ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering
(IJERMCE)
Vol 6, Issue 10, October 2021

Design and Analysis of Mass Flow Measurement Unit for Supersonic inlet

[1] Chinta Venkat Sai, [2]Bhargav Yedla, [3]Siva Surya Narayana Podipireddy, [4]Vasanthakumar P
[1][2][3] Student , Hindustan Institute of Technology and Science, Chennai, India
[4] Professor, Hindustan Institute of Technology and Science, Chennai, India

Abstract:**-- A numerical study was carried out to determine the mass flow rate of a typical supersonic inlet using Venturi Meter setup as the Mass flow Measurement Unit. The mass flow rate of the intake is a characteristic flow property that determines the thrust developed by the engine. The concept of determining the mass flow rate in the supersonic regime is quite complex without any compensation in the measurement. A Venturi meter was designed and carried out simulation to make the flow incompressible in the settling chamber and to prevent this complexity. The flow properties can be easily measured if the flow is incompressible using the most straightforward formulae. The results from the CFD analysis are validated with the theoretical ones.**

Index Terms:- Mass flow measurement unit, mass flow rate, Computational Fluid Dynamics, Venturi meter, Incompressible flow

I. INTRODUCTION

The world witnessed rapid development in the aviation industry during and after world war-2. Aircraft range, altitude, and speed are rapidly increased. During this time, the aircraft crossed the sound barrier in terms of speed. The vehicle design that operates in supersonic flight regimes is a challenging task for engineers and scientists because the properties of air change from incompressible to compressible at supersonic speeds. This property of air causes shock wave formation, which affects the pressure, velocity, density, and temperature of the flow. Different types of engines have been developed to make supersonic flight sustainable and possible. Ramjet engine is most popular among the class. Ramjet does not consist of any rotating parts. It is a ducted engine where isentropic compression takes place through the shocks generated from the inlet design followed by constant pressure combustion and isentropic expansion.

The operating conditions of the ramjet engine are supersonic, which means the free stream velocity at the intake is supersonic. The optimum design of the inlet plays a vital role in determining the thrust produced by the engine. Extensive research is carried out to improve air-breathing engines for military application to improve efficiency when flying through the atmosphere. An adequately engineered air intake ensures the quality and quantity of air supplied to the engine for efficient operation at various Mach number regimes. The air intake can disperse the incoming ambient air to a desired subsonic Mach number appropriate for the engine's supersonic Mach numbers. External or internal compressions and a combination of external and internal compression are used to achieve this diffusion mechanism.

The mass flow rate, which determines the amount of thrust that an engine can generate, is determined by the air intake. The primary purpose of the supersonic air intake is to provide adequate air requirements for the combustion chamber of an air-breathing engine, especially at supersonic flow conditions. It has a significant effect in evaluating the performance of an air-breathing engine by providing the desired mass flow rate, low distortion flow while operating at various operating conditions of supersonic flow regimes.

The mass flow rate of the inlets can be calculated using a variety of methods and apparatus. Venturi meter is a mass flow measurement device that works on the principle of Bernoulli's equation. Due to the complexity of the flow, high velocity, and boundary layer interactions, determining the mass flow rate of that particular inlet becomes difficult once the flow velocity exceeds the Sonic Mach number. Therefore, the area of focus is to design and test the suitable mass flow measuring equipment for the supersonic flow of a Ramjet engine intake using Venturi meter in CFD to co-relate with the theoretical values and check for precision.

The optimum design was selected using method used by Nithin [1]. The turbulence model was selected based on comparative study by Florian R Menter [2]. The simulation data from [3], [4], [5] and [6] is taken as reference for our research work. The mass flow rate analysis and the comparative study between theoretical and simulated data, done in this research work was referred from simulation carried out by Ameresh [4]. The theoretical and fluid dynamics-based calculations were performed after extensive study from study material [7], [8], [9] and [10].

ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)

Vol 6, Issue 10, October 2021

II. GEOMETRICAL SETUP

The MFU setup comprises a transition pipe right after the inlet followed by two elbow joints with a duct connecting them and a venturi meter setup with a diffuser and settling chamber. The dimensions of the venturi setup were tabulated in TABLE 2.1. The 3D model of MFU was modelled in CATIA V5. The Designed model is the extracted fluid volume from the Actual setup.

Table 2.1. Dimensions of Venturi Meter

Components	Angle	Diameter	Length
Diffuser	10^0	D1-50, D2-150	276
Settling Chamber	0	150	450
Converging Section	10.5^0	D1-150, D2-90	162
Throat	0	90	90
Diverging Section	3.5^0	150	486

All lengths are in millimetres (mm).
Pipe length = 400 mm
Pipe diameter = 65.5 mm
Elbow outer radius = 200 mm, turn angle = 90^0
Transition horizontal length = 95.81mm

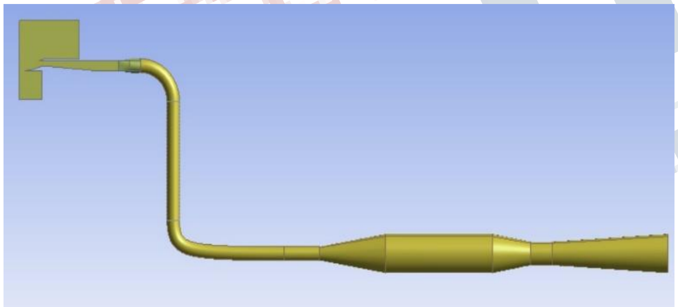

FIG 2.1 3D DESIGNED MODEL OF MFU

III. COMPUTATIONAL PROCESS

A. Grid Generation

The Grid generation for the designed model was carried out in ICEM CFD software to obtain structured meshing, yielding better results than the unstructured one. The grid was generated in such a way that the elemental and orthogonal quality of the grid was maintained above 0.6.

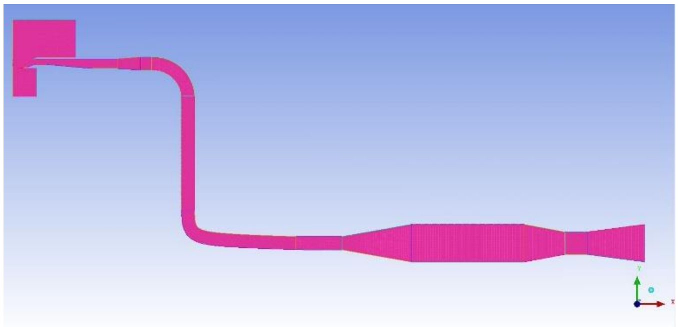

FIG 3.1 GRID GENERATION OF THE WHOLE MODEL IN ICEM

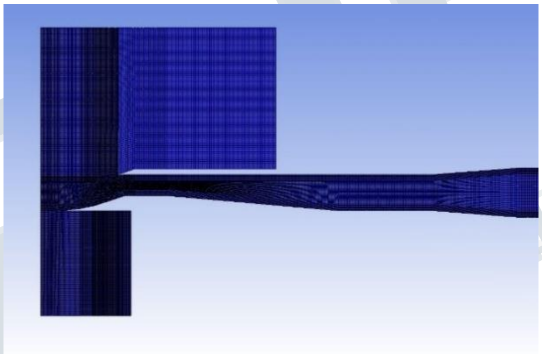

FIG 3.2 ENHANCED IMAGE OF GRID

B. Computational Approach

The theoretical mass flow rate through the inlet is computed using the Continuity equation it is found out to be 0.772738 kg/s. For every CFD simulation carried out, the mass flow rate values will be cross-checked with this theoretical value.

$\dot{m}_{th}$ **= 0.7727 kg/s**

In the present study, the three-dimensional problem was solved in Fluent using a density-based implicit formulation with double precision. The turbulence model selected was k-ω with SST because of its preference over the k-ϵ model and its ability to compute the flow fields near the wall and flow in ducts and pipes. The fluid air is set as IDEAL GAS. So, the density is derived from it automatically. The Specific heat of fluid is a piece-wise polynomial function of temperature. The viscosity of the fluid is calculated using the three coefficient Sutherland's equation. Thermal conductivity is considered to be constant. For this problem, the grid values are initialized from the Inlet values using Standard Initialization. Then, FULL MULTIGRID (FMG) Initialization is used to accelerate the solution for convergence. The Courant number is initially set to 0.1 and is gradually increased with iterations because it helps the solution converge faster and get accurate results. The calculation process is started with Spatial discretization settings such as Least Squares, Cell-Based

Gradient, and Second-Order Upwind flow. The first order upwind flow setting is not selected cause the FMG initialization helps in yielding the initial and approximate solution at the same time. The model taken for simulation is the cut section about the Z-axis of the designed model.

The Named selections are given as per our problem description, and the boundary conditions are assigned respectively.

Inlet - Pressure Inlet
Total Pressure – 480000 Pascals
Static Pressure – 38386.84 Pascals
Total Temperature – 300 Kelvin
External Flow field – Pressure Far-field
Gauge pressure – 38386.84
Mach Number – 2.3
Static Temperature – 146.0619 Kelvin
Outlet – Pressure Outlet
Pressure – 101325 Pascals

IV. RESULTS AND DISCUSSION

The simulation was carried out with prescribed boundary conditions in the previous section. The results were extracted from the raw simulated data, and the discussions were carried out with the help of contours and graphs from the resultant data.

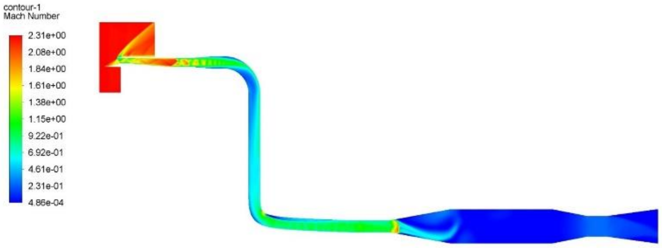

FIG 4.1 MACH NUMBER CONTOUR OF MFU SETUP

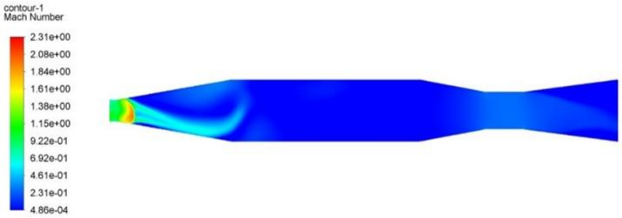

FIG 4.2 MACH NUMBER CONTOUR OF VENTURI METER SETUP

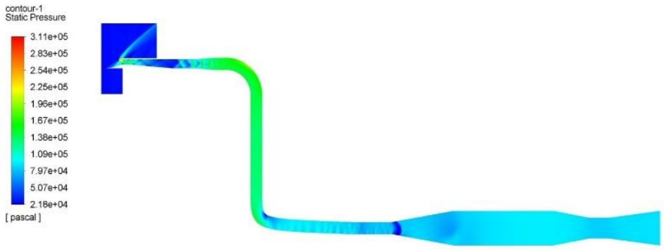

FIG 4.3 STATIC PRESSURE CONTOUR OF MFU SETUP

The mass flow rate through the inlet of intake is 0.3588 kg/s.

From CFD analysis, the mass flow rate through the outlet is 0.3520 kg/s. As we simulated a model which is a cut section, the total mass flow rate through the outlet is given as,
The total mass flow rate at outlet = $\dot{m}$ **= 0.7042 kg/s**

The mass flow rate, in this case, is calculated by Fluent and it is 0.7042 kg/s which is 0.0685 kg/s deviated from the theoretical one. The deviation is due to the errors in mesh and frictional losses at the elbow joint and flexible joint in a minimal fashion.

The flow parameters are plotted along the centerline of the venturi meter to strengthen the assertion of establishing incompressible flow regime.

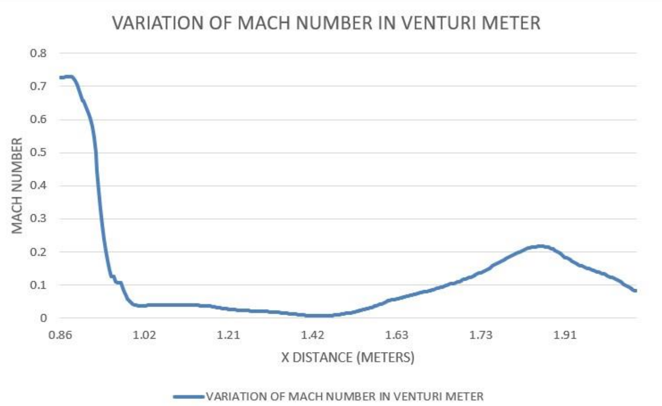

FIG 4.4 VARIATION OF MACH NUMBER IN VENTURI METER

IFERP
connecting engineers... developing research

ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 6, Issue 10, October 2021

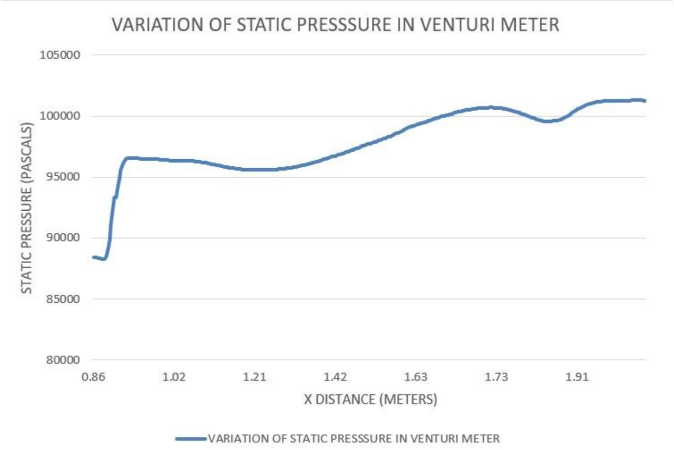

FIG 4.5 VARIATION OF STATIC PRESSURE IN VENTURI METER

From the simulation carried out in the ANSYS Fluent. The flow is compressed from the ramps outside the inlet on the formation of the oblique shocks; a terminal shock is formed initially due to the cowl lip of the intake, the three oblique shocks generated from the ramps are intersecting at the cowl lip, forming a terminal shock. This can be observed from the Mach contour that the velocity of the flow decreases soon after the terminal shock formation. The shock train travels along the inlet and the elbow joint provides backpressure towards the freestream; as a result, the velocity of the flow decreased downstream of the elbow joint.

Flow expands through the diffuser section; since the exit pressure is atmospheric, there would be a back pressure effect on the flow and settles further to become subsonic flow. The flow at the throat section of the venturi meter is subsonic, which is a desired property of the simulation. The variation of Mach number and static pressure can be observed from the contours (figures 4.1- 4.3). The interpretation of flow parameters near the wall and mainly at the elbow and other joints can be observed clearly in the resultant contours, which depict the frictional and pressure losses that specifically account for the creation of backpressure. The figures (4.4, 4.5) infer the variation of Mach number and Static Pressure along the centerline of the venturi meter to strengthen our assertions about the incompressible flow regime in the settling chamber of the Venturi meter.

The mass flow rate measurement from the CFD analysis showed precision with minimal losses strengthening the fact that the Mass Flow Measurement Unit provides an accurate analysis of flow parameters without any compensation in accuracy.

V. CONCLUSION

In this study, the MASS FLOW MEASUREMENT UNIT was designed with specific design considerations, and it was simulated in FLUENT to obtain the results. The CFD results were validated with the Theoretical mass flow rate calculations, and the precision of the simulated results was checked. The visual interpretation of the variation of flow parameters can be observed with the help of contours. The graphs were also plotted to scrutinize Mach Number and Static pressure variation in a precise mathematical fashion. The results from the simulation depict the occurrence of incompressible and subsonic flow regimes near the throat region of the Venturi meter setup. The flow regime attaining an incompressible range infers that the calculation of flow parameters can be precise without inaccuracy.

	Theoretical Approach	Computational Approach
Mass Flow rate(kg/s)	0.7727	0.7042

The mass flow measurements from the theoretical and CFD results show the degree of precision attained in the present study. This Mass flow measurement unit comprising the Venturi meter is the efficient and most compatible way to measure the flow properties without involving the complexity of the supersonic flow regime.

VI. ACKNOWLEDGMENT

The authors would like to express their gratitude to all the scientists in DOAT and DOAD of DRDL, Hyderabad for providing this opportunity to carry out this research project. The authors would also like to thank the staff of DOAT and Hindustan Institute of Technology and Science for their contribution to the success of this research project.

REFERENCES

[1] T. Nitin, Nikhil Jain, and Adarsha Hiriyannaih, Optimization of Venturi Flow Meter Model for the Angle of Divergence with Minimal Pressure Drop by Computational Fluid Dynamics Method, 2012, *International Conference on Challenges and Opportunities in Mechanical Engineering, Industrial Engineering and Management Studies (ICCOMIM)*

[2] Florian R. Menter Eloret Institute, Sunny vale, Zonal Two Equation k-ω Turbulence Models for Aerodynamic Flows, 1993, *24th Fluid Dynamics Conference, American Institute of Aeronautics and Astronautics.*

[3] A Tukimin, M Zuber and K A Ahmad, CFD analysis of flow through Venturi tube and its discharge coefficient, 2016, *IOP Conf. Series: Materials Science and Engineering*

[4] H.Ameresh, DepaSandeep, P. Ravikanth Raju, R. Venkat Reddy, Investigation of Mass Flow Rate in Venturimeter Using CFD Analysis, 2017, *Int. Journal of Engineering Research and Application, Vol. 7, Issue 12, (Part -7) December 2017, pp.86-90*

[5] Ekong, Godwin I, The Performance Analysis of a Venturimeter Flow Rig, 2020, *International Journal of Engineering Science Invention (IJESI) ISSN (Online): 2319-6734, ISSN (Print): 2319-6726 ||Volume 9 Issue 5 Series III || May 2020 || PP 14-23*

[6] Manish R. Bhatkar, Preeti V. Ban, "Review Study on Analysis of Venturimeter using Computational Fluid Dynamics (CFD) for Performance Improvement'', 2019, *International Research Journal of Engineering and Technology (IRJET), e-ISSN:2395-0056, Volume: 06 Issue: 10 | Oct 2019*

[7] Harris. Michael J. (2011), Venturi Meters,

[8] Fundamentals of Aerodynamics, Book by John D. Anderson.

[9] Computational Fluid Dynamics: An Introduction Book by John D. Anderson

[10] Richard W. Miller, Flow Measurement Engineering Handbook,1996.