COMPRESSIBLE FLOW THROUGH A SUPERSONIC NOZZLE- A COMPUTATIONAL APPROACH.

Tapas Kumar Nandi and Prosun Roy
Department of Mechanical Engineering Techno India College of Technology, WBUT, Kolkata, India.
Email: nashtake555@gmail.com

Received: 15, July, 2016
Accepted: 26, Aug, 2016

INTRODUCTION

Nozzle is used to convert the chemical thermal energy generated in the combustion chamber into kinetic energy. The nozzle converts the low velocity, high pressure, high temperature gas in the combustion chamber into high velocity gas of lower pressure and temperature. Computational fluid dynamics is a versatile technique of modeling and simulation of flow fields which provides accurate results regarding the flow characteristics of devices like flow over an airfoil, if simulated using CFD code provides the user with each and everything required for the efficient designing- pressure, velocity, density at each fluid particle. A nozzle is a relatively simple device, just a specially shaped tube through which hot gases flow. However, the mathematics, which describes the operation of the nozzle, takes some careful thought. CFD plays a vital role in modern techniques of design optimization by providing the offhand solutions of the flow problems. The solution of Reynolds averaged Navier-Stokes (RANS) equations being transient in nature imposes the complexity in the computational studies of the flow field through CD nozzles and the implementation of an appropriate turbulence model for closure of the RANS equations. The compressible flow regions in nozzles being dominated by strong pressure gradients and complex secondary flows induce discrepancies between the numerical simulations and the experimental measurements. The flow that particularly concerns us is a high-speed flow and very high temperature where it produces shock waves that interact with the boundary layers. The existence of shocks in this type of flow produces heavy load losses. These flow phenomena in the nozzles are very harmful to the solid structures of the nozzle. In this nozzle, the flow first converges down to the minimum area or throat, and then is expanded through the divergent section to the exit at the right. The variable geometry causes these nozzles to be heavier than a fixed geometry nozzle, but variable geometry provides efficient engine operation over a wider airflow range than a simple fixed nozzle. Rocket engines also use nozzles to accelerate hot exhaust to produce thrust. Rocket engines usually have a fixed geometry CD nozzle with a much larger divergent section than is required for a gas turbine. Supersonic nozzle flow separation occurs in CD nozzles at pressure ratios far below their design value that results in shock formation inside the nozzle. In the one-dimensional analysis it is treated that the shock is normal, the flow past the shock stays attached to the wall, and the subsonic flow downstream of the shock expands isentropically to the level of back pressure at the nozzle exit. But in reality the flow detaches from the wall and forms a separation region, subsequently the flow downstream becomes non-uniform and unstable.

LITERATURE REVIEW

Supersonic flow separation in a CD nozzle leads to unstable plume in the exit region. Qing Xiao., et. Al., [1] demonstrated that such separation can be efficiently utilized to enhance the jet mixing. Potential applications include propulsion of jet engines, turbofans, turbo jets, spin-stabilized rockets and ramjet engines.

A. A. Khan and T. R. Shembharkar [2] confirmed that the classical one-dimensional inviscid theory does not reveal the complex flow features in a choked CD nozzle accurately. They used the code Fluent to compute RANS flow in a 2-D CD nozzle for nozzle pressure ratios (NPR) corresponding to presence of shock inside the diverging part of the nozzle. The computed solutions differed from the simple theory so far as shock location, shock structure and after-shocks are concerned.

Pardhasaradhi Natta et al [3] found in the nozzles with different divergence angles considering default divergence angle as 7 degrees that; In the nozzle with divergence angle of 20 degrees Mach number is 1.15 at throat and at divergence angle of 7 degrees Mach number is 1.19. From Default angle Mach number is increasing up to 2.917 at the nozzle exit while for divergence angle
of 20 degrees the Mach number at exit is nearly 2.84. At the throat the velocity magnitude is same for all divergence degrees of angle and it is 260 m/s. Near the wall, the Mach number is decreasing for all the nozzles. This is due to the viscosity and turbulence in the fluid. For a nozzle of divergence angle of 20 degrees the Mach number at exit is very low compared to other nozzles. While when the divergence angle is 30 degrees the Mach number at nozzle exit is 3.06 and but an divergence angle 40 degrees it gives the Mach number at nozzle exit is 3.19 and it is lowest at an divergence angle 20 degrees. The turbulence intensity is very high for a divergence angle of 20 degrees at exit. For maximum velocity we can go with 30 or 40 degrees of divergence angle conical nozzle.

Mykhaylo Fedorov [4] published a paper in which described that exact analytical solutions of one-dimensional gas dynamics are intensively applied in engineering practice as a tool in modelling and simulating the piping systems that utilize a compressible medium as their working fluids. Well-known exact analytical solutions for simple types of flows, i.e. for flow processes in which only a single effect is taken into account (e.g. such limiting cases of flows as isentropic, adiabatic or isothermal), are classics of modern one-dimensional gas dynamics theory formed in the first half of last century. At present, gas dynamics does not possess an exact analytical solution for more than a single factor bringing about changes in fluid properties. In this paper the possibility of obtaining general and particular solutions of a nonlinear ordinary differential equation (ODE) system describing one-dimensional steady-state flow of compressible ideal gas in constant area ducts with a constant heat flux and friction factor is discussed. It shows that ODE system variables can be separated, and integrals can be taken in terms of elementary functions. Since an analytical solution is the most important result of the paper, its detailed derivation is presented.

R. Haouia and all [5] presented the results of a flow of high temperature gas in an axisymmetric nozzle at hypersonic regime. They used a gas mixture composed of five chemical species (O₂, N₂, NO, O, N). The in-stationary partial differential equations (Euler equations), which governs this flow is solved with an explicit scheme using digital finite volume method with two kinetic models to Zeldovich (3 and 17 reactions chemical). They used a 150 mesh node along the X axis and 10 nodes according to the radius Y. They got very interesting results in both cases models mentioned above.

Adamson and Nicholls [6], analyzed nozzle jets experimentally and presented an analytical method for calculating the position of shock inside the nozzle, whereas Lewis and Carlson [7] experimentally determined the distance of the first Mach disc in under expanded supersonic nozzles issuing gas from the nozzle exit plane.

Romine [8] presented the mechanisms of flow separation from the nozzle wall. Back et.al [9] presented the comparison of measured and predicted flows through conical supersonic nozzles. They also presented the wall static pressure for various conical nozzles in the region of 2-D flow as quasi-one dimensional theories is not applicable. They claim that flow through the transonic region was found to depend essentially on local configuration, i.e., on the ratio of radii \( r/r^* \).

Vincent Lijo [10] explained a numerical investigation of transient flows in an axisymmetric over-expanded thrust optimized contour nozzle is presented. These nozzles experience side-loads during start-up and shut-down operations, because of the flow separation at nozzle walls. Two types of flow separations such as free shock separation (FSS) and restricted shock separation (RSS) shock structure occur. A two-dimensional axisymmetric numerical simulation has been carried for a thrust-optimized contour nozzle to validate present results and investigate oscillatory flow characteristics during the start-up processes. Reynolds-

Averaged Navier–Stokes equations are numerically solved using a fully implicit finite volume scheme. Governing equations are solved by coupled implicit scheme. The present work is concerned with comprehensive assessment of the flow features by using Reynolds stress turbulence model. Computed pressure at the nozzle wall closely matched with the experimental data. A hysteresis phenomenon has been observed between these two shock structures. The transition from FSS to RSS pattern during start-up process has shown maximum nozzle wall pressure. Nozzle wall pressure and shear stress values have shown fluctuations during the FSS to RSS transition. The oscillatory pressure has been observed on the nozzle wall for high pressure ratio. Present results have shown that magnitude of the nozzle wall pressure variation is high for the oscillatory phenomenon.

A.Nebbache [11] presented the separation phenomenon in an asymmetric nozzle, where the test gas is assumed as an ideal gas. The system of equations governing the flow is solved using the finite volume method fully implicit type predictor-corrector Mac - Cormack. The turbulence model used is (k-\( \omega \)). This work is based on two configurations: The first configuration consists of (a main nozzle, a box and a secondary nozzle removable collar); the mesh size used is \((236 \times 200)\). The second box without configuration and field of study comprises (a main nozzle and noted areas "Jet" and "Wind"). Several meshes (four meshes) were tested to study the independence of the mesh. The results were compared in both configurations. He presented the separation phenomenon in an asymmetric nozzle, where the test gas is assumed as an ideal gas. The system of equations governing the flow is solved using the finite volume method fully implicit type predictor-corrector Mac - Cormack. The turbulence model used is (k-\( \omega \)). This work is based on two configurations: The first configuration consists of (a main nozzle, a box and a secondary nozzle removable collar); the mesh size used is \((236 \times 200)\). The second box without configuration and field of study comprises (a main nozzle and noted areas "Jet" and "Wind"). Several meshes (four meshes) were tested to study the independence of the mesh. The results were compared in both configurations. The same author [12] studied numerically the separation of a turbulent flow in an axisymmetric nozzle truncated where the test gas is perfect supposed nitrogen. The system of equations governing the flow is solved using the finite volume method with a fully implicit scheme predictor-corrector type of MacCormack. The field of digital integration of this study consists of three distinct parts: the nozzle, the jet and the lower field. Three meshes have been used to study the independence of the mesh.

E.Mahfoudi and All [13] worked on the physical analysis and numerical simulation of turbulent separated flow in a supersonic nozzle truncated ideal contour, the turbulence is modeled by a statistical approach (FRANS) in generalized coordinates with the use of the model (SST-Menter). The system of equations governing this flow is solved using the finite volume method structured mesh. The time integration is performed by the fully implicit numerical scheme predictor corrector type of MacCormack. M.Y. Bouzid and R. Dizene [14] Have studied by the two-dimensional numerical simulation of the behavior of compressible flow was highly turbulent through a nozzle of converging-diverging supersonic, with the use of four turbulence models built into the system of Navier Stokes averaged by the statistical method Favre.

Alak Bandyopadhyay and Alok Majumdar [15] described the
verification and validation of a quasi one-dimensional pressure based finite volume algorithm, implemented in Generalized Fluid System Simulation Program (GFSSP), for predicting compressible flow with friction, heat transfer and area change. The numerical predictions were compared with two classical solutions of compressible flow, i.e. Fanno and Rayleigh flow. Fanno flow provides an analytical solution of compressible flow in a long slender pipe where incoming subsonic flow can be choked due to friction. On the other hand, Raleigh flow provides analytical solution of frictionless compressible flow with heat transfer where incoming subsonic flow can be choked at the outlet boundary with heat addition to the control volume. Non uniform grid distribution improves the accuracy of numerical prediction. A benchmark numerical solution of compressible flow in a converging-diverging nozzle with friction and heat transfer has been developed to verify GFSSP's numerical predictions. The numerical predictions compare favorably in all cases.

Nozzle Shocks

A shock wave is nothing but a type of propagating disturbance. In general shock waves are defined by an abrupt or nearly discontinuous change in the characteristics of the medium. Moreover, across a shock there must have an extremely rapid rise in pressure, temperature and density of the flow. In Supersonic flows, expansion is achieved through an expansion fan. A shock wave travels through most media at a higher speed than an ordinary wave. Shock waves form when speeds of a gas changes by more than the speed of sound. At the region where this occur sound waves travelling against the flow reach a point where they cannot travel any further upstream and the pressure progressively builds in that region, and a high pressure shock wave rapidly forms. Shock waves are not conventional sound waves; a shock wave takes the form of a very sharp change in the gas properties on the order of a few mean free paths (roughly micrometers at atmospheric conditions) in thickness.[16]

Supersonic nozzle flow separation occurs in CD nozzles at pressure ratios far below their design value that results in shock formation inside the nozzle. In the one-dimensional analysis it is treated that the shock is normal, the flow past the shock stays attached to the wall, and the subsonic flow downstream of the shock expands isentropically to the level of back pressure at the nozzle exit. But in reality the flow detaches from the wall and forms a separation region, subsequently the flow downstream becomes non-uniform and unstable.[19]

MODEL OF TURBULENCE

The mathematical models selected for this work are the standard K-ε model and Spalart-Allmaras model which are based on the Reynolds Averaged Navier-Stoke (RANS) model available in Fluent. K-ε Model: The standard K-ε model is the most widely used transport model. The standard K-ε model is a two-equation model and the two model equations are as follows:

The model equation for the turbulent kinetic energy K is:

Where

\[

t_o = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]

\[
a_j = -2 \frac{C_T}{\sigma} \frac{\partial u_i}{\partial x_j}
\]

\[
P_V = \frac{\mu_k}{\kappa}
\]

GOVERNING EQUATIONS

The flow model considered here is supersonic flow through the nozzle. Air is considered as the working fluid, flow being inviscid. For flow compressible, viscous and perfect supposed, the fundamental equations of flow can be given by following laws: Conservation of masses, the conservation of momentum, and the conservation of energy.

