# Steady Analysis of NACA Flush Inlet at High Subsonic and Supersonic Speeds

Taimur A. Shams[1], Masud J[2]

***Abstract***— **Essence of this research is to report computational analysis of NACA flush inlets at high subsonic and supersonic free stream Mach numbers at sea level conditions. Flow physics inside flush inlet has been computationally captured including counter rotating vortices, accelerated flow on inlet ramp and boundary layer divergence. The use of various turbulence does not change the basic qualitative nature of the flow. Generally mass flow rate, mass flow ratio and total pressure recovery of air passing through the inlet decreases with increasing free stream Mach number. Steady results depict that SA turbulence model computed higher values of flow performance parameters and lowest values of effective viscosity at all Mach number regimes as compared to other turbulence models.**

***Key Words***—**NACA Flush Inlet, CFD, Turbulence, Simulations, Pressure Recovery**

## I. INTRODUCTION

Flush / submerged inlet consists of a shallow ramp with curved walls recessed into the exposed surface of a streamlined body. A part of the free stream air is forced into the inlet by its carefully designed sidewall contour and bottom wall ramp angle. At subsonic Mach numbers sidewalls form strong vortices that play an important part in forcing air into the inlet also they help in removing some of the low energy viscous boundary layer. That is why flush/ submerged inlets show better performance in subsonic regime. The flush / submerged inlet has been the focus of significant research in the 1950s by various departments of the then National Advisory Committee for Aeronautics (NACA), later upgraded and renamed as the National Aeronautics and Space Administration (NASA). The overwhelming conclusion was that NACA flush / submerged had adequate performance in subsonic regime as an intake for jet propulsion systems but its transonic and supersonic performance was inadequate compared to some of the other intake configurations.

Due to these features the NACA flush inlet is not seen in any of the supersonic aircraft propulsion system. However the NACA flush inlet has found secondary applications in aircraft auxiliary systems for providing cooling air where the pressure recovery and flow quality are not of prime importance [1]. This is because the NACA flush inlet offers drag reduction in subsonic and supersonic flight regimes (lesser wetted area etc) once compared to a conventional protruding pitot-type ram air scoop.

Additionally, the NACA flush inlet also offers certain weight savings. Due to these aspects the auxiliary application of NACA flush inlet is seen on most modern commercial airliners like Boeing 747, supersonic fighters like Euro fighter and high speed racing cars like Ferrari and F-1as shown

Fig.1. NACA Flush inlets on racing car

This study involves computational analysis of NACA flush inlet at high subsonic and supersonic mach regimes in order to verify different flow characteristics like mass flow rate, mass flow ratio and pressure recovery. Serious CFD analysis first requires verification specially if the problem is highly complex such as turbulent / compressible flow. The most desirable form of verification is done against known test data, which for the present configuration is not readily available (wind tunnel tests). Due to this fact the present analysis covers a broad range of computational variables that can be evaluated and refined once the corresponding experimental data is available.

## II. RELATED LITERATURE REVIEW

The purpose of intake is to ensure that an aircraft engine is properly supplied with air under all conditions of aircraft operation [1].

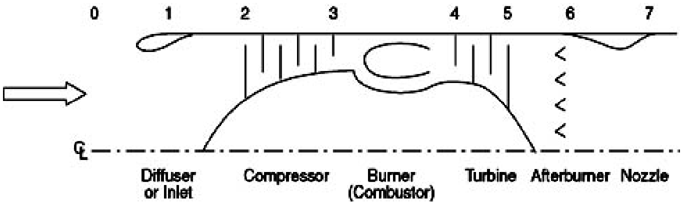

Fig.2. Basic layout of an engine

The incoming air towards duct is divided in internal and external flow. Internal flow feeds the engine, whereas

Manuscript received on Jan 09, 2015;

Taimur Ali Shams (corresponding author) is Assistant Professor in Department of Aerospace Engineering, College of Aeronautical Engineering, National University of Science and Technology, Islamabad, H-12, Pakistan. (taimur12389 @hotmail.com) +923434463352

Masud. J is Professor in Institute of Avionics and Aeronautics, Air University, Islamabad, Pakistan

external flow affects the aerodynamics of airframe. Speed of air at engine face is required to be moderately subsonic, lower than the principle aircraft speed. Due to this fact, the basic shape of the duct becomes important. In order to reduce the speed of incoming air, the front part of the duct is in the form of a diffuser; increasing in area from the entry, to a position representing the engine face. The rear part of the duct is then convergent, simulating in essence the engine nozzle system [1].Principal stations in the flow are indicated in Figure 1.

Mass Flow Rate and Pressure Recovery are the factors among many which govern the performance of inlet. Mass flow rate (kg/s) is the primary variable upon which all the others are dependent [4]: denoted by m, it is defined by

$$m = \rho A V \qquad (1)$$

In high speed flight an air inlet is a form of compressor; it accepts air initially at free stream Mach number and pressure, and converts it to lower Mach number and correspondingly higher static pressure, as required by the engine. For compressible flow conditions, this definition can further simplified

$$\eta = \frac{P_{intake\,entry}}{P_{Compressor\,face}} \qquad (2)$$

Equation 2 is the total pressure ratio. The efficiency of the inlet is defined by this equation is termed as pressure recovery of the inlet. This efficiency denotes the loss of total pressure from the free stream.

Loss of total pressure can occur due to friction on the walls of the duct and on any external surface which is wetted by flow going into the duct, turbulent mixing associated with flow separation and due to shock waves [1]. Pressure recovery has a determining effect upon engine thrust [4], which may be defined as the resultant force in the direction of flight produced on the aerodynamic duct system by the internal flow.

## III. SUBMERGED INLET

NACA duct or NACA scoop is a common form of low-drag intake design. Originally it was developed by the National Advisory Committee for Aeronautics (precursor to NASA) in 1945. When properly implemented, it allows air to be drawn into an internal duct, often for cooling purposes, with a minimal disturbance to the flow. The design was originally called a "submerged inlet," since it consists of a shallow ramp with curved walls recessed into the exposed surface of a streamlined body, such as an aircraft. Submerged inlet experiments showed poor pressure recovery due to slow-moving boundary layer entering the intake. This design is believed to work because the combination of the gentle ramp angle and the curvature profile of the walls create counter-rotating vortices which deflect the boundary layer away from the intake and draws in the faster moving air, while avoiding the form drag and flow separation. This type of flush inlet generally cannot achieve the larger pressure recovery and flow quality(low distortion factor) of an external design, and so is rarely used for the jet aircraft intake application. It is, however, common for aircraft cooling applications in the form of submerged scoops.

### *A. Working Principle of Flushed Inlet*

The working principle of submerged inlet is based on the formation of two counter-rotating vortices along the sharp leading edges of the surfaces leading up to the entry plane [1]. These vortices sweep the ramp boundary layer sideways and carry a proportion of it past the ends of the entry and out into the external stream. Thus the ramp arrangement works as a form of boundary layer diverter, reducing the effective position ratio of the inlet. Results can sometimes be enhanced by the addition of small ridges along the divergent sidewalls, increasing their effective height, and also by increasing the sidewall divergence to allow for slots at the ends of the entry, easing the passage of the boundary layer [4]. The other function of these vortices is that they drag the external flow into the inlet by their rotation.

Other factors affecting success of the NACA submerged inlet depends on practical considerations of layout, such as space for a comparatively longer ramp at a sufficiently small angle (about 7degrees ) and on having some indication of the extent to which the boundary layer diverted from the inlet adds to the drag of the aircraft. Submerged inlet are generally considered unsuitable for supersonic speeds, because velocity, or Mach number, on the initial part of the ramp is higher than that of the free stream, however in the context of small auxiliary inlets, they can also be used at supersonic speeds [1].

Submerged inlets offer few distinctive advantages like low drag, small overall dimensions and small radar cross-section. These inlets also have some drawbacks like high attitude sensitivity (particularly at negative incidence), limited pressure recoveries, especially if the upstream Mach number exceeds unity and large internal volume requirement (as the inlets are long).

### *B. Basic Parameters of Flush Inlets*

Submerged inlet has three basic parameters [6] which are Ramp Planform, Throat Aspect Ratio (Width to depth Ratio) and Ramp Angle. Different types of ramp plans were evaluated including Rectangular, straight divergent and curved divergent. It has been observed from previous investigations [5] that curved divergent shape give higher pressure recovery.

Higher pressure recoveries are obtained for diverging ramps over a wide range of inlet velocity ratios [6]. These experimental results [5] suggest that pressure recovery becomes higher as the throat gets wider for the same entrance area. Due to this extension of throat width, the divergent angle of ramp walls is also increased and the height of throat becomes low. The vortex generated along ramp walls separates early and becomes stronger due to increased divergent angle. As the roll-up vortex moves downstream in the duct, the downwash of the vortex results in thinning the boundary layer on the ramp floor [19]. Thin boundary layer shall ultimately aid in achieving high pressure recovery.

Ramp angle of the floor effect the shape of the inlet cavity. The resulting change in length of the ramp affects the pressure recovery characteristics of submerged inlet. It is found that low ramp angles give the higher pressure recoveries, whereas increase in ramp angle is accompanied with decrease in pressure recovery [5]. Ramp angle of 7 degrees give higher pressure recoveries for low velocity ratios. For non divergent ramp walls the decrease in pressure recovery is caused by thickening of boundary layer due to more adverse pressure gradients along the ramp. In case of divergent inlet shape, increasing the ramp angle increase the angle between the diverging walls [5]. This produces two adverse effects. First it increases the tendency of flow toward separation and secondly it increases the obliquity between the ramp walls and the free stream flow. Due to this it is difficult for free stream flowing along the outside edge to follow the contour of inlet cavity. This causes boundary layer admitting into the ramp area and cross flow between this air and the air flowing down the ramp. Combination of these two adverse effects results into large pressure losses due to increase in ramp angle.

## IV. GEOMETRY PREPARATION AND MODELLING

The basic principle of this model layout is that free stream air enters through the flush inlet, passes through the duct and exit at a downstream location. The three dimensional schematic layout of the NACA flush inlet model is shown.

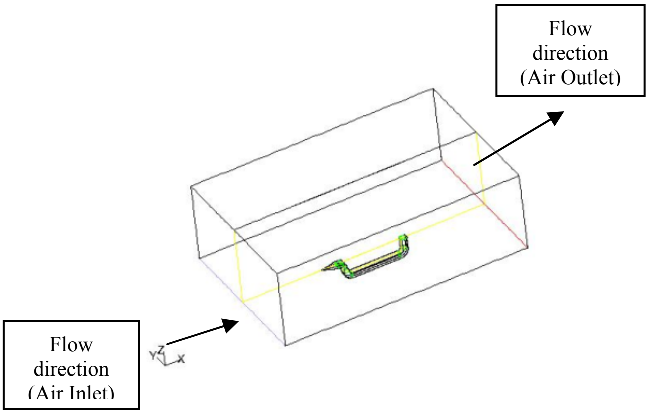

Fig.3. Basic layout for computational analysis

The exact contour of the NACA flush inlet itself (without the duct) is shown below along with numerous other sources that provide similar information. It can be seen that except for two NACA reports, all other sources give more or less same plan form coordinates. The bottom wall ramp angle of 10 degrees has been used which is close to 5~7 degrees range given in the other sources. The inlet lip radius of the model is 2 mm that apparently is sharp enough for supersonic applications.

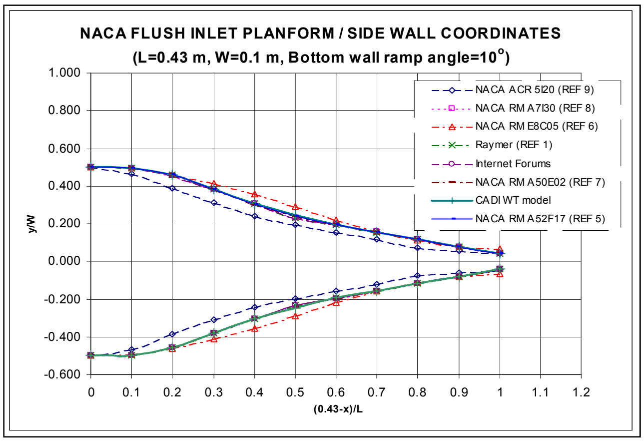

Fig 4. NACA flush inlet plan form coordinates

Duct with square cross section has been used. Duct cross sectional area is 0.0176m2 and its length is 2.27m. A square cross-section duct poses somewhat severe conditions (pressure loss etc) to the internal flow compared to a circular duct having the same cross-section area therefore this approximation is conservative. Under mentioned figure shows the general layout of NACA flush inlet with duct. Contour between NACA inlet exit plane and duct is because of the merging of rectangular cross section of exit plane with square cross section of duct.

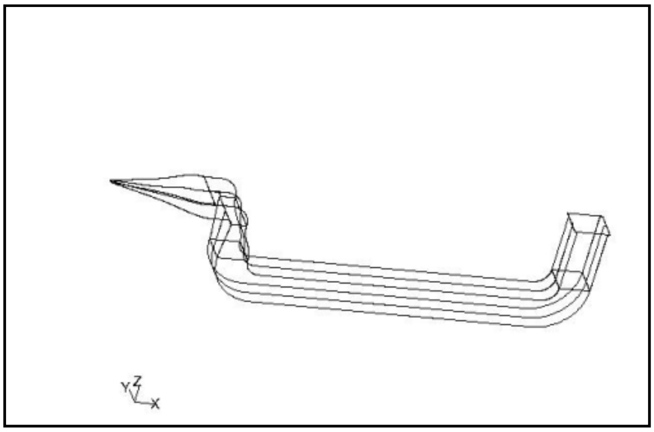

Fig.5. Flush inlet with square cross section duct

In addition to the flush inlet and duct, the complete flow domain is modeled. This included the upstream (flow inlet), downstream (flow outlet) and the top and side flow containment walls. Complete geometry is shown below. Mapped hexahedral brick elements were used to mesh the computational domain due to their better solution stability. Elements were graded near the surface of the flush inlet and duct outlet to resolve important fluid dynamic phenomena like separation, vortices and shock waves etc. Elements were also graded near solid (no slip) walls to resolve the viscous boundary layers. Additionally, elements were also graded in the wake region of the inlet and duct outlet to resolve the wake and stabilize the solution. The overall mesh layout of the computational domain is shown below. The 3D NACA flush inlet was built and meshed in FLUENT® for analysis.

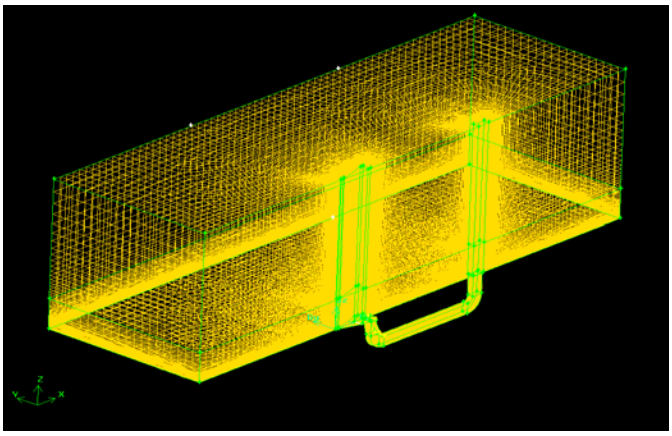

Fig.6. Meshing of complete geometry (4.09 million cells)

The flow through and around the NACA flush inlet is modeled as follows:

a) The computational domain was modeled as three-dimensional (Cartesian coordinates) with a plane of symmetry (X-Z plane), which significantly reduced the computational effort.

b) Flow was modeled as turbulent and steady for some computations

c) Air was modeled as a compressible fluid obeying the ideal gas law.

d) Temperature variations / effects are inherent in compressible analysis; therefore the energy equation was solved as well. Since heat transfer is not of prime interest in the present analysis therefore adiabatic conditions on flow domain boundaries (default) were accepted. Viscous dissipation was ignored since it is not expected to influence the computed results.

e) Reynolds-averaged Navier-Stokes (RANS) system of equations for turbulent flow was solved.

f) The two-equation Standard K-ε,, RKE, SST KW and Spalart-Allmaras (SA) turbulence models were used to determine the sensitivity of computed results with the turbulence model.

No-slip velocity boundary conditions specified at the bottom wall that is flush inlet and the duct. Zero-shear (slip) boundary conditions specified at the top wall and the side wall in order to avoid the unnecessary formation of viscous boundary layers.

At the inlet of the domain the total pressure, total temperature, static pressure and velocity were specified corresponding to the desired Mach number under standard sea-level conditions as part of the "pressure-inlet" boundary condition in FLUENT®. Symmetry boundary condition (i.e. zero normal temperature gradient, zero-shear and zero normal velocity) was specified on the symmetry plane (X-Z).

At the outlet of domain, static pressure and total temperature were specified as part of the "pressure-outlet" boundary condition. The convergence criteria for the scaled residuals for 3D steady analysis was set at 1x10-4.

## V. GRID INDEPENDENCE STUDY

Numerically computed results change with the type and fineness of the mesh / grid used for computations. Because each result is an approximate solution of non linear partial differential equations, truncation errors exist in the flow domain solution. The truncation errors depend on the grid resolution hence it is important to decide that the grid size is sufficient to solve the flow domain accurately and small enough to save Computational time. To be able to decide the sufficient number of grid points, a grid sensitivity analysis for this specific problem is necessary. The purpose of the study is to determine the effects of truncation errors involved in the numerical analysis of submerged inlet. In order to determine the dependence of the computational results on grid, three different grids were computed at Mach 0.8, one with 0.36 million, 2.16 million and 4.09 million mesh size. SA turbulence model was used as turbulence model with default values of its constants. Mass flow rate and pressure recovery were selected as prime parameters for convergence. Following table shows that fine mesh constituting 4.09 million size gives best result with experimental values as calculated by Seddon [1]. Thus further analysis is done on fine mesh geometry.

TABLE1
GRID INDEPENDENCE TABLE AT MACH 0.8

| Grid | Cell Size | MFR | PR | Y+ | REMARKS |
|---|---|---|---|---|---|
| Coarse | 364440 | 0.48 | 0.92 | 950 | Not selected |
| Medium | 2169861 | 0.49 | 0.91 | 375 | Not selected |
| **Fine** | **4097303** | **0.52** | **0.89** | **104** | **Selected** |

## VI. RESULTS

Primary performance parameter for the present NACA flush inlet is the mass flow rate of air passing through the inlet because present configuration is representative of auxiliary cooling application.

Since present configuration also represents an air intake system therefore secondary performance parameter is the total pressure recovery of the inlet. The computed mass flow rate of air passing through the inlet under different free stream Mach numbers is shown for various turbulence models. The mass flow rate data is made non-dimensional in terms of mass flow ratio which is the inlet mass flow rate divided by free stream mass flow rate through inlet capture area. Total pressure recovery is also shown for Mach numbers of 0.8, 1.2, 1.5 and 1.8 with different turbulence models. Total pressure recovery has been calculated at the NACA flush inlet exit plane.

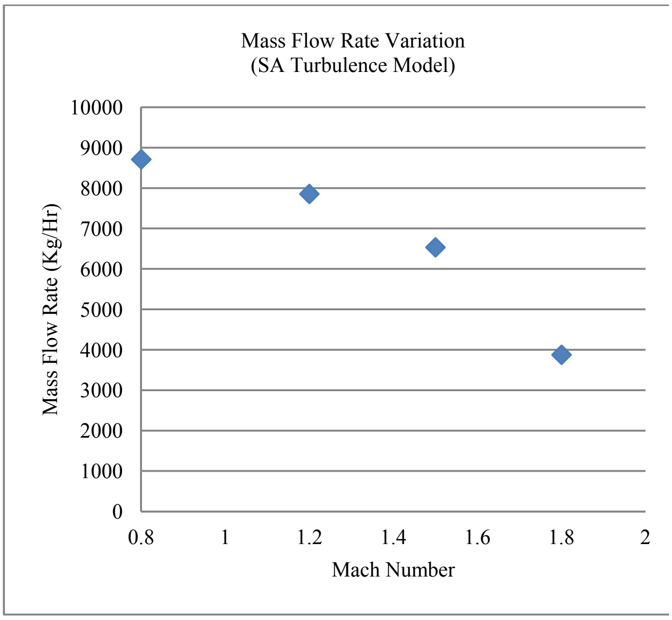

Fig.7. Mass Flow Rate at Mach 0.8,1.2,1.5 and 2.0

Mass flow rate of air passing through the inlet / duct decreases with increasing free stream Mach number. This is caused by the increased backpressure in the duct due to the stronger interaction of duct exit airflow with the free stream as the Mach number increases.

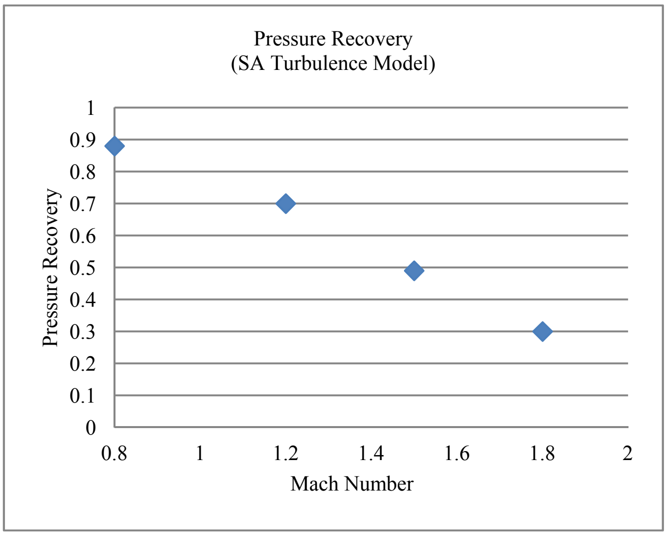

Fig.8. Total pressure recovery at Mach 0.8,1.2,1.5 and 2.0

Generally the total pressure recovery of air passing through the inlet / duct decreases with increasing free stream Mach number. This is caused by the increased total pressure loss associated with flow across a stronger shockwave in the flush inlet as the free stream Mach number (supersonic) increases. At subsonic free stream conditions (M=0.8) the computed total pressure recovery at the inlet exit plane is around 0.88 which is the optimum value indicated by past research on the flush inlet.

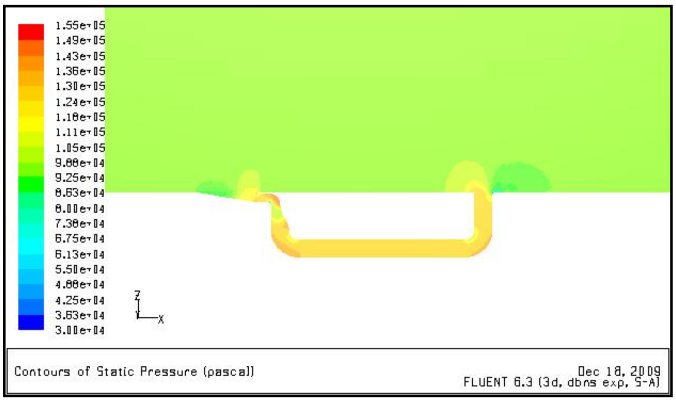

Fig.9. Pressure contour on symmetry plane flush inlet (Mach 0.8)

Basic flow field in and around the NACA flush inlet at subsonic free stream Mach numbers consists of a part of the free stream airflow being forced into the intake by virtue of two strong counter-rotating vortices that form on the sidewalls. The airflow inside the inlet then passes through the duct having three (03) 90° bends and about 2.27 m average length (from inlet throat to duct exit) and exits back to the free stream at a downstream location.

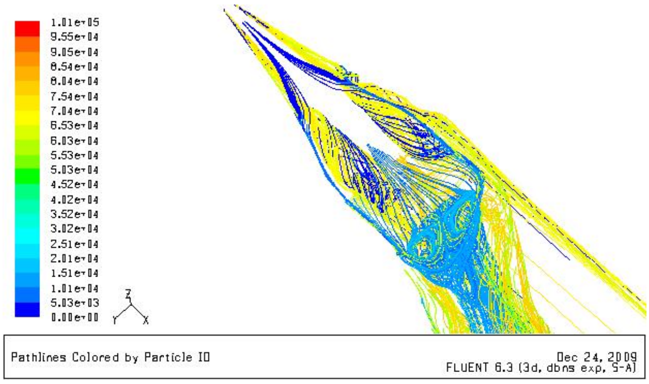

Fig.10. counter rotating vortices in flush inlet (Mach 0.8)

The formation of the counter-rotating vortices at the flush inlet sidewalls influences the flow quite a distance above the flush surface and forces it inside the inlet. However the strength and location of these vortices do not solely determine the amount of airflow going inside the inlet. This is due to the uniqueness of the present configuration i.e. the airflow inside the duct being exited again to the free stream. Region of local flow acceleration at the forward half of the flush inlet is also evident and is probably caused by the close proximity of the sidewall vortices.

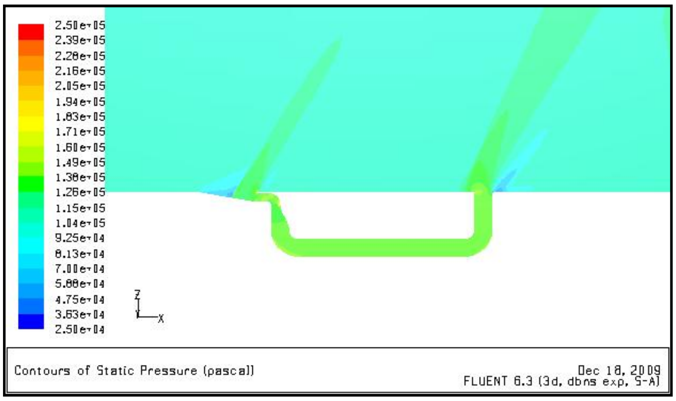

Fig.11. Pressure contour on symmetry plane flush inlet (Mach 1.2)

Region of local flow acceleration at the forward half of the flush inlet (before the shockwave) is evident and is caused by the supersonic expansion in the divergent channel formed by inlet geometry. After initial acceleration, the airflow undergoes rapid change through a series of shockwaves that are identifiable by rapid pressure increase and Mach number reduction on the symmetry plane. These shockwaves are located significantly ahead of throat. The location of the front shockwave is primarily determined by the amount of airflow entering the inlet. At design point this shockwave should form just ahead of the throat with highest airflow into the inlet.

## VII. CONCLUSIONS

It is concluded that NACA flush inlet can be used in cooling application rather than primary source of air inlet due decreased mass flow rate, mass flow ratio and decreased total pressure recovery.

## REFERNCES

[1] Seddon, J. and E.L. Goldsmith, Intake aerodynamics. 2nd ed. AIAA education series. 1999, Reston, VA: American Institute of Aeronautics and Astronautics / lackwell Science Ltd.
[2] Raymer, D.P., Aircraft Design : A Conceptual Approach. 3rd ed. AIAA Education Series. 1999: AIAA
[3] Masud, J., Computational Analysis Of NACA Flush Inlet for Secondary Air Source Applications, in 3rd International Bhurban Conference on Applied Sciences and Technology. 2004: Bhurban, Pakistan.
[4] Seddon, J. and E.L. Goldsmith, Practical intake aerodynamic design. 1993, Oxford ; Boston: Blackwell Scientific Publications.
[5] Mossman, E.A. and L.M. Randall, An Experimental Investigation of the Design Variables for NACA Submerged Duct Entrances. 1948.
[6] Frick, C.W., et al., An Experimental Investigation of NACA Submerged Duct Enterances. 1945, Ames Aeronautical Laboratory:Moffett Field, Calif.
[7] ESDU, Drag and pressure recovery characteristics of auxiliary air inlets at subsonic speeds. 2004, ESDU International: London, UK.
[8] Pope, S. B. 2000. Turbulent Flows. Cambridge university press. ISBN 0-521-59125-2.
[9] Sacks, A.H. and J.R. Spreiter, Theoretical Investigation of Submerged Inlets at Low Speeds. 1951.
[10] Frank, J.L., Pressure Distribution and Ram Recovery characteristics of NACA submerged inlets at high subsonic speed. 1950, NACA.
[11] Frank, J.L. and R.A. Taylor, Comparison of Drag, Pressure Recovery, and Surface Pressure of a scoop type inlet and a NACA Submerged Inlet at Transonic Speeds. 1951, NACA.