

MODELING AND CFD ANALYSIS ON JET NOZZLES

¹A. Chinnamahammad bhasha, ²N. VijayaRami Reddy, ³B. Rajnaveen

¹pursuing M.Tech, Dept of ME, Vignan University, India.

²Asst. prof, Dept of ME, Mahatma Gandhi Institute of Technology, Hyd, India.

³Asst. prof, Dept of ME, Mahatma Gandhi Institute of Technology, Hyd, India.

Abstract:

Supersonic flows associated with missiles, aircraft, missile engine intake and rocket nozzles are often steady. The shape of the nozzle geometry is increasingly attractive in heating, ventilation and air conditioning applications. The objective of the present work is to simulate and understand Supersonic flows with single jet Flow at various Mach numbers. The purpose is 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 (1.74, 2, and 3), pressure ratio and dimensionless spacing single jet 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 premium and CFD analysis is carried out in solid works flow simulation.

Keywords — jet nozzles, Supersonic flows, fluid dynamics, solid works, CFD analysis.

I. INTRODUCTION:

The possibility of using a reaction jet had interested aircraft designers for a long time, but initially the low speeds of early aircraft and the unsuitability of a piston engine for producing the large high velocity airflow necessary for the jet presented many obstacles. A French engineer, René Lorin, patented a jet propulsion engine (fig. 1-1) in 1913, but this was an anodyne (Para. 11) and was at that period impossible to manufacture or use, since suitable heat resisting materials had not then been developed and, in the second place, jet propulsion would have been extremely inefficient at the low speeds of the aircraft of those days. However, today the modern ram jet is very similar to Lorin's conception. In 1930 Frank Whittle was granted his first patent for using a gas turbine to produce a propulsive jet,

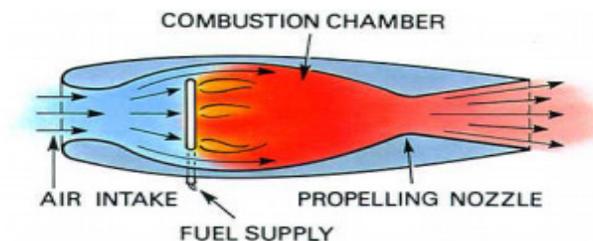


Fig. 1-1 Lorin's jet engine.

but it was eleven years before his engine completed its first flight. The Whittle engine formed the basis of the modern gas turbine engine, and from it was developed the Rolls-Royce Welland, Derwent, Nene and Dart engines. The Derwent and

Nene turbo-jet engines had world-wide military applications; the Dart turbo-propeller engine became world famous as the power plant for the Vickers Viscount aircraft. Although other aircraft may be fitted; with later engines termed twin-spool, triple-spool, by-pass, ducted fan, inducted fan and prop fan, these are inevitable developments of Whittle's early engine.

II. LITERATURE REVIEW

Jet engine nozzle is designed for attaining speeds that are greater than speed of sound. The design of this nozzle came from the area-velocity relation $(dA/dV) = (A/V) (1-M^2)$ M is the Mach number (which means ratio of local speed of flow to the local speed of sound) A is area and V is velocity the following information can be derived from the area-velocity relation

- I. For incompressible flow limit, i.e. for M tends to zero, $AV = \text{constant}$. This is the famous volume conservation equation or continuity equation for incompressible flow.
- II. For $M < 1$, a decrease in area results in increase of velocity and vice versa. Therefore, the velocity increases in a convergent duct and decreases in a Divergent duct. This result for compressible subsonic flows is the same as that for incompressible flow.
- III. For $M > 1$, an increase in area results in increases of velocity and vice versa, i.e. the velocity increases in a divergent duct and decreases in a convergent duct. This is directly opposite to the behavior of subsonic flow in divergent and convergent ducts.

IV. For $M = 1$, $dA/A = 0$, which implies that the location where the Mach number is unity, the area of the passage is either minimum or maximum. We can easily show that the minimum in area is the only physically realistic solution.

One important point is that to attain supersonic speeds we have to maintain favorable pressure ratios across the nozzle. One example is attain just sonic speeds at the throat, pressure ratio to e maintained is $(P_{throat} / P_{inlet}) = 0.528$.

