COMPUTATIONAL ANALYSIS OF ISOLATOR FOR DE-LAVAL NOZZLE FOR SUPERSONIC FLOW

K.M. Pandey*, Aditya. P. Singh* Surendra Yadav*
*NIT Silchar, (INDIA) Department of Mechanical Engineering

ABSTRACT

This paper presents solutions of supersonic flow fields in two-dimensional De Laval nozzles with a duct. The phenomenon of sudden expansion of fluid is visualized in most of the day to day activities, which has prompted many of the researchers to explore in this field. The present study is aimed at investigating the sudden expansion of supersonic flow in de Laval nozzle, with 1.74 Mach numbers for various L/D, into a duct. The flow is simulated using a Fluent. The flow parameters, like pressure ratio, area of duct to area of nozzle exit ratio, and the mach no. of the flow at the nozzle exit is defined prior to the simulation. The flow field of an axi-symmetric flow on sudden expansion is accompanied with flow reversal, flow separation and vortex shedding near the nozzle exit region. The nature of flow development is declared as a smooth flow only when the flow gets attached and the flow is streamlined. The suddenly expanded cavity not only causes head losses but also is accompanied by flow oscillations due to phenomenon called vortex shedding.

Keywords: - Supersonic flow, two-dimension, De Laval Nozzle, Mach No., Pressure ratio, flow reversal, stream line, cavity.

I. INTRODUCTION

Swedish engineer of French descent who, in trying to develop a more efficient steam engine, designed a turbine that was turned by jets of steam. The critical component – the one in which heat energy of the hot high-pressure steam from the boiler was converted into kinetic energy – was the nozzle from which the jet blew onto the wheel. De Laval found that the most efficient conversion occurred when the nozzle first narrowed, increasing the speed of the jet to the speed of sound, and then expanded again. Above the speed of sound (but not below it) this expansion caused a further increase in the speed of the jet and led to a very efficient conversion of heat energy to motion.

The theory of air resistance was first proposed by Sir Isaac Newton in 1726. According to him, an aerodynamic force depends on the density and velocity of the fluid, and the shape and the size of the displacing object. Newton’s theory was soon followed by other theoretical solution of fluid motion problems. All these were restricted to flow under idealized conditions, i.e. air was assumed to posses constant density and to move in response to pressure and inertia.

electric power stations and large ships, although they usually have a different design-to make best use of the fast steam jet, de Laval’s turbine had to run at an impractically high speed. But for rockets the de Laval nozzle was just what was needed. But now with the advancement of computational facility and computational tools available in present day it becomes easier to simulate the flow field in a mere personal computer (PC). There are various dedicated software now available for flow simulation such as FLUENT, STAR-CD, LS DYNA, PDE TOOL, AFINA, CFX, NUMECA, PHOENICS etc. Out of this FLUENT is the most popular one for its versatility and user friendly.

Since the fluid flow has such an overwhelming impact on industrial life, we need to be able to estimate it effectively. This ability can result from an understanding of the nature of the processes and from a methodology with which to predict them quantitatively. The prediction of fluid flow in a given situation consists of predicting the values of the relevant variables governing the process of interest, and knowing how these quantities would change in response to changes in geometry, flow rate, fluid properties, etc.

Armed with this expertise, the designer of an engineering device is able to choose the optimum design from among a number of alternative possibilities and can ensure the desired performance. Prediction offer economic benefits and contribute to human well-being.

The investigation of flow processes can be done by two main methods namely, experimental, theoretical. Experimental investigation offers the most reliable
information about a physical process. However, there are serious difficulties of measurements in many situations and the measuring instruments are not free from errors. Often such measurement itself interferes significantly with the process being measured, thus making total experimental knowledge of the process impossible to obtain. Theoretical investigation works out the consequence of a mathematical model of the process, which often consists of a set of partial differential equations for the physical quantities of interest. These equations are often of such complexity that if the methods of classical mathematics were to be used for solving them there would be a little hope of predicting many cases of practical interest. Fortunately, the development of numerical methods and the availability of large digital computers allow mathematical model to be solved for many practical problems. The advantage of theoretical investigation over a corresponding experimental investigation is its low cost, remarkable speed, detailed and complete information of the process under different conditions. Even with the remarkable success of numerical solutions, few accept them uncritically without some experimental validation. As in the saying by Albert Einstein, “A theory is something nobody believes except the person proposing the theory and an experiment is something everybody believes except the person doing the experiment”.

II. MATHEMATICAL FORMULATION
Mathematical modeling is usually central to the analysis of engineering systems, which are often very complicated. For a typical fluids system, this complexity arises mainly due to the time dependent, multidimensional nature of the fluid flow and the complex supplementary conditions that govern these systems. In addition, the non-linearity of the flow equations makes the analysis all the more complicated. Consequently, a real system is often simplified to a computational model resembling the original in shape, geometry and other physical characteristics in the gross features, but not in every detail. Thus, by the application of fundamental physical laws, and by incorporating approximations and idealizations, a mathematical model is generally amenable to numerical simulations, which hopefully without involving exorbitant time and effort in computation give an adequate picture of the physics of the system.

PHYSICAL MODEL
The problem being considered is the supersonic flow through de Laval nozzle to numerically simulate the flow.
3. GOVERNING EQUATIONS

The advantage of employing the complete Navier-Stokes equations extends not only to the investigations that can be carried out on a wide range of flight conditions and geometries, but also in the process the location of shock wave, as well as the physical characteristics of the shock layer, can be precisely determined. We begin by describing the three-dimensional forms of the Navier-Stokes equations below. Note that the two-dimensional forms are just simplification of the governing equations in the three dimensions by the omission of the component variables in one of the coordinate directions. Neglecting the presence of body forces and volumetric heating, the three-dimensional Navier-Stokes equations are

\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0 \]  

\[ \frac{\partial \rho u}{\partial t} + \frac{\partial (\rho u^2)}{\partial x} + \frac{\partial (\rho uv)}{\partial y} + \frac{\partial (\rho uw)}{\partial z} = - \frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left( \tau_{xx} \right) + \frac{\partial}{\partial y} \left( \tau_{xy} \right) + \frac{\partial}{\partial z} \left( \tau_{xz} \right) \]  

\[ \frac{\partial \rho v}{\partial t} + \frac{\partial (\rho vu)}{\partial x} + \frac{\partial (\rho vv)}{\partial y} + \frac{\partial (\rho vw)}{\partial z} = - \frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left( \tau_{yx} \right) + \frac{\partial}{\partial y} \left( \tau_{yy} \right) + \frac{\partial}{\partial z} \left( \tau_{yz} \right) \]  

\[ \frac{\partial \rho w}{\partial t} + \frac{\partial (\rho uw)}{\partial x} + \frac{\partial (\rho vw)}{\partial y} + \frac{\partial (\rho ww)}{\partial z} = - \frac{\partial p}{\partial z} + \frac{\partial}{\partial x} \left( \tau_{zx} \right) + \frac{\partial}{\partial y} \left( \tau_{zy} \right) + \frac{\partial}{\partial z} \left( \tau_{zz} \right) \]  

\[ \frac{\partial e}{\partial t} + \frac{\partial (e u)}{\partial x} + \frac{\partial (e v)}{\partial y} + \frac{\partial (e w)}{\partial z} = \frac{\partial}{\partial x} \left( \tau_{xx} \right) + \frac{\partial}{\partial y} \left( \tau_{yx} \right) + \frac{\partial}{\partial z} \left( \tau_{xz} \right) \]  

Assuming a Newtonian fluid, the normal stress \( \sigma_{xx}, \sigma_{yy}, \) and \( \sigma_{zz} \) can be taken as combination of the pressure \( p \) and the normal viscous stress components \( \tau_{xx}, \tau_{yy}, \) and \( \tau_{zz} \) while the remaining components are the tangential viscous stress components whereby \( \tau_{xy} = \tau_{yx}, \tau_{yz} = \tau_{zy}, \) and \( \tau_{zx} = \tau_{xz}. \) For the energy conservation for supersonic flows, the specific energy \( E \) is solved instead of the usual thermal energy \( H \) applied in sub-sonic flow problems. In three dimensions, the specific energy \( E \) is repeated below for convenience:

\[ E = e + \frac{1}{2} \left( u^2 + v^2 + w^2 \right) \]  

It is evident from above that the kinetic energy term contributes greatly to the conservation of energy because of the high velocities that can be attained for flows, where \( Ma > 1. \) Equations (1)-(6) represent the form of governing equations that are adopted for compressible flows.

The solution to the above governing equations nonetheless requires additional equations to close the system. First, the equation of state on the assumption of a perfect gas in employed, that is,

\[ p = \rho RT, \]

where \( R \) is the gas constant.

Second, assuming that the air is calorically perfect, the following relation holds for the internal energy:

\[ e = C_v T, \]

where \( C_v \) is the specific heat of constant volume. Third, if the Prandtl number is assumed constant (approximately 0.71 for calorically perfect air), the thermal conductivity can be evaluated by the following:

\[ k = \frac{\mu C_p}{Pr} \]

The Sutherland’s law is typically used to evaluate viscosity \( \mu, \) which is provided by

\[ \mu = \mu_0 \left( \frac{T}{T_0} \right)^{1.5} \frac{T_0 + 120}{T + 120} \]

where \( \mu_0 \) and \( T_0 \) are reference values at standard sea level conditions.

Generalized form of Turbulence Equations is as follows:

\[ (k) \frac{\partial k}{\partial t} + \frac{\partial (k u)}{\partial x} + \frac{\partial (k v)}{\partial y} + \frac{\partial (k w)}{\partial z} = \frac{\partial}{\partial x} \left( \tau_{kkx} \right) + \frac{\partial}{\partial y} \left( \tau_{kky} \right) + \frac{\partial}{\partial z} \left( \tau_{kkz} \right) \]

\[ + \frac{\partial}{\partial x} \left( \tau_{kkx} \right) + \frac{\partial}{\partial y} \left( \tau_{kky} \right) + \frac{\partial}{\partial z} \left( \tau_{kkz} \right) + (S_k = p - D) \]
\[
\frac{\partial}{\partial t} \left( \epsilon \right) + \frac{\partial (\epsilon u)}{\partial x} + \frac{\partial (\epsilon v)}{\partial y} + \frac{\partial (\epsilon w)}{\partial z} = \frac{\partial}{\partial y} \left[ \frac{\nu_T \partial \epsilon}{\sigma_k \frac{\partial y}{\partial y}} \right] + \frac{\partial}{\partial z} \left[ \frac{\nu_T \partial \epsilon}{\sigma_k \frac{\partial z}{\partial z}} \right] + (S_\epsilon = \frac{\epsilon}{k} (C_{\epsilon 1} P - C_{\epsilon 2} D))
\]

where

\[P = 2 \nu_T \left[ \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial w}{\partial z} \right)^2 \right] + \nu_T \left[ \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial x} + \frac{\partial w}{\partial z} \right)^2 \right] \text{ and } D = \epsilon\]

III. GEOMETRY AND GRID ARRANGEMENT

A 2d axi-symmetric computational domain was considered, the initial design parameters for de Laval nozzle\[\] and isolator for Mach number 3. This was obtained by method of characteristics of nozzle program and trial and error method for isolator. Here the grid arrangement for both nozzle and isolator at Mach 3 is given below. And the length of the isolator from throat is 2m.

Figure 2: De-Laval Nozzle for Mach 3

Figure 3: De-Laval Nozzle for With Isolator

IV. RESULT AND DISCUSSION

Mach number 3:-

Static pressure:

Static pressure is the pressure that is exerted by a fluid. Specifically, it is the pressure measured when the fluid is still, or at rest. The above figure reveals the fact that the gas gets over expanded at the nozzle exit. The value of static pressure at nozzle exit is 9.17e+03 Pa. The static pressure in the nozzle falls. The static pressure in the convergent section is observed to be 3.22e+05 Pa at the inlet which then decreases to 1.66e+05 Pa up to the throat and then to 9.17e+03 Pa at exit.

Total pressure

From the figure it is observed that, near the throat just below the wall, the total pressure increases up to 3.68e+05 Pa in a patch which is like a disc, while the total pressure at the nozzle exit at the centre is 3.52e+05 Pa and
that at the inlet is 3.68e+05Pa. You can easily visualize in the above figure that, there is decrease in stagnation pressure near the nozzle walls due to viscous effects, whereas the stagnation pressure remains almost constant in the centre.

**Velocity Magnitude:**

From the figure, it is clear that the flow is symmetric and flowing across an angle as a characteristic of the De-Laval nozzle. At the inlet of the nozzle, the velocity is found to be 2.21e+02 m/s and it rapidly increases up to the throat to nearly 4.85e+02 m/s. At the nozzle exit, the velocity is found to be around 8.82e+02 m/s. The flow is turbulent, so near the wall flow separation takes place causing the velocity to decreases to nearly 8.82e+01 m/s while at the centre the velocity is 7.94e+02 m/s.

**Turbulence Intensity:**

The above figure reveals the fact that the turbulence intensity is very low in the convergent section and up to the throat it is 8.39e+02% while at the centre of the nozzle the turbulent intensity is maximum up to 5.65e+03%. The flow is totally symmetric. Due to friction, near the wall, the turbulent intensity is quite higher than that at the nozzle exit. In the divergent section, since there is stabilization of flow, there is a decrease in the turbulent intensity.

**Mach number 3 with isolator:**

In this, the Isolator is designed to reduce the Mach number. From figure, it is clearly visualized that in the convergent section at inlet point, Mach number, is in the Sub-sonic region while at the throat, flow becomes Sonic and at the nozzle exit it becomes Super-Sonic for which the nozzle is designed. Near the wall, the Mach number is 1.98 because of the viscosity and turbulence in the fluid. The flow travels in the nozzle along the angle of direction. When the flow comes in the isolator parts. The Mach number starts decreasing as well as the property of flow has been also changing as can observe from figure. At the exit of the isolator the Mach number is 1.97.

**Static pressure:**

The above figure reveals the fact that the gas gets over expanded at the nozzle exit. The value of static pressure at nozzle exit is 4.87e+03 Pa. The static pressure in the convergent section is observed to be 7.85e+05Pa at the inlet which then decreases to 2.00e+05Pa up to the throat and then to 4.87e+03Pa at exit. From the above observations, it is clear that static pressure reduces in the nozzle as we move from inlet to exit and it starts again increasing as the flow goes into the isolator and throughout it is constant.
Total Pressure

![Total Pressure Image](image1)

The total pressure at the nozzle exit at the centre is 2.72e+05 Pa and at the inlet is 3.68e+05 Pa as given in the boundary condition. The contour of nozzle and nozzle with isolator like disc patch is same in both figure. There is a decrease in stagnation pressure near the nozzle walls due to viscous effects, whereas the stagnation pressure remains almost constant in the centre which is the phenomenon observed in Mach 3 as well. In Mach 3 there was a disc like patch near the wall but here it extends till the exit of the nozzle. Overall a major change is not observed in the total pressure in the nozzle. While in the isolator there is sudden variation in the total pressure and near the wall the stagnation pressure is around 1.29 bar.

Velocity Magnitude:

![Velocity Magnitude Image](image2)

As noted from the above figure, at the inlet of the nozzle, the velocity is found to be 2.20e+02 m/s and it rapidly increases up to the throat to nearly 5.21e+02 m/s. At the nozzle exit, the velocity is found to be around 8.63e+02 m/s, as in case of Mach 3. when the flow comes in isolator the flow characteristics starts changing and the velocity at the exit is nearly 6.5e+02 m/s. It is clear that the flow is symmetric and flowing across an angle as a characteristic of the De-Laval nozzle.

Turbulence Intensity:

![Turbulence Intensity Image](image3)

The above figure reveals the fact that the turbulence intensity is very low in the convergent section and up to the throat it is 3.51e+02% while at the centre of the nozzle the turbulent intensity is maximum up to 5.31e+03%. This is due to the sudden expansion in area which causes the turbulence just after the throat. Due to friction, near the wall, the turbulent intensity is quite higher than that at the nozzle exit. In the divergent section, as we move towards the exit of isolator there is rapidly a change, since there is stabilization of flow, there is a decrease in the turbulent intensity. The flow is totally symmetric.

V. GRID INDEPENDENCE TEST

The grid independence test has been done for Mach number 3.

Mach Number 3 De-Laval Nozzle:

<table>
<thead>
<tr>
<th>grid size (original / adapted / change)</th>
</tr>
</thead>
<tbody>
<tr>
<td>cells (14400 / 17934 / 3534)</td>
</tr>
<tr>
<td>faces (29100 / 36504 / 7404)</td>
</tr>
<tr>
<td>nodes (14701 / 18571 / 3870)</td>
</tr>
</tbody>
</table>

The grid adaption method is done in respect of Mach number. The changes in cell, faces and nodes are 3534, 7404 and 3870 respectively. As shown in the figure. The solution is converging after 919 iteration.

The mass flow rate plot respect to iteration is shown in figure and the mach contour after grid adaption is shown below.
It is observed that there is no change in Mach number. So the number of nodes as given in previous was accurate.

RESULT VALIDATION WITH MATHEMATICAL MODEL

In this section, the numerical Analysis of nozzle at mach 3 has been done on the basis of table 1. This is based on theoretical formulation.

ACKNOWLEDGEMENT

The authors acknowledge the valuable suggestions from Prof. E. Rathakrishnan, Professor, Department of aerospace engineering, IIT Kanpur, India. The authors acknowledge the financial help provided by AICTE from the project AICTE: 8023/RID/BOIII/NCP(21) 2007-2008. The Project id at IIT Guwahati is ME/P/USD/4.

VI. CONCLUSION:

It is observed that the nozzle which designed for, flow travel along with the direction and at throat Mach number is 1 in both nozzles. The result is validating with mathematical procedure of method of characteristics. For Mach 3 the total pressure increases up to 3.68e+05Pa in a patch which is like a disc, while the total pressure at the nozzle exit at the centre is 3.52e+05Pa and that at the inlet is 3.68e+05Pa. Turbulence intensity is very low in the convergent section and up to the throat it is 3.51e+02% while at the centre of the nozzle the turbulent intensity is maximum up to 5.31e+03%. This is due to the sudden expansion in area which causes the turbulence just after the throat. Due to friction, near the wall, the turbulent intensity is quite higher than that at the nozzle exit. In the divergent section, as we move towards the exit of isolator there is rapidly a change, since there is stabilization of flow, there is a decrease in the turbulent intensity. The flow is totally symmetric.

VII. REFERENCE

[23] K.M. Pandey and A.P Singh, “Design and development of De-Laval Nozzle at Mach 3 and 4 with fluent software”. Accepted and Published in 30 June 2010 IJME.