Critical Review Paper of Modelling of Compressible Nozzle Flow through Variable Area Duct

Snehil Varghese[1], Dushyant Kalihari[2], Brijesh Patel[3]
[1]PG Scholar, Department of Mechanical Engineering
[2]Asst. Professor, Department of Mechanical Engineering
[3]Asst. Professor, Department of Mechanical Engineering
MATS University, Raipur (C.G.)

***Abstract*—This paper deals with review of all the research papers related to modelling of compressible nozzle flow and analysis of heat transfer, frictional and isentropic flow through variable area duct.**
In this paper, we will collect all the information related to above topic and then find out their problems.

***Keywords:* Nozzle, duct**

I. INTRODUCTION

Every thermal system (TS) consists of different types of elements among which are heat exchangers, compressors, turbines, ducts. Most commercial network flow analysis codes lack the capability to simulate one dimensional flow in a rocket engine nozzle. Simulation of one-dimensional flow in rocket nozzle requires a numerical algorithm capable of modeling compressible flow with friction, heat transfer, variable cross-sectional area and chemical reaction. One of the primary requirements of compressible flow simulation is to accurately model the inertia force which is often neglected in many network flow analysis codes. Generalized Fluid System Simulation Program (GFSSP) which is widely used for the design of Main Propulsion System of Launch Vehicle and secondary flow analysis of turbopump. GFSSP employs a finite volume formulation of mass, momentum and energy conservation equations in a network consisting of nodes and branches . Energy conservation equations are expressed in terms of entropy with entropy generation due to viscous dissipation. Mass and energy conservation equations are solved for pressures and entropies at the nodes while the momentum conservation equation is solved at the branches to calculate flow rate.

Large scale launch vehicle requires nozzle which can produce maximum specific impulse and thrust with reduced nozzle length. Until now, various supersonic nozzles such as thrust-optimized contour (TOC), compressed truncated perfect (CTP) contours have been developed to meet such demands. The coupling of heat transfer at a solid/fluid interface is known as conjugate heat transfer (CHT). CHT problems are commonly found in real-world applications such as turbo-machinery, reentry vehicles, laser irradiation applications, heating ducts and more. In order to properly simulate a conjugate heat transfer problem, a code needs to be able to accurately model the convective heat transfer in the fluid and the conductive heat transfer in the solid.

II. LITERATURE SURVEY

Different relevant studied papers are shown as below:

In 2006, Eric C. Marineau and Joseph A. Schetz focused on convective heat transfer with complex wall temperature variations, as well as conjugate heat transfer problems involving solids are performed. Simulations are made using the GASP Computational Fluid Dynamics (CFD) code, which solves the Reynolds-Averaged Navier-Stokes equations (RANS). GASP has been modified by adding a solid heat conduction solver, enabling the coupling of RANS with the heat equation for an isotropic solid. Validation cases involving convective heat transfer are considered.

In 2007, Alak Bandyopadhyay and Alok Majumdar 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.

In 2010, Vincent Lijo 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.

In 2010, Mykhaylo Fedorov 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.
The mathematical properties of the solution are analyzed in order to calculate this type of compressible flow. For the purpose of this analysis, the functions of the solution and duct performance characteristics of the flow model are demonstrated for various flows and heat flux values, as well as distributions of flow parameters along the duct for both subsonic and supersonic flows. The thermodynamic constraints of the solution are also studied. The analytical solution formulas obtained in this paper may serve as a definition of heat flux and friction factor in ducts from a viewpoint of one-dimensional gas dynamics.

III. APPROXIMATION OF EXPERIMENTAL SET UP

Figure 1 shows the schematic view of four melt delivery nozzles under consideration in this study, each with differing melt tip length, but the same internal profile. In this study, high-speed gas flow through each of the three nozzles has been generated by commercial CFD code, ANSYS Fluent 13.

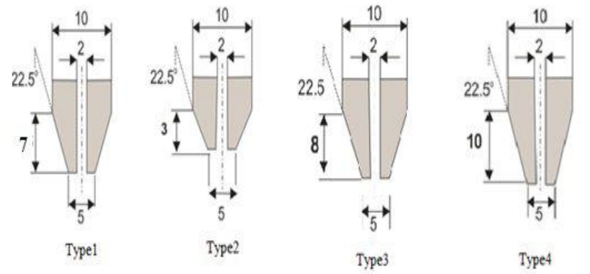

FIGURE 1 THE SCHEMATIC VIEW OF FOUR MELT DELIVERY NOZZLES

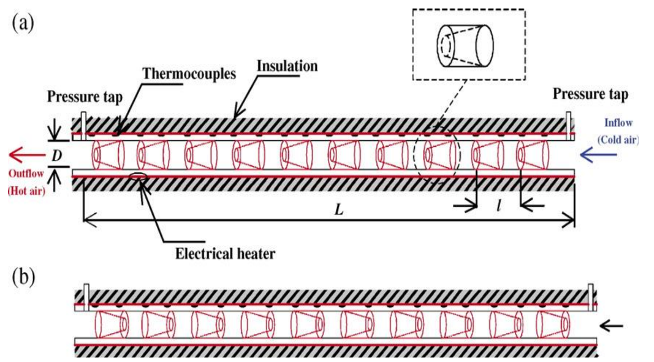

FIG. 2 TEST TUBE FITTED WITH THE NOZZLE-TURBULATORS, (A) C-NOZZLE AND (B) D-NOZZLE ARRANGEMENTS.

Experimental procedure

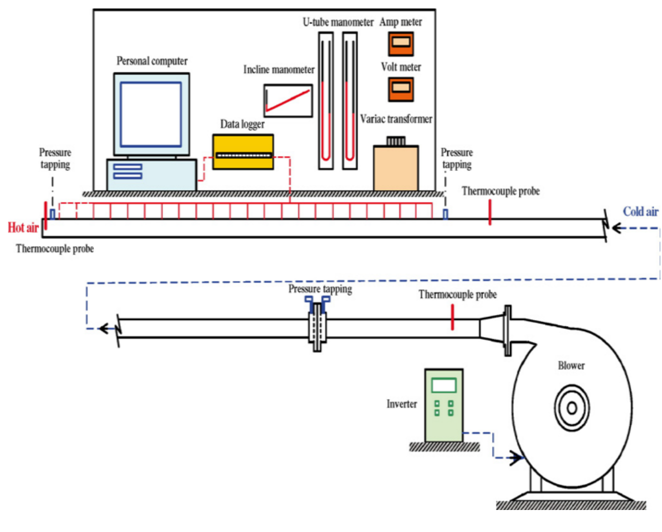

Fig. 3. Schematic diagram of experimental heat transfer set-up.

In the apparatus setting above, the inlet bulk air at 25 °C from a 7.5kW blower was directed through an orifice meter and passed to the heat transfer test section. The air flow rate was measured by the orifice meter, built according to ASME standard. Manometric fluid was used in U-tube manometers with specific gravity (SG.) of 0.826 to ensure reasonably accurate measurement of the low pressure drop encountered at low Reynolds numbers. Also, the pressure drop of the heat transfer test tube was measured with an inclined U-tube manometer. The volumetric air flow rates from the blower were adjusted by varying motor speed through the inverter and measured by the orifice meter situated upstream of the test tube.

During the experiments, the bulk air was heated by an adjustable electrical heater wrapping along the test section. Both the inlet and outlet temperatures of the bulk air from the tube were measured by multi-channel RTD PT 100 type temperature sensors, calibrated within ±0.1C deviation by thermostat before being used. It was necessary to measure the temperature at 15 stations altogether on the outer surface of the heat transfer test pipe for finding out the average Nusselt number.

In each test run, it was necessary to record the data of temperature, volumetric flow rate and pressure drop of the bulk air at steady state conditions in which the inlet air temperature was maintained at 25 °C. The Reynolds number of the bulk air was varied from 8000 to 18,000. The various characteristics of the flow, the Nusselt number, and the Reynolds numbers were based on the average of tube wall temperatures and of inlet and outlet air temperatures. The local wall temperatures, inlet and outlet air temperatures, the pressure drop across the test section and air flow velocity were measured and recorded to evaluate and analyze the heat transfer rate of the heated tube. The average Nusselt numbers were calculated and discussed where all fluid properties were determined at the overall bulk mean temperature.

Model assumptions

In order to simplify the numerical calculations and establish the numerical model of gas flow pattern, the following assumptions have been made as below:

1- Flow is considered to be steady-state.
2- The effects of gravity are neglected.
3- Flow is 2D axis-symmetric.
4- Flow is considered as air and modelled as a compressible ideal gas.
5- The impact of the molten metal is not considered.

Domain and Meshing

A detailed study of both the domain size and boundary locations were undertaken to identify the effect of inlet /outlet moving boundaries and the size of computational domain. The final geometry was constructed and domain independence was demonstrated. A series of high-quality meshes were developed with increasing spatial resolution to evaluate the effect of that on the CFD results. These contained 15000, 22500 and 30762 elements, respectively. It was established that the results where the mesh containing 30762 elements were acceptably mesh independent and these were subsequently used for the simulations reported here. The domain size and a sample of mesh independent can be seen in Figure.

Boundary Conditions

Atomization gas pressures of 0.5, 1, 1.5, 2, 2.5,3, and 4MPa were considered for the pressure inlet boundary condition to the nozzle gas chamber, these pressures being based on commercial operating practise for gas atomisation. The outlet of the domain (down stream of the melt nozzle) was taken as a pressure condition at atmospheric pressure. The outer boundaries for the gas chamber, melt delivery nozzle and the gas die were considered as a wall with no-slip velocity condition. For the boundary labelled 'Upper domain boundary' in figure 4, two boundary models were considered, a wall with no-slip condition and an atmospheric pressure outlet. Calculations have been undertaken for both above a part of the domain independence test. It was noted that the results were comparable for both boundary conditions and, as such, the wall boundary was used since this provided more consistent model convergence. In addition, for the thermal boundary conditions, for upper boundary flow inlet and outlet, the gas temperature was set at 300K.

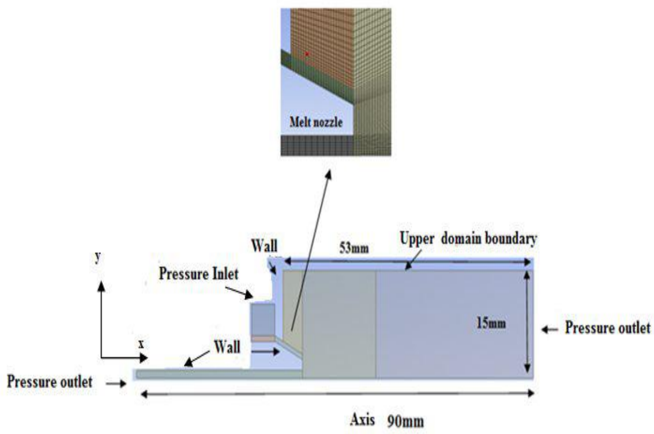

Figure 4 Computational domain and example of the mesh close to the melt nozzle tip for nozzle type1

IV. CONCLUSIONS

We concluded that all the research paper studied and found that it was interesting topic in which we can satisfactorily work and we can use the software for modelling and simulation and then analysis for compressible nozzle flow through the variable duct.

V. REFERENCES

[1] G.H. Junkhan, A.E. Bergles, V. Nirmalan, T. Ravigururajan, Investigation of turbulators for fire tube boilers, ASME Journal Heat Transfer 107(1985)354–360.
[2] ASME Standard, Measurement of Fluid Flow in Pipes Using Orifice, Nozzle and Venturi, ASME MFC-3M-1984, United Engineering Center 345East 47th Street, New York, 1984.
[3] T.Ayhan,Y. Azak, C.Demirtas,B.Ayhan, Numerical and experimental investigation of enhancement of turbulent flow heat transfer in tubes by means of truncated hollow cone inserts, Heat Transfer Enhancement of Heat Exchangers, Kluwer Academic Publishers, 1999, pp. 347–356.
[4] Shahed Motaman, Use of Computational Modelling For Investigation the Effect of Melt Delivery Nozzle Tip Length on Gas Flow Separation in Supersonic Gas Atomization, 12th Triennial International Conference on Liquid Atomization and Spray Systems, Heidelberg, Germany, September 2-6, 2012.
[5] R.Unal, Investigation of the metal powder production efficiency of a new convergent-divergent nozzle in close-coupled gas atomization, Powder Metal: 302-306 (2007).

[6] Alak Bandyopadhyay & Alok Majumdar, Modeling of Compressible Flow with Friction and Heat Transfer using the Generalized Fluid System Simulation Program (GFSSP), Paper submitted to Thermal Fluid Analysis Workshop (TFAWS) to be hosted by NASA Glenn Research Center and Corporate College, Cleveland, September 10-14, 2007.
[7] Mykhaylo Fedorov, Analytical study of steady-state compressible flow of perfect gas with constant heat flux and friction in constant area ducts, International Journal of Thermal Sciences 49 (2010) 1277- 1287.
[8] Vincent Lijo, Numerical simulation of transient flows in a rocket propulsion nozzle, International Journal of Heat and Fluid Flow 31 (2010) 409–417.