- Conservation of Mass

- Conservation of Momentum

- Conservation of Energy

Where

\[
\tau_o = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]

\[
a_j = -2 \frac{C_T}{\sigma} \frac{\partial u_i}{\partial x_j}
\]

\[
P_V = \frac{\mu_k}{\kappa}
\]
eddy turbulent viscosity [18]. The turbulent dynamic viscosity is computed from $\mu_t = \rho \overline{f_{\nu}}$

Table (1) CFD analysis process

<table>
<thead>
<tr>
<th>No</th>
<th>Steps</th>
<th>Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Problem Statement</td>
<td>Information about the flow and working parameters</td>
</tr>
<tr>
<td>2</td>
<td>Mathematical Model</td>
<td>Nozzle Geometry</td>
</tr>
<tr>
<td>3</td>
<td>Mesh generation</td>
<td>Nodes/cells, time instants</td>
</tr>
<tr>
<td>4</td>
<td>Space discretization</td>
<td>ODE/DAE systems</td>
</tr>
<tr>
<td>5</td>
<td>Time discretization</td>
<td>Algebraic system Ax=b</td>
</tr>
<tr>
<td>6</td>
<td>solver</td>
<td>Discrete function values</td>
</tr>
<tr>
<td>7</td>
<td>CFD software</td>
<td>Implementation, debugging</td>
</tr>
<tr>
<td>8</td>
<td>Simulation run</td>
<td>Parameters, stopping criteria</td>
</tr>
<tr>
<td>9</td>
<td>Post processing</td>
<td>Visualization, analysis of data</td>
</tr>
</tbody>
</table>

GEOMETRIC MODELLING

The 2-D layout of the nozzle studied is illustrated in Figure 2. Converging-diverging nozzle has a non-swirling axisymmetric geometry. High-pressure low velocity gas, which is air, flows through the convergent section in a subsonic condition and contracts in the throat. Then, the low-pressure high velocity air expands in divergent section in supersonic conditions. Therefore, discretized method should be employed and the geometry in the shape of mesh has been developed.

FLOW VISUALIZATION & ANALYSIS

Static pressure is the pressure that is exerted by a fluid. Contours of static pressure in convergent, throat, divergent and exit section are shown. The below Figure reveals the fact that the gas gets expanded in the nozzle exit. The static pressure in the inlet is observed to be 10.1e+07 Pa and we move towards the throat there is a decrease and the value at the throat is found out to be 6.05e+06 Pa. After the throat, there is a sudden increase in the static pressure at the exit which indicates the occurrence of the shock. Then it reduces to a value of 1.19e+06 Pa at the exit section due to expansion of the fluid towards the exit of the nozzle.

Fig(4) Contours of Static Pressure (Pa)

The temperature almost remains a constant from the inlet up to the throat after which it tends to decrease. At the inlet and the throat the temperature is 4.11e+03 K. After the throat, the temperature decreases till the exit. As we move from the centre vertically upwards and downward temperature increased. As we have assumed the combustion property of fluid, the static pressure is directly proportional to the static temperature.

The static temperature decrease corresponding to decrease in static pressure. There is formation of shock hence static temperature increase due to decrease Mach number across the shock.

Fig(5) Contours of Temperature (K)
The regime of the inlet of the nozzle remains almost steady at the level of convergent and the regime is subsonic because the value of the Mach number is strictly less than one. Then it follows by a sharp increase in the vicinity of throat of the nozzle exactly between the first points of tangency at the level of converging and the first point of tangency in the divergent in this zone the regime becomes transonic Mach number, as there was a small sharp decrease in the center nozzle, at the inlet to the zone of the divergent. Parallel flow is observed which is a characteristic of the conical nozzle and its design purpose (for supersonic speed) is also solved. Velocity magnitude near the wall is less due to the viscosity and turbulence.

Fig(6) Contours of Mach number

The density profile takes two different paths, the first path is from the inlet of the nozzle up to its throat, and the density in this portion is almost constant at its maximum value. The second path that begins just after the throat of the nozzle the density thus undergoes a small sharp increase then continues its decrease until the exit of the nozzle because the flow is compressible.

CONCLUSION

After conducting this investigation, it is observed that oblique shocks are formed during flow through the nozzle and the shock is completely eliminated when the divergent angle is increased. The static pressure decrease with increasing divergent angle. This numerical simulation of flow dynamics through a nozzle, using the computer code (FLUENT) by the use the type of turbulence model (k-ω) at high velocity and temperature, which we provides an approach to the behavior of compressible flow in this nozzle and the effects of the geometry of the latter and their parameters on the flow behavior in 2D. Exit Mach number and Mach number ahead of the shock goes on increasing by decreasing the operating pressure ratio. Near the wall, the Mach number is decreasing for all the nozzles. This is due to the viscosity and turbulence in the fluid.

Fig(7) Contours of Mach number

References


