found in the jet vane TVC nozzle of 9(design) × 2(Operating Condition) = 18 times different configuration values of analysis results. Optimum parameters for ambient condition are divergent angle of 30° and Throat radius of 12.5mm. Similarly Optimum parameters for space condition are divergent angle of 7.5° and Throat radius of 12.5mm.Two operating conditions were considered to determine the optimum parameters for nozzle design for both rockets and missiles.

K. Nehru et al.[10] (2015) this paper focused on designing a c-d nozzle with various case of vectoring and non-vectoring to attain supersonic flow and optimizing it to achieve maximum drop pressure, velocity, Mach number with various cases. The analysis has been performed according to the c-d nozzle with various casas and keeping the same input condition. Our objective is to investigate the various case of vectoring and non-vectoring c-d nozzle. In this, initially modelling of the c-d nozzles has been done in GAMBIT and later on mesh generation and analysis has been carried out in GAMBIT FLUENT 6.3 and various contour like pressure, velocity have been taken and their variation according to different case C-D nozzle has been studied. Comparing the c-d nozzle with various degrees. Which can varying Mach number, velocity and pressure values which are used to thrust vectoring. CFD analysis was done on c-d nozzle at different angles. It has been found that c-d nozzle gives to increase velocity and Mach number and drop pressure. Maximum deflect angle at 20 degree the Mach number, velocity increase so, thrust vectoring applicable in c-d nozzle. Further deflecting with various angle thrust vectoring will be varying and increased. Then the solution is converged around 280 iteration residual is ignored after 10-6 value.

Karen A. Deere et al.[11] (2000) they conducted a computational investigation of an axisymmetric Dual Throat Nozzle concept. This fluidic thrust-vectoring nozzle was designed with a recessed cavity to enhance the throat shifting technique for improved thrust vectoring. The structured-grid, unsteady Reynolds Averaged Naiver-Stokes flow solver PAB3D was used to guide the nozzle design and analyse performance. Nozzle design variables included extent of circumferential injection, cavity divergence angle, cavity length, and cavity convergence angle. Internal nozzle performance (wind-off conditions) and thrust vector angles were computed for several configurations over a range of nozzle pressure ratios from 1.89 to 10, with the fluidic injection flow rate equal to zero and up to 4 percent of the primary flow rate. The effect of a variable expansion ratio on nozzle performance over a range of freestream Mach numbers up to 2 was investigated. Results indicated that a 60° circumferential injection was a good compromise between large thrust vector angles and efficient internal nozzle performance. A cavity divergence angle greater than 10° was detrimental to thrust vector angle. Shortening the cavity length improved internal nozzle performance with a small penalty to thrust vector angle. Contrary to expectations, a variable expansion ratio did not improve thrust efficiency at the flight conditions investigated.

INTRODUCTION

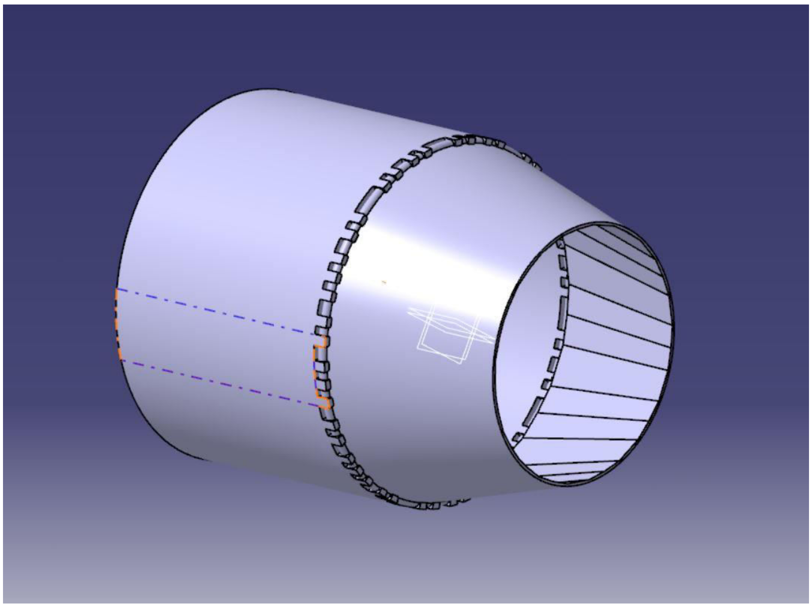
Engine thrust maximisation in fighter aircraft requires variable nozzle geometry in order to optimise jet expansion with engine setting and flight conditions. Maximum thrust is achieved when jet exit pressure and ambient pressure are equal (adapted nozzle). The most common solution in modem fighter aircraft equipped with reheated engines is convergent divergent mono-parametric nozzles. In those nozzles, throat area constitutes the parameter to be controlled according to the following criteria: maximum thrust is envisaged, surge margin should be kept within reasonable limits and nozzle actuator loads can not be exceeded. Logic to cater for these criteria is implemented within the engine controller. However, ideal maximum thrust cannot be achieved with this configuration for every flight condition and engine setting as the exit area follows a mechanically pre-defined relationship to the throat area. The problem can be overcome adding a second degree of freedom leading to a bi-parametric nozzle, i.e. throat and exit areas are controllable. For every particular case an assessment is needed to estimate if the advantage of the extra thrust / reduced specific fuel consumption compensates for the additional systems for exit area actuation plus the increased overall system complexity and weight.

Thrust Vectoring constitutes the next step in nozzle optimisation and increased functionality. The nozzle is intended to direct the jet in directions other than the engine axis in order to generate lateral forces and moments around the aircraft centre of gravity that can be used for aircraft manoeuvring. In 2-D Pitch only nozzles the jet can be deflected within the vertical plane, so the nozzle complements horizontal control surfaces. Pitch vectoring can be achieved by both rectangular (better stealth characteristics) and round nozzles. In 3-D Pitch and Yaw nozzles, the jet can be deflected in any direction so complementing both horizontal and vertical control surfaces. Regarding the mechanical design, jet deflection can be achieved by either deflecting the whole nozzle or deflecting the divergent section. The second, although more complex from a kinematic point of view, is preferred as it does not have any effect in engine controllability and performance (throat area is normally choked so changes in downstream geometry do not impact upstream flow) and requires a lighter actuation system.

Thrust vectoring nozzle on a military jet.

DESIGNING OF NOZZLE

The geometrical model of thrust vectoring nozzle was designed using the CATIA software Geometric modelling is a branch of applied mathematics and computational geometry that discusses the mathematical methods behind modelling the realistic objects for computer graphics and computer aided design. The software helps in geometric modelling of critical components and structural assemblies. The nozzle consists of the flaps which are controlled by the actuators the outer casing for the air to flow.



3D Design of thrust vectoring control nozzle.

The flaps at the end can move in up and down has well has right and left side direction by means of controlled mechanism by the ANSYS software where furtherer static structure and fluent analysis have been done.

ANALYSIS

Meshing of the model is generated to approximate the geometry and reduces the degree of freedom from infinite to finite. This meshing process is vital in the finite element analysis as the quality of the results generated directly depends on the quality of the mesh. At the same time the computation time will be affected depending on the number of elements (number of nodes).

STATIC STRUCTURAL ANALYSIS

Project Objective:

In this project, we will be able to define total

- deformation and stress, etc.
- Create the static structural analysis system
- Apply different types of materials
- Applying of boundary conditions
- Apply a different type of constraints
- Apply different loads
- Generate the results as per required
- Generate project reports.

Directional Deformation, Equivalent Stress, Maximum Principal Stress, and Minimum Principal Stress. The Static Structural analysis is one of the important analyses in ANSYS Workbench. It is available as Static Structural analysis system under the Analysis System toolbox in the Toolbox window, this system analyses the structural components for displacements (deformation), stresses, strains, and forces under different loading conditions. The loads in this analysis system are assumed not to have damping characteristics (time dependent). Steady loading and damping conditions are assumed in this type of analysis system. To start a new Static Structural analysis system, double-click on Static Structural in the Analysis Systems toolbox in the Toolbox window; the Static Structural analysis system will be added to the Project Schematic window. To start an analysis, first you need to specify the geometry on which the analysis is to be done. To do so, you can import the geometry from an external CAD package, or you can create the geometry in the ANSYS's Design Modeller software. After the model is specified for an analysis, you need to double-click on the Model cell of the Static Structural analysis system to open the Mechanical window. In this window, you can specify the parameters and run the analysis.

FLUENT ANALYSIS

For CFD Analysis of TVC nozzle, Ansys 14.5 software is used for flow simulation.

Analysis procedure is carried out with following setup Procedure Details Solution Setup General Type-Pressure based Velocity Formulation AbsoluteTime-Steady2D space-Planar Models Energy Equation-Off Viscous-K-epsilon Materials Fluid-Air, Density

Procedure

- Nozzle has been drawn along with the duct of length LD 600mm and with required dimensions.
- The surface from the edges of sketch has been generated.
- Meshed it with Fine meshing and on the proximity and the curvature in the sizing of meshing.
- Named the edges that Inlet, Wall and Outlet of the nozzle.
- Analysis procedure has been carried out with required boundary conditions mentioned above.
- Initialisation of the solution with standard Compute from Inlet condition.
- Velocity contour and Pressure Contour have to be made that illustrates the variation Pressure and Velocity in the TVC nozzle. Probe value has been taken at Inlet and Outlet.
- Same procedure has been repeated for different cases i.e. different deflection angle of Nozzle. And values of Flow parameters have to be noted and tabulated.

CONCLUSION

By the analysis of subsonic thrust vectoring control nozzle for 3D model, Effectiveness of Deflection Angle ϴ have to be optimized. Which will give the result of increase in Thrust force, Hence by this analysis and optimization of Nozzle parameters such as Exit velocity, exit pressure and Mass flow rate at Outlet me is to be directly proportional to one of the Geometric parameter of Thrust vectoring nozzle Nozzle i.e. Deflection angle ϴ.