Table: 1

| Regime Mach | Subsonic | Transonic | Sonic | Supersonic | Hypersonic | High-hypersonic |
|-------------|----------|-----------|-------|------------|------------|-----------------|
| | <1.0 | 0.8-1.2 | 1.0 | 1.0-5.0 | 5.0-10.0 | >10.0 |

From table.1 at transonic speeds, the flow field around the object includes both sub- and supersonic parts. The transonic period begins when first zones of $M > 1$ flow appear around the object. In case of an airfoil (such as an aircraft's wing), this typically happens above the wing. Supersonic flow can decelerate back to subsonic only in a normal shock; this typically happens before the trailing edge. As the speed increases, the zone of $M > 1$ flow increases towards both leading and trailing edges. As $M=1$ is reached and passed, the normal shock reaches the trailing edge and becomes a weak oblique shock: the flow decelerates over the shock, but remains supersonic. A normal shock is created ahead of the object, and the only subsonic zone in the flow field is a small area around the object's leading edge.

The governing continuity, momentum, and energy equations for this quasi one-dimensional, steady, isentropic flow can be expressed, respectively.

III. METHODS OF JET PROPULSION

The types of jet engine, whether ram jet, pulse jet, rocket, gas turbine, turbo/ram jet or turbo-rocket, differ only in the way in which the 'thrust provider', or engine, supplies and converts the energy into power for flight. The ram jet engine (fig. 1-6) is an athodyd, or 'aero-thermodynamic-duct' to give it its full name. It has no major rotating parts and consists of a duct with a divergent entry and a convergent or Convergent-divergent exit.

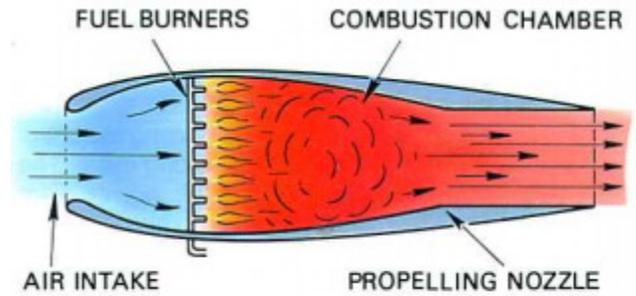


Fig. 1-6 A ram Jet engine.

When forward motion is imparted to it from an external source, air is forced into the air intake where it loses velocity or kinetic energy and increases its pressure energy as it passes through the diverging duct. The total energy is then increased by the combustion of fuel, and the expanding gases accelerate to atmosphere through the outlet duct. A ram jet is often the power plant for missiles and target vehicles; but is unsuitable as an aircraft power plant "because it requires forward motion imparting to it before any thrust is produced.

The pulse jet engine (fig. 1-7) uses the principle of intermittent combustion and unlike the ram jet it can be run at a static condition. The engine is formed by an aerodynamic duct similar to the ram jet but, due to the higher pressures involved; it is of more robust construction. The duct inlet has a series of inlet 'valves' that are spring-loaded into the open position. Air drawn through the open valves passes into the combustion chamber and is heated by the burning of fuel injected into the chamber. The resulting expansion causes a rise in pressure, forcing.

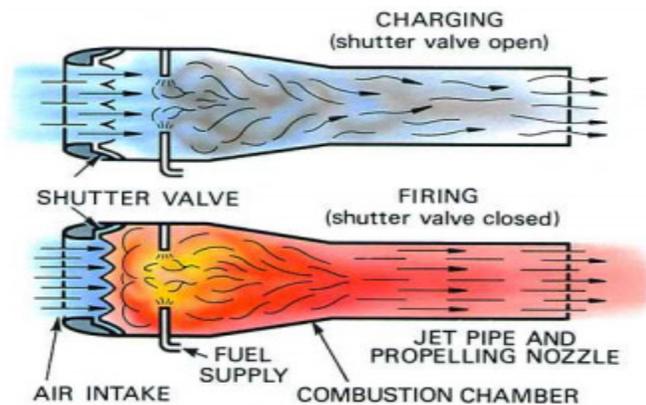


Fig. 1-7 A pulse jet engine.

The valves to close and the expanding gases are then ejected rearwards. A depression created by the exhausting gases allows the valves to open and repeat the cycle. Pulse jets have been designed for helicopter rotor propulsion and some dispense with inlet valves by careful

design of the ducting to control the changing pressures of the resonating cycle. The pulse jet is unsuitable as an aircraft power plant because it has high fuel consumption and is unable to equal the performance of the modern gas turbine engine.

The mechanical arrangement of the gas turbine engine is simple, for it consists of only two main rotating parts, a compressor and a turbine, and one or a number of combustion chambers. This simplicity, however, does not apply to all aspects of the engine, for as described in subsequent Parts the thermo and aerodynamic problems are somewhat complex. They result from the high operating temperatures of the combustion chamber and turbine, the effects of varying flows across the compressor.

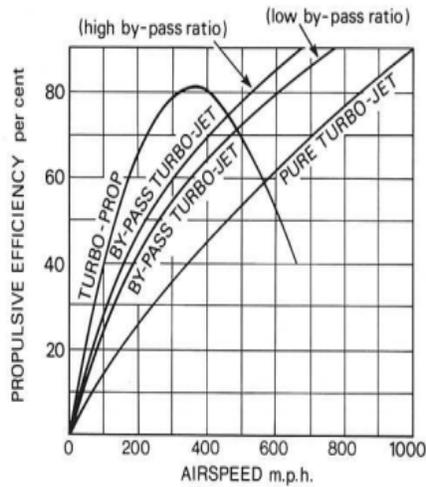


Fig. 2.1: Effects of varying flows

IV.WORKING CYCLE

The working cycle of the gas turbine engine is similar to that of the four-stroke piston engine. However, in the gas turbine engine, combustion occurs at a constant pressure, whereas in the piston engine it occurs at a constant volume. Both engine cycles show that in each instance there is induction, compression, combustion and exhaust. These processes are intermittent in the case of the piston engine whilst they occur continuously in the gas turbine. In the piston engine only one stroke is utilized in the production of power, the others being involved in the charging, compressing and exhausting of the working fluid. In contrast, the turbine engine eliminates the three 'idle' strokes, thus enabling more fuel to be burnt in a shorter time; hence it produces a greater power output for a given size of engine. Due to the continuous action of the turbine engine and the fact that the combustion chamber is not an enclosed space, the pressure of the air does not rise, like that of the piston engine, during combustion but its volume does increase. This process is known as heating at constant pressure. Under these conditions there is no peak or

fluctuating pressures to be withstood, as is the case with the piston engine with its peak pressures in excess of 1,000 lb. per sq. in. It is these peak pressures which make it necessary for the piston engine to employ cylinders of heavy construction and to use high octane fuels, in contrast to the low octane fuels and the light fabricated combustion chambers used on the turbine engine. The working cycle upon which the gas turbine engine functions is, in its simplest form, represented by the cycle shown on the pressure volume diagram in fig. 2-2. Point A represents air at atmospheric pressure that is compressed along the line AB. From B to C heat is added to the air by introducing and burning fuel at constant pressure, thereby considerably increasing the volume of air. Pressure losses in the combustion chambers are indicated by the drop between B and C. From C to D the gases resulting from combustion expand through the turbine and jet pipe back to atmosphere. During this part of the cycle, some of the energy in the expanding gases is turned into mechanical power by

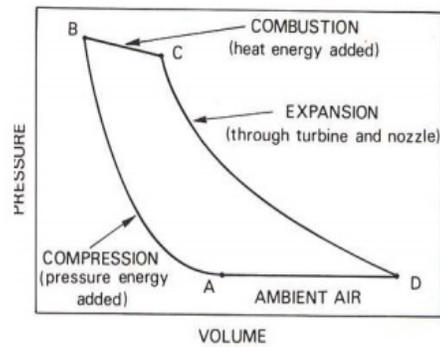
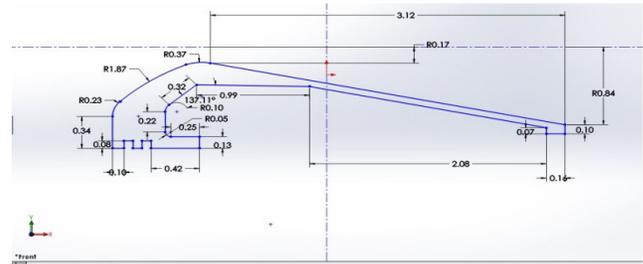
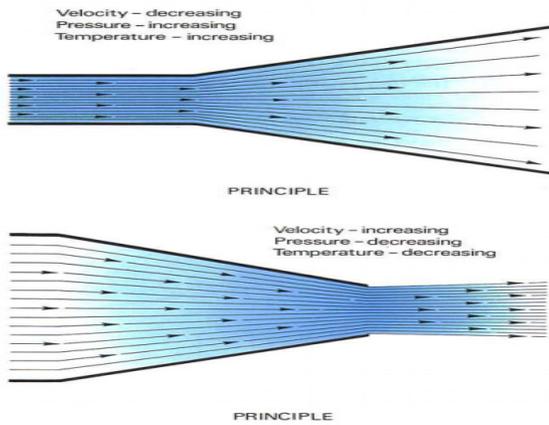


Fig. 2-2 The working cycle on a pressure-volume diagram.

The turbine; the remainder, on its discharge to atmosphere, provides a propulsive jet. Because the turbo-jet engine is a heat engine, the higher the temperature of combustion the greater is the expansion of the gases. The combustion temperature, however, must not exceed a value that gives a turbine gas entry temperature suitable for the design and materials of the turbine assembly. The use of air-cooled blades in the turbine assembly permits a higher gas temperature and a consequently higher thermal efficiency

VI. MODELING OF JET NOZZLES



By revolving the above sketch we get the model of jet nozzle.

VII. SOLIDWORKS FLOW SIMULATION PROJECT:

V. CONSTRUCTION AND MATERIALS

The exhaust system must be capable of withstanding the high gas temperatures and is therefore manufactured from nickel or titanium. It is also necessary to prevent any heat being transferred to the surrounding aircraft structure. This is achieved by passing ventilating air around the jet pipe, or by lagging the section of the exhaust system with an insulating blanket. Each blanket has an inner layer of fibrous insulating material contained by an outer skin of thin stainless steel, which is dimpled to increase its strength. In addition, acoustically absorbent materials are sometimes applied to the exhaust system to reduce engine noise. When the gas temperature is very high (for example, when afterburning is employed), the complete jet pipe is usually of double-wall construction with an annular space between the two walls. The hot gases leaving the propelling nozzle induce, by ejector action, a flow of air through the annular space of the engine nacelle. This flow of air cools the inner wall of the jet pipe and acts as an insulating blanket by reducing the transfer of heat from the inner to the outer wall. The cone and streamline fairings in the exhaust unit are subjected to the pressure of the exhaust gases; therefore, to prevent any distortion, vent holes are provided to obtain a pressure balance. The mixer unit used in low by-pass ratio engines consists of a number of chutes through which the bypass air flows into the exhaust gases. A bonded honeycomb structure is used for the integrated nozzle assembly of high by-pass ratio engines to give lightweight strength to this large component. Due to the wide variations of temperature to which the exhaust system is subjected, it must be mounted and have its sections joined together in such a manner as to allow for expansion and contraction without distortion or damage.

A fluid flow analysis using Flow Simulation involves a number of basic steps that are shown in the following flowchart in figure. The process of setting up a Flow Simulation project includes the following general setting steps in order: choosing the analysis type, selecting a fluid and a solid and settings of wall condition and initial and ambient conditions. Any fluid flow problem that is solved using Flow Simulation must be categorized as either internal bounded or external unbounded flow. Examples of internal flows include flows bounded by walls such as pipe- and channel flows, heat exchangers and obstruction flow meters. External flow examples include flows around airfoils and fuselages of airplanes and fluid flow related to different sports such as flows over golf balls, baseballs and soccer balls. Furthermore, during the project setup process a fluid is chosen as belonging to one of the following six categories: gas, liquid, non-Newtonian liquid, compressible liquid, real gas or steam. Physical features that can be taken into account include heat conduction in solids, radiation, time-varying flows, gravity and rotation. Roughness of surfaces can be specified and different thermal conditions for walls can be chosen including adiabatic walls or specified heat flux, heat transfer rate or wall temperature.

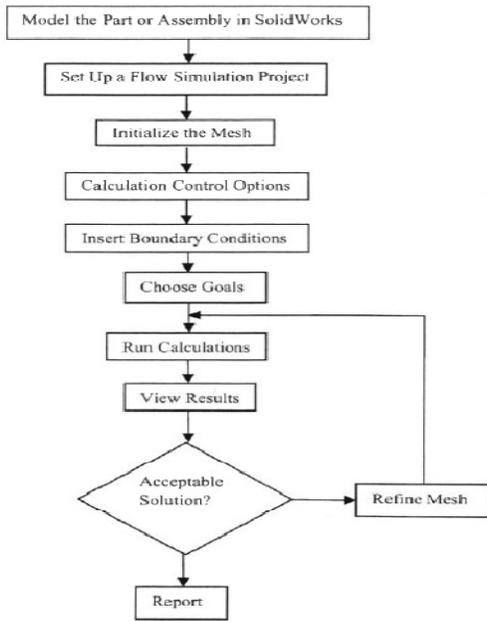


Figure 3: Flowchart for fluid flow analysis using Solid works Flow Simulation

VIII.MESHING IN SOLID WORKS FLOW SIMULATION:

Mesh consists of cells in the form of rectangular parallelepipeds. The Flow Simulation mesh can contain basic cells of three different types: fluid cells, partial cells and solid cells see figure below. Basic cells can be split during the process of refinement. During refinement, each basic cell is split in eight smaller cells with the same volume. Therefore, the volume of each refined cell is only 1/8 of the original volume. The computational mesh is recommended to be constructed in the following order:

- a) Start by using the automatic mesh setting. Set the minimum gap size and minimum wall thickness to appropriate values.
- b) Turn off automatic settings and set your own basic mesh values with both the small solid features refinement level and the curvature refinement level set to zero. Disable the narrow channel refinement.
- c) Increase both the small solid features refinement level and the curvature refinement level in steps and enable the narrow channel refinement.

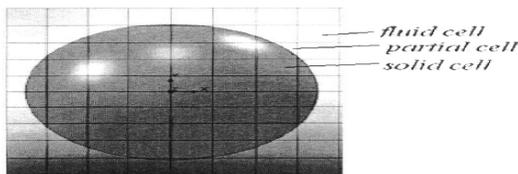


Figure 4: Different types of mesh shells

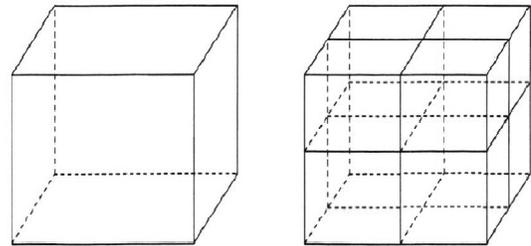
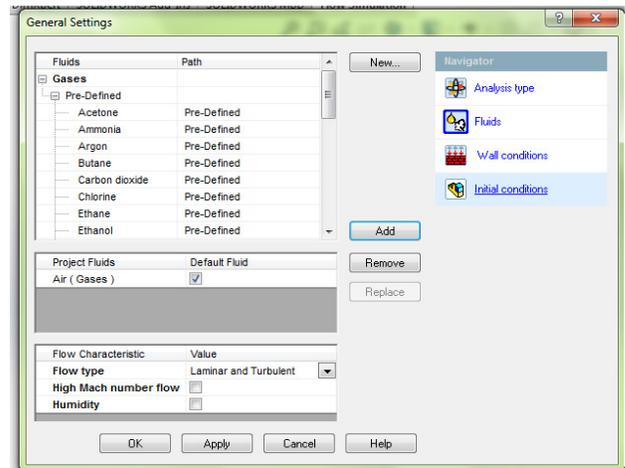


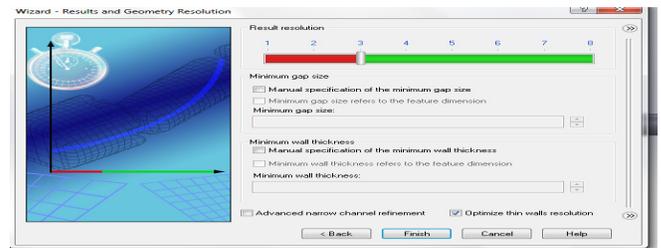
Figure 5: Refinement of rectangular parallelepiped

XI.APPLYING BOUNDARY CONDITIONS ON JET NOZZLES IN SOLIDS WORKS FLOW SIMULATION

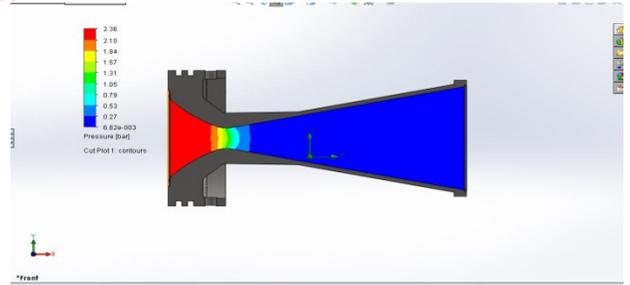
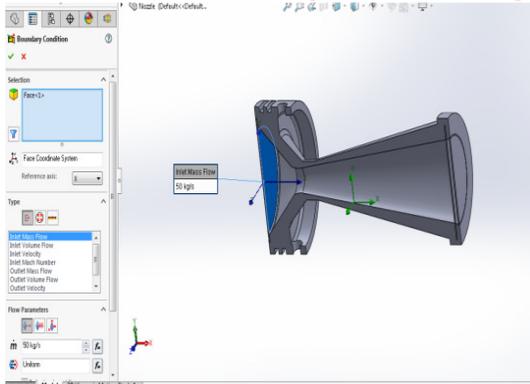
Select default fluid as air



Using the result resolution of 3 as shown below:

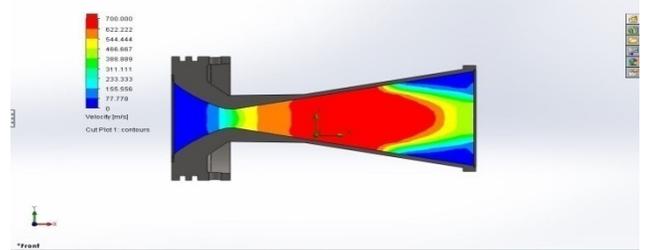
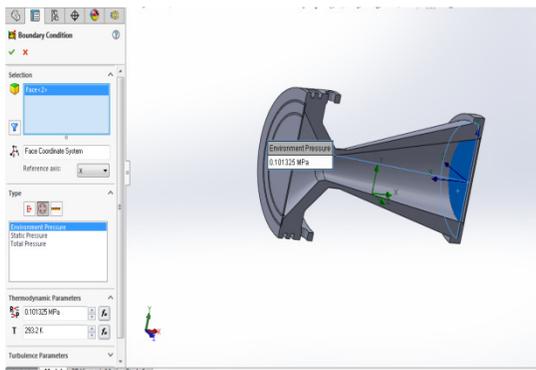


Mass flow inlet 50 kg/sec



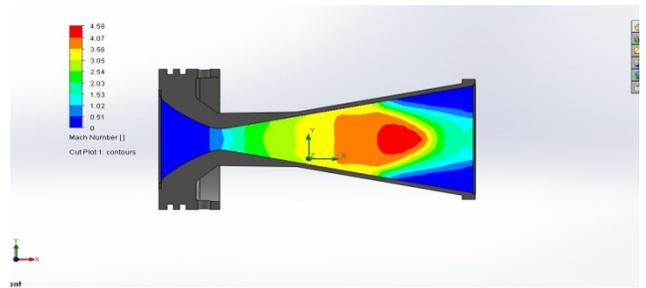
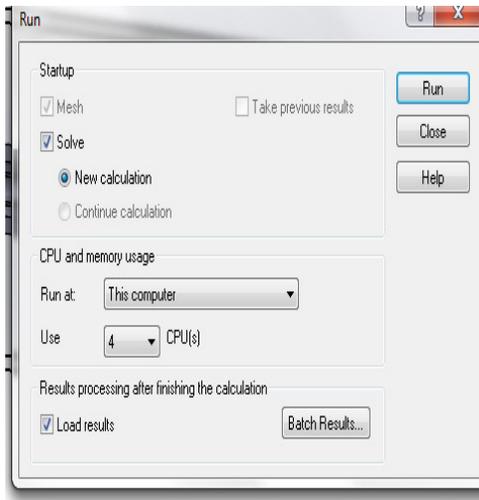
Velocity counters

At out let atmospheric pressure is taken



Mach number

Then solving the flow simulation as shown:



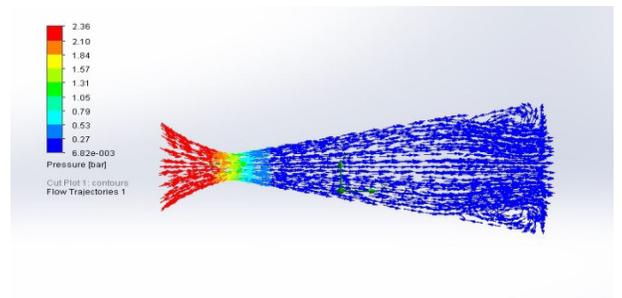
Pressure Trajectories

After solving the results we achieved proper convergence

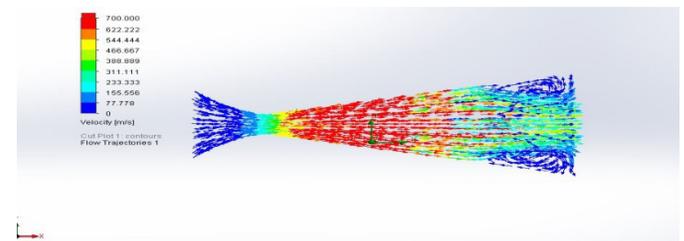
X. RESULTS AND CONCLUSION

Designing of jet nozzle is made in solid works

Pressure counters

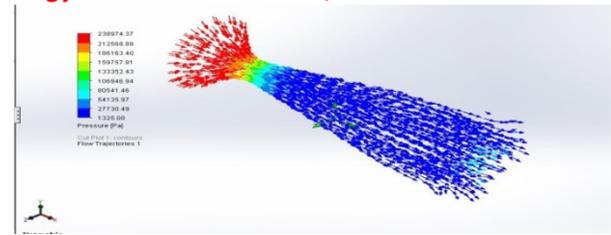
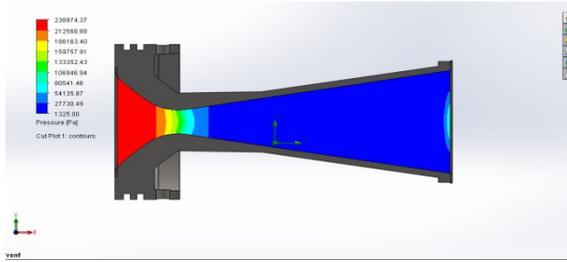


Velocity trajectory

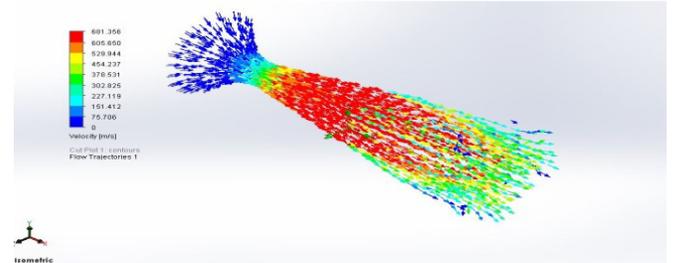


After changing the throat area and divergent area

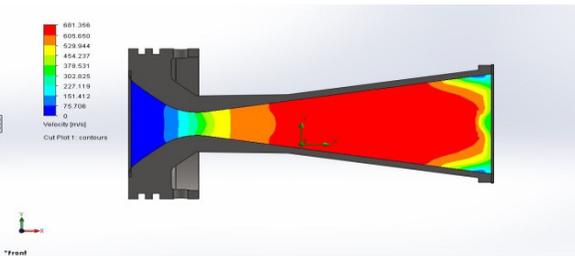
Pressure counters



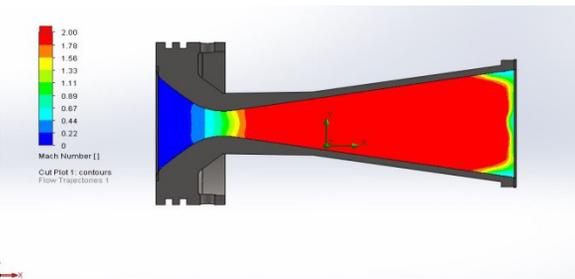
Velocity trajectory



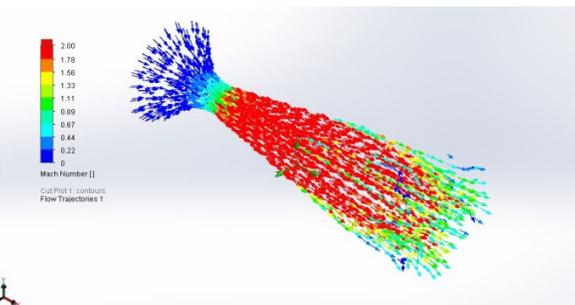
Velocity counters



Mach number



Mach number trajectory



Pressure trajectory

Designing of jet nozzle and CFD analysis made in solid works. It is observed from the results that the maximum velocity is attained at throat area and divergent part of the nozzle. The flow is reached up to 7 Mach number at convergent region; Pressure is decreased at convergent region due to sudden expansion of flue gasses. Shocks waves are detected in jet nozzle due to high Mach number of flue gasses. If we vary the throat area and convergent area it may increase the performance of jet nozzle.

IX. REFERENCES

- [1] K.M.Pandey., Wall Static Pressure Variation in Sudden Expansion in Flow through De Laval Nozzle at Mach 1.74 and 2.23: A Fuzzy Logic Approach, 2010 Second International Conference on Machine Learning and Computing, IEEE, DOI 10.1109/ICMLC.2010.75, pp. 243- 247.
- [2] K.M.Pandey., Wall Static Pressure Variation in Sudden Expansion in Cylindrical Ducts with Supersonic Flow: A Fuzzy Logic Approach, 2010 Second International Conference on Machine Learning and Computing, IEEE, DOI 10.1109/ICMLC.2010.74, PP. 237-242.
- [3] K.M. Pandey and A.P. Singh., CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software, International Journal of Chemical Engineering and Applications, Vol. 1, No. 2, August 2010, pp.179-185.

- [4] K. M. Pandey, R. Jagannath and N. G. Naresh., Studies on pressure loss in sudden expansion in flow through nozzles: a fuzzy logic approach, ARPN Journal of Engineering and Applied Sciences, vol. 2, no. 2, April 2007, pp.50-91.
- [5] K. M. Pandey and E.Rathakrishnan., Influence of Cavities on Flow Development in Sudden Expansion, International Journal of Turbo And Jet Engines, vol. 23, no.2, 2006, pp. 97-112.
- [6] K. M. Pandey and E.Rathakrishnan., Annular Cavities for Base Flow Control, International Journal of Turbo And Jet Engines, vol. 23, no.2, 2006, pp.113-127.
- [7] K. M. Pandey, Prateek Shrivastava, K.C.sharma and A.P.Singh., Studies on Supersonic Flows in the De Laval Nozzle at Mach No. 1.5 and its flow Development into a Suddenly Expanded Duct, The 10th Asian Symposium on Visualization, SRM University, Chennai, March1-5,2010., pp. 164-169.
- [8] K. M. Pandey, Surendra Yadav, and A.P.Singh., Study on Rocket Nozzles with Combustion Chamber Using Fluent Software at Mach 2.1, The 10th Asian Symposium on Visualization, SRM University, Chennai, March1-5, 2010, pp. 171-177.
- [9] K. M. Pandey, Prateek Shrivastava, K.C.sharma and A.P.Singh., Studies on Supersonic Flows in the De Laval Nozzle at Mach No. 1.5 and its Base Pressure into a suddenly Expanded Duct, The 10th Asian Symposium on Visualization, SRM University, Chennai, March1-5, 2010, pp.624-631.
- [10] K. M. Pandey, Virendra Kumar, and A.P.Singh., Study on Supersonic Free Single Jet Flow: A Numerical Analysis with Fluent Software, The 10th Asian Symposium on Visualization, SRM University, Chennai, March1-5, 2010, pp.179-184.
- [11] K.M. Pandey and A.P. Singh., Design and Development of De Laval nozzle for Mach 3 & 4 using methods of Characteristics with Fluent Software, ISST Journal of Mechanical Engineering, Vol.1, Issue 1, 2010, pp. 61-72.
- [12] B. Bouderah, A. Gasmi and H. Serguine, Zero Gravity of FreeSurface Jet Flow, International Mathematical Forum, 3273 – 3277, (2007)
- [13] Yin Zhao-qin, Experimental Study on The Flow Field Characteristics in the Mixing Region of Twin Jets, journal of Hydrodynamics, 309-313, (2007)
- [14] J.R. Berg, S.J. Ormiston, H.M. Soliman, Prediction of the flow structure in a turbulent rectangular free jet, International Communications in Heat and Mass Transfer,(33), 552–563, (2006)
- [15] Abburi, N.R., Dixit, U.S., A knowledge-based system for the prediction of surface roughness in turning process, Robotics and Computer-Integrated manufacturing, (22), 363-372, (2006).
- [16] E. Rathakrishnan, A Numerical Approach to Single and Twin Supersonic Jet Flows, IE (I) Journal AS, January, 32-34, (2004).
- [17] X. Zhang, 'An inclined rectangular jet in a turbulent boundary layer-vortex flow, Experiments in Fluids, (28), 344-354, (2000).
- [18] Wilcox, J.F. Jr., 'Passive venting system for modifying cavity flow fields at supersonic speeds', AIAA Journal, Vol. 26, No. 3, 1988, p. 374-376.
- [19] Rathakrishnan, E., 'Effect of ribs on suddenly expanded flows', AIAA Journal, Vol. 39.

BIOGRAPHY



Mr.A.Chinnamahammad received the B.Tech degree in Mechanical Engineering from MRRITS affiliation to JNTU Anantapur in 2013.He is currently pursuing the M.Tech (Machine Design) from Vignan University A.P. Two years of experience as a service engineer in L&T construction equipment & mining machinery division.



Mr. N.Vijay rami reddy received the B.Tech degree in Mechanical Engineering from MRRITS affiliation to JNTU anantapur in 2013. He is received the M.Tech degree in Machine Design from JNTU Hyderabad in 2015.

One year of teaching experience in MGIT and presently working as Asst. Prof in MGIT, Hyd.



Mr. B.Rajnaveen received the B.Tech degree in Mechanical Engineering from Acharya Nagarjuna University. He is received the M.Tech degree in CAD/CAM, Osmania University, Hyderabad. One year of teaching experience in MGIT and presently working as Asst. Prof in MGIT, Hyd.