Numerical Simulation of Uneven Heating of Missile Surface Travelling at Supersonic Speed

#1Kishor Gajanan Malokar, #2Dr. Narendra R. Deore, #3Dr. Mahesh M.S.

[1]krrishnamalokar@gmail.com
[2]naren.deore@gmail.com
[3]maheshms@diat.ac.in

[1]M.E, Heat Power, Pimpri Chinchwad College of Engineering Savitribai Phule Pune University,Pune.
[2]Professor, Department of Mechanical Engineering, Pimpri Chinchwad College of Engineering.
[3]Assistance Professor, Department of Aerospace Engineering, Defence Institute of Advanced Technology, Pune

Abstract: Missile Running At Supersonic Speed Generally Have High Impact Of Fluid Compressibility Over The Body. The High Speed Fluid Particle Transfer The Kinetic Energy Pussesed By The Fluid Particle In Terms Of Heat At The Missile Surface.The Unsteady Flow Around The Missile Is Essential To Understand For Predicting The Aerodynamic Forces And Its Effect. When Missile Runs At High Angle Of Attack Windward Side Have More Friction Force As That Of The Leeward Side Which Results Into The More Heat Generation At Windward Side. This Friction Force Causes The Uneven Temperature Distribution Over Missile Body. The Trajectory Followed Should Be As Accurate As Possible, But The Bending Due To Uneven Heating Will Increase The Drag And Sometime Trajectory Loss Of Missile. Hence, The Presented Work Is Useful To Estimate The Temperature Distribution And Heat Transfer Rate Over The Missile Surface (Windward Side And Leeward Side) With Three Dimensional, Compressible, Stationary, And Viscous Flow. The Physics Of Fluid Behavior Has Been Simulated Using The FLUENT With The Standard K-W Model. Results Obtained Will Useful For Understanding Thermal Effect On Missile Body At Supersonic Speed (Mach 3).

Keywords : Computational Fluid Dynamics ,Aerodynamic Heating ,Compressible Boundary Layer Flow.

Nomenclature

e	Enegy Dissipation Rate(m^2/S^3)
ω	Sp. Turbulence Dissipation Rate(1/S)
k	Turbulence Kinetic Energy(m^2/S^2)
ρ	Density(kg/m^3)
V	Velocity (m/s)
Pr	Pressure (Pa)
T	Temperature (k)
μ	Dynamic Viscocity (N-s/m^2)
Cf	Coefficient of Friction
h	Heat Transfer Coefficient (W/(m^2 k)

I. INTRODUCTION

There has been recent interest in the aerodynamic field, which include the effect of fluid flow over the body and its efficiency. The performance of high speed vehicles (Mis- siles) are largely dependent upon the accurate prediction of flow around the body.The purpose of missile is to minimize the effects of enemy attacks by detecting and destroying en- emy aircraft and missiles approaching a defended area [1]. These systems must be capable of defending strategic areas against attack from high altitude, high speed enemy aircraft capable of taking evasive action while performing precision bombing. Therefore, to intercept targets that can take evasive action, the system must be capable of trajectory corrections after missile launch. Also it is desirable that the system be capable of self-defense against tactical surface targets [2]. At supersonic speed, the temperature rise in the boundary layer is about 600 F and at a Mach number of 5 it is about 1600 F [3]. The unsteady flow around the Missile is essen- tial to understand for predicting the aerodynamic forces and shock formation . Computational flow analysis involves the study of object under moving condition, which gives varia- tion of the thermal properties over the body. The flow sep- aration doesn't allow to study the flow characteristics with time-dependent motion study. In area of missile aerodynam- ics, it is important to select optimum shape of the body, to obtained the streamline flow around the body to reduces the drag associated with the body. Missile travels at high Mach no, so drag and skin friction over the body is totally depend on shape [4]. A missile carries main four components, guid- ance system, flight system, propulsion system and warhead. Supersonic Missile produced the drag component

that results to the shock waves formed [5]. Normally at the forebody (nose section) drag and shock is observed due to the sudden compression to the incoming flow. This shock wave causes the changes in the properties like pressure, temperature and velocity over the missile surface. Since the wave drag may be the prevailing drag form at supersonic speeds, careful se- lection of the nose and tail shapes is mandatory to ensure performance and operation of the over-all system [6]. Ab- dulkareem [7] suggest that curved ogive shape nose gives the streamline flow of particle over the Missile which minimize the strength of shock formation and separation effect from the body surface.

Excessive thermal stress is known to be the main cause of mechanical failure of most long-range, high-speed ballis- tic Missiles [8]. Thus the analysis of the missiles for the abil- ity to accurately predict the aerodynamic heating is essential during its run time. At high angle of attack the separation is observed over a body and this leads to form wake region in the leeward side. This create disturbance to streamline flow and setup asymetricity over the missile body.Simulation over the missile body is carried out at Mach 3 with different an- gle of attack i.e 8^0, 12^0, 16^0 and 20^0 degree to determine the variation of pressure and temperature over the surface. The ongoing study gives the understanding the temperature vari- ation heat transfer over missile and of dealing the fluid flow behavior over 2- Dimensional and 3-Dimensional Missile.

1.1 Numerical Analysis Techniques

The use of computers to solve the Navier-Stokes equa- tions is a major finding for the study of aerodynamic prob- lems. In engineers and researchers were left with only using actual wind tunnel tests and approximations to determine the aerodynamic characteristics under high speed flow [9]. The study of the aerodynamic characteristics is solve using the three approach.

1.Theoretical Approach
2.Experimental Approach (wind tunnel testing) 3.Numerical Analysis (Computational Fluid Dynamics).

The most obvious method is the use of a wind tunnel and an actual model is to be tested. This method gives the most accurate aerodynamic characteristics of the vehicle as real case condition. The Numerical analysis uses software that contains a database of wind tunnel tests and other analytical data to predict the aerodynamic characteristics of the vehicle using the Navier-Stokers equations to be solve in the fluid domain.

II. NUMERICAL ANALYSIS OF FLOW OVER MISSILE SURFACE AT SUPERSONIC SPEED

Long range ballistic missiles (LRBM) traveling at super- sonic speed has to sustain high frictional and drag force due to viscous heating cause to mechanical failure like bending. At different angle of attack missile surface interaction to the fluid is different, thus at higher angle of attack the skin fric- tion is more at the windward side of Missile surface. To es- timate these temperature, pressure and velocity distribution we are solving the fluid domain using CFD. The aerodynam- ics of missile is characterized by multiple shocks and highly unsteady and vertical flow in contact body boundary layers. Numerical simulation of the body gives the understanding of intensity of shock and its location which is challenge in field of aerodynamic to surpass. [10].

1.2 Shock-Induced Separated-Flow Heating

In a supersonic flow, a shock is generated over the sur- face when there is local surface slope increases abruptly. Since the static pressure increases as the shock generated, the higher pressure is fed upstream of the boundary layer, which causes the flow to separate from the body. The in- crease in pressure beyond the boundary leads to more sepa- ration in laminar boundary layer than the turbulent boundary layer because of the molecular random nature [11]. A region of recirculating flow is formed when separation takes place in the fluid adjacent to the wall. The heating in the separated- flow region is increased where the boundary layer reattaches on the compression surface.

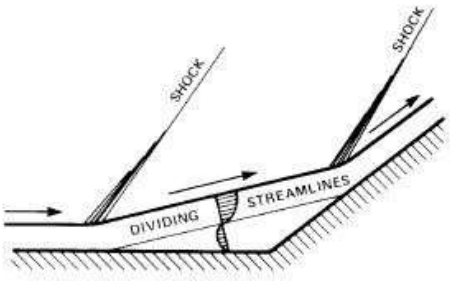

Fig. 1. Supersonic compression corner flow [7]

Shock-induced boundary layer separation and increased local heating also occur when a shock wave impinges on the flow over Missile body.The problem of determining the heat- ing when flow separation occurs has been attacked success- fully numerically in this paper.

1.3 Static Temperature at the wall surface

In a fluid flow domain forced convection is the most sig- nificant factor which affects the heating analysis of a high- speed missile. Missile travels at very high Mach, in which the areas near to the stagnation point generate most of their heat from the compression of the air [3]. Heat energy gen- erated due to the compression of the air will be available to raise the temperature of body and its surroundings though there is conduction along the body surface will occur. This will affect the ability of the boundary layer to transfer heat energy to the missile surface. Thus heat carrying during lam- inar boundary layer is more than that of the turbulent bound- ary layer. The stagnation temperature for the supersonic mis- sile is strongly dependent on the Mach number. The stagna- tion temperature (T_0) is a function of the free stream static temperature T_∞ and Mach number M_∞ only:

$$\frac{T_0}{T_\infty} = 1 + \frac{\gamma - 1}{2} m_\infty^2 \qquad (1)$$

missile areas where impact of stagnation point is less,there viscous heating due to deceleration of air inside the boundary layer. For case of supersonic flow, high-speed boundary layer, near the boundary fluid velocity is almost negligible, fluid particles are brought to rest in a thermo- dynamically irreversible process. Energy of the particle is loss into heat and part is dissipated as viscous work [9]. The boundary layer thickness is very small in comparison to the missile, so the temperature gradient in the boundary layer is very large, especially at the largerMach numbers. The nature of the boundary layer is thinwhich tells that there is little heat generation capacity, this governs the amount of heat is gen-erated over missile surface Conduction of heat through the wall of missile is comparatively than heat generated inside the boundary layer. Just outside of the boundary layer there is recovery temperature is a function of the nearby tempera-ture and Mach number. The recovery temperature is higher for a turbulent boundary layer flow. The recovery tempera-ture (TR) at each location over themissile surface is obtained from the trajectory profile for the baseline model of the mis-sile. The following equation is solved to determine the ratio between the recovery temperature and the stagnation temper-ature (T0) at the boundary layer edge

$$\frac{T_R}{T_0} = \gamma + \frac{1 - r}{1 + \frac{\gamma - 1}{2} m_L^2} \qquad (2)$$

The variable r is defined as recovery parameter, for Laminar flows

$$r = \sqrt{Pr} \qquad (3)$$

and for Turbulent flows

$$r = \sqrt[3]{Pr} \qquad (4)$$

Pr is the Prandtl number. The most of missiles have a relatively short travel time. The heat transfer is takes place in a transient state condition. Missile surface heating/thermal response prediction requires considerations of two different types of prediction method applicable to two different types of surfaces. A thermally thin surface will be approximated as one-dimensional heat transfer with nearly uniform internal temperature that increases with time, Finally it approaching to the recovery temperature.

2.3 Heat transfer coefficient

Estimation of the heat transfer coefficient along the mis-sile body, uses the concept of reference temperature. The ref-erence temperature does become the average of the two, one is the temperature of the wall temperature and other is the low speed flow. This reference temperature gives more accu-rate temperature distribution that occurs within the boundary layer, It also gives impact on the skin temperature of body. The transport properties should be evaluated at the reference temperature, which is defined by Eckert(1955),

$$T^4 = T_L + 0.5(T_s - T_L) + 0.22(T_{AW} - T_L) \quad (5)$$

The Nusselt number can be define as,

$$R_e = \frac{\rho M_L[1 + 0.5(\gamma - 1)M_L{}^2]^{\frac{\gamma+1}{2(\gamma-1)}}}{\mu} x \quad (6)$$

and

$$Pr = \frac{\mu C_p}{k} \quad (7)$$

Thus, by using above formula we can define the convective heat transfer coefficient.

$$h = \frac{k.Nu}{x} \quad (8)$$

Navier stoke equation for fluid flow is solved using simplified finite difference method.

2.4 Skin Friction Drag

Skin friction is important foe all missiles traveling at very high speed, at Mach number 3, skin friction perform equal role in total drag on missile body. The temperature rise due to aerodynamic heating during subsonic speed is not effective but it increases as the speed increases. In steady state condition, the Missile structure will be heated suddenly as there is temperatures conduct the heat from the hot air into the boundary layer. Time required for the heat to flow from boundary layer to the Missile, thus the temperature of boundary layer will be more than the temperature of the mis- sile structure. Heat transfer rate is defined as the the rate at which heat flows from the air to the structure. Thus majorly two factors are important skin drag and form drag, in which form drag reduced the speed of the object and skin friction drag increase the temperature of Missile body by friction.

Skin friction drag and form drag are the over the same body will define accordingly the type of boundary layer. The laminar layer gives skin friction effect less compare to the turbulent flow, and thus laminar flow will have lower rate of heat transfer. The laminar boundary layer produces a skin friction only about one-sixth as great as that produced by the eddying flow of the turbulent boundary layer. Energy dissi- pation rate in the fluid flow is proportional to the skin fric- tion. Since both the skin friction and the rate of heat trans- fer are lower for the laminar than for the turbulent boundary layer, it is essential that the maximum. possible extent of laminar boundary layer be maintained over the surface of a high-speed Missiles[6].

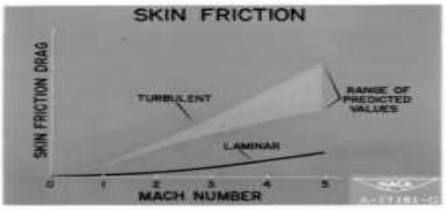

Fig. 2. Skin Friction vs Mach Number for Laminar and Turbulent. [8]

For laminar Flow,

$$C_f = \frac{1.328}{\sqrt{(Re)}} \quad (9)$$

For turbulent Flow,

$$C_f = \frac{0.445}{\sqrt{Re^{2.58}}} \quad (10)$$

Where,

$$Re = \frac{\rho V D}{\mu} \quad (11)$$

Total Drag Force is Calculated as,

$$F = C_f \times \frac{\rho * V^2 * A}{2} \quad (12)$$

3 Governing Equation

General governing equation for fluid flow consider the effect of viscosity, The viscous effects do not influence the basic principle of mass conservation and hence the continuity equation given below is same as for the inviscid flows. However, a moving fluid element have shear and viscous stresses on the surface of the element appearing I momentum and energy equations.

Continuity Equation

$$\frac{\partial \rho}{\partial t} + \nabla(\rho u) = 0 \qquad (13)$$

Momentum Equation

x Momentum :

$$\rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \qquad (14)$$

y Momentum :

$$\rho \frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} \qquad (15)$$

z Momentum :

$$\rho \frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \qquad (16)$$

III. NAVIER-STOKES EQUATION

Moreover, thermal conduction across the surface of the ele-ment provides an additional mode of energy transfer, which appears in the energy equation, the resulting governing equa-tions called the NavierStokes equations. Exact solutions of the complete Navier-Stokes equations for practical problems were virtually non-existent. Even today, Numerical solutions of these equations require a lot of computational power, as well as human resources to generate the computer solutions. Therefore, reasons exist for simpler viscous-flow solutions, by a suitable order-of-magnitude reduction of Navier-Stokes equations, a simpler set of equations. For a viscous flow, the relationships between the nor-mal/shear stresses and the rate of deformation (velocity field variation) can be determined by making a simple assump-tion. That is, the stresses are linearly related to the rate of deformation. The proportional constant for the relation is the dynamic viscosity of the fluid. Based on this, Navier and Stokes derived the famous Navier-Stokes equations which is used in solving fluid flow behaviour and its effect with wall boundary of the body.

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = -\frac{\partial P}{\partial x} + \rho g_x + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \qquad (17)$$

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = -\frac{\partial P}{\partial y} + \rho g_y + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) \qquad (18)$$

$$\rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = -\frac{\partial P}{\partial z} + \rho g_z + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) \qquad (19)$$

We might imagine that, to a reasonable approximation, even the small scale structure of this flow becomes fixed as time progresses. A flow which does not depend on time is known as a steady state flow. In cases where we know, that the fluid flow problem has a steady state solution, we try to determine that solution by solving the Navier Stokes equations and looking at the behavior of the solution as time increases indefinitely [?]. An alternative way to reach the steady state solution is to assume that the variables v, r, and p do not depend on time, plug them into the Navier Stokes equations, for fluid flow simulation over missile body.

IV. NUMERICAL SIMULATION FOR FLUID FLOW OVER MISSILE

Numerical simulation is done in ANSYS-FLUENT Software. The fluid flow over Missile observed at supersonic speed with Mach 3. The procedure setup is detailed below in stepwise. Missile specification foe geometry is taken from literature as shape and the L/D ration for long range ballis-tic missile. The solution for the 2D geometry solve for un-derstanding the physic of flow over missile body and also the pressure and temperature variation have been calculated, further flow has been simulated for the 3 Dimensional Mis-sile geometry with same dimension as given in the 2D case, so that we can estimate and conclude the result obtained foe pressure and temperature contours .The steps followed by the in CFD software are. The methodology used for computation for flow over missile body at Mach within supersonic region is explain and computation is done using CFD software.[8].

1. Constructing the geometry for Missile Body with boundary conditions and initial conditions.

2. Discretized the domain into Small elements with fine grid near the Missile surface.

3. Perform steady-state CFD computation to obtain initial guess in solver setup. Define the initial condition with respect to geometry and computational domain.

4. Solution setup decides the order of method and monitor-ing for fluid domain.

5. Solve for iteration to obtain the result for supersonic flow over missile body.

6. plot and analyses the graph and contour plot available to estimate the aerodynamic properties of Missile traveling at that particular Mach number.

4.1 Computational Domain

Geometry generated in ANSYS FLUENT with fineness ratio between 6 to 8, and other geometric detailed as shown in fig below. The domain selected is sufficiently large to that of the missile body.

4.1.1 Missile Geometric in 2D section with rectangular domain

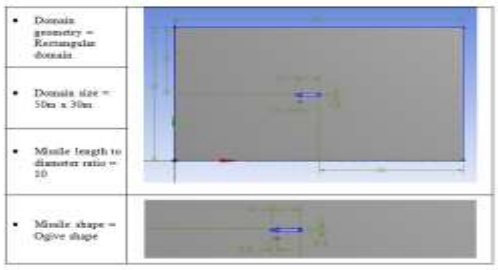

Fig. 3. Missile Geometric in 2D section with rectangular domain.

4.1.2 3D Domain Detailed

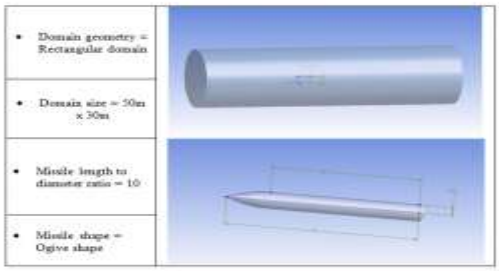

Fig. 4. Missile Geometric in 3D section with circular domain

4.2 Computational Grid

Meshing Specifications for 2D Geometry, The meshing is done as two-dimensional structured in x-y direction giving rise to average quadrilateral cells of about 28430 for all of the Missile geometry.

4.2.1 2D Domain Detailed

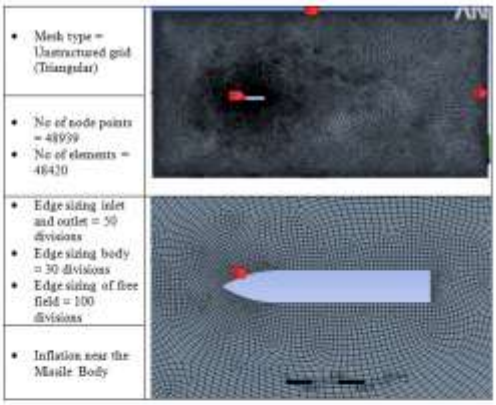

Fig. 5. Meshing in 2D domain

4.2.2 3D Domain Detailed

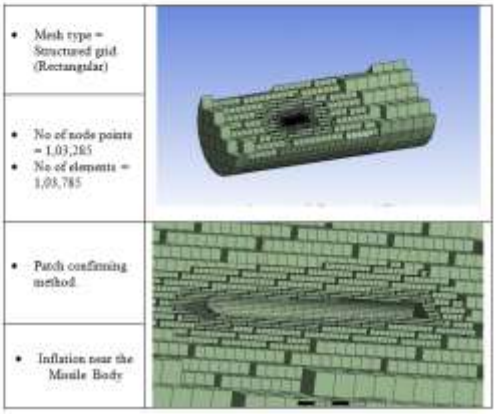

Fig. 6. Meshing in 3D domain

4.3 Computational setup

4.3.1 Boundary Condition

Different boundaries of the physical domain are shown in geometry diagram. The wall of the missile body, is as-sumed to be adiabatic. The free stream pressure is assumed at the inlet and outlet of the domain. The far field pressure condition is used at boundary near the wall.. Finally the sym-metric boundary condition has been implemented.

1. Initial Condition

Inlet as Velocity = 1020 m/s

outlet as Pressure base = 0 Pa.

wall with specific shear at zero value.

2. Solver - Density Base

Specific Heat (Cp) = 1006.43 J/Kgk

3. Model - k-w (Effectively catch wall boundary effects)

V. RESULT AND DISCUSSION

From the CFD simulation it is concluded that at Mach 3 a detached bow shock at the front of the body generates, which highly influence the flow properties around the body. Mach number suddenly decrease drastically behind the wave and flow compressed to a high level at the stagnation point which gives rise to temperature at stagnation point is high, due to this high heat transfer rate is set up between the flow and body. Flowvariables at a point just behind the bow shock wave confirm that at the bow shock wave can be treated as a normal shock. A high-speed flow over a Missile body gen-erates a bow shock wave in front of body, which causes a high surface pressure result into high aerodynamic drag. The static pressure contours show that upper surface and lower surface of the Missile having equal pressure at zero angle of attack, which is not case for higher angle of attack. The vari-ation in velocity, which is caused due to the curvature and shape of the body sections, can be clearly observed from the contours of velocity contours.

1. As the Mach number increases the reattachment of flow length after nose section increases.

2. The Boundary layer region form near the wall surface is difficult to understand since the flow past the body creates turbulence which disturb the flow pattern, which rise to give change of property near wall surface.

3. Skin friction drag associate with the body is larger, which need to reduce so that the effect of heating over missile surface should reduce.

4. The pressure coefficient observations see the transition from laminar to turbulent occurred just after highest point of the peak curve of geometry surface.

5.1 Velocity Contours

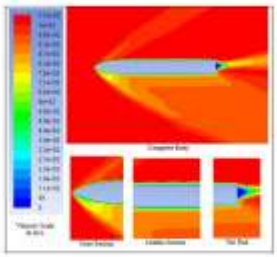

Fig. 7. Velocity Contours Over Missile at angle 08^0

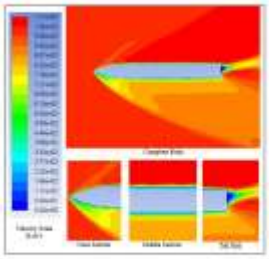

Fig. 8. Velocity Contours Over Missile at angle 12^0

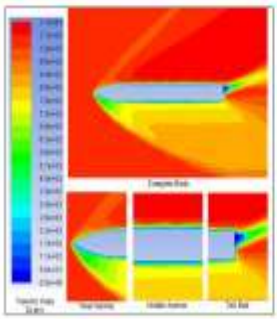

Fig. 9. Velocity Contours Over Missile at angle 16^0

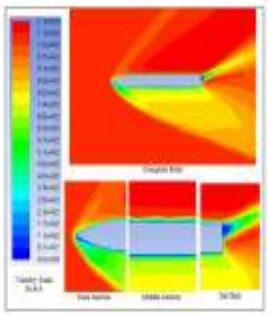

Fig. 10. Velocity Contours Over Missile at angle 20^0

1. Boundary layer formation causes fluid particle to lose its velocity.

2. Velocity reduction is considerable after the shock for-mation.

3. Zero velocity region is available just behind the missile.

4. The Mach flow near to the nose cavity of the spherical model was very low subsonic. An oblique shock region was detected. From the velocity contour over the spherical nose, it is clear that a high Mach region occurs before the nose section. But near to the nose there occurs a region of low Mach flow.

5.2 Pressure Contours

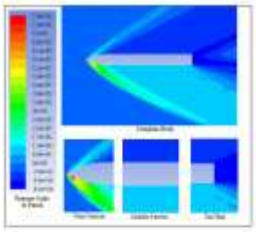

Fig. 11. Pressure Contours Over Missile at angle 08^0

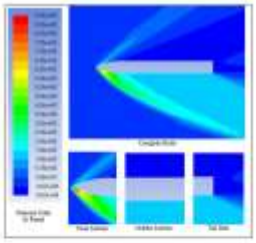

Fig. 12. Pressure Contours Over Missile at angle 12^0

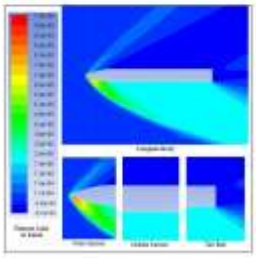

Fig. 13. Pressure Contours Over Missile at angle 16^0

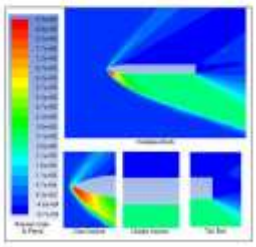

Fig. 14. Pressure Contours Over Missile at angle 20^0

1. Static pressure contours states that there is rise in ad-verse pressure gradient behind the trail edge.

2. Adverse pressure region behind the body lead to unsta-ble the body.

3. Region of high pressure is created before the nose sec-tion.

4. By ideal gas law pressure is directly proportional to tem-perature. So high pressure in the section increases tem-perature near the wall. The pressure contour over the nose curvature spherical model was shown in the above figure.

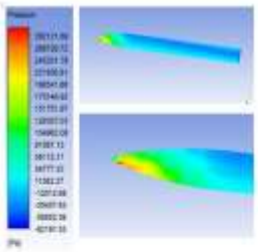

Fig. 15. Pressure Distribution Over 3D Missile Body at 08^0

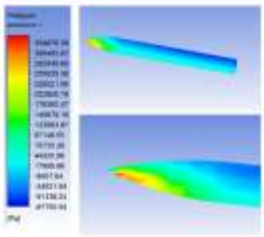

Fig. 16. Pressure Distribution Over 3D Missile Body at 12^0

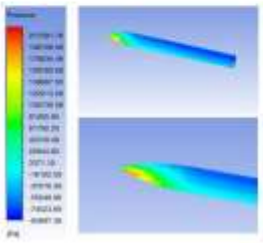

Fig. 17. Pressure Distribution Over 3D Missile Body at 16^0

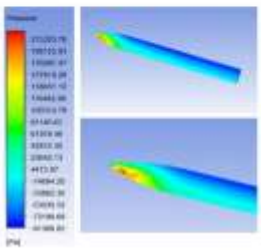

Fig. 18. Pressure Distribution Over 3D Missile Body at 20^0

5.3 Temperature Contours

Heat is produced over missile surface by fluid compres-sion at and near stagnation points,particle carrying the ki-netic energy is loss suddenly when they come in contact with the missile and convert kinetic energy into the heat. Addi-tional heat results from friction along the missile skin inside the boundary layer gives sudden increase in body tempera-ture. Missile when running at 8 0 angle of attack temperature due to compression ans stagnation point is less than that of for the 200 angle of attackwhere the large surface of the body comes in contact with fluid and this lead to the uneven heat-ing of missile body surface causes increase in the thermal stresses in the body. Excessive thermal stress in the body is the main cause of mechanical failure. Temperature rise across the shock is observed, also the wall boundary has higher temperature due skin friction ef-fect. the Skin friction causes highest temperature region just near the surface of body. A region of high temperature is

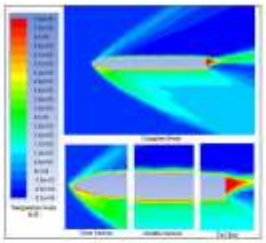

Fig. 19. Temperature Contours Over Missile at angle 08^0

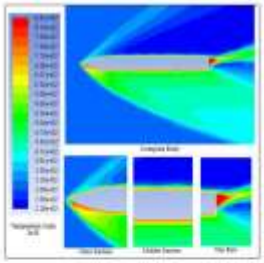

Fig. 20. Temperature Contours Over Missile at angle 12^0

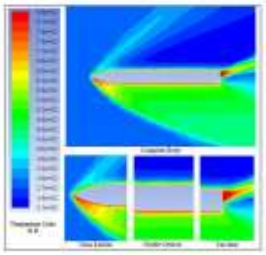

Fig. 21. Temperature Contours Over Missile at angle 16^0

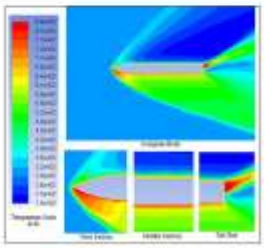

Fig. 22. Temperature Contours Over Missile at angle 20^0

created during a supersonic flow of Mach 3. When the fluid temperature increase, a high temperature gradient set up be-tween body and fluid and this cause high heat transfer rate over the body. Ogive structure selection gives decrease the aerody-namic heating over the body. From the literature study, flow behind the shock wave is rotational, because each stream-line pass through a different strength shock wave and entropy change behind the shock wave is different for every stream-line. Aerodynamic heat in body depends over the ratio of co-efficient of friction and coefficient of drag or temperature rise at the stagnation point over the bodies Compare to all bodies. In the flow field around the Missile body, due to supersonic flow a shock wave generate ahead of the body, called as bow shock .This shock is detached from the body due to the high deflection angle. Pressure gets changes drastically across the shock wave, at the stagnation point pressure is at peak value because at the stagnation pressure shock wave is normal to the body. Area of the contour shows the sonic region here velocity of the flow is subsonic.

VII. CONCLUSION

In this present study the flow over missile surface has been solve for different angle of attack at supersonic speed. The results are validated by grid independent test as well as the author T.F.Zien and W.C.Ragsdale presented the empiri-cal study on tactical missiles. However aerodynamic heating at different altitude is not able predict exactly. The valida-tion to the missile heating is not perfect agreement to the results obtained from the Simulation, the true comparison come only with the experimental data. The results from this study gives close approximation to temperature and shock relative to the missile body which can be used in the future for missile uneven heating problems studies. The capabili-ties of numerical simulation study is able to give the solution for complex flow behavior over missile surface. In future similar study of uneven missile heating will be conducted for time dependent flow analysis, and allows the study of the hypersonic missiles.

REFERENCES

[1] Cohen, J., 1951. “Aerodynamic characteristics of four bodies of revolution showing some effects of afterbod shape and fineness ratio at free-stream at mach num-bers from 1.5 to 1.99”. NACA research memorandum, 15(03), pp. 483–500.

[2] Gendell, S., 1990. “The fluid-structure interaction of an elastically mounted ogive cylinder at high angle of attack”. AIAA Paper, 28.

[3] Jules, B. “Crossflow effects on steady and fluctuating pressures on an ogive-cylinder cone-frustum model in supersonic separated flow”. NASA Technical Paper.

[4] Jackson, C., 1991. “Bodies with noncircular cross-sections and bank-to-turn missiles”. Journal of Aero-nautics, 41, pp. 365–389.

[5] Allen, M., 1951. “Characteristics of flow over inclined bodies of revolution naca rm a50l07”. IOSR Journal of Mechanical and Civil Engineering, 11(5), pp. 173–197.

[6] Lee, G., 2013. “Aerodynamic performance analysis of bodies with different cross-sections”. International Journal of Heat and Fluid Flow, 42, pp. 173–197.

[7] Abdulkareem, S., 2008. “Effect of body shape on the aerodynamics of projectiles at supersonic speeds”. Journal Of Engineering Science And Technology, 03(03), p. 278 292.

[8] atabi Mushtak, A., 2008. “Effect of body shape on the aerodynamics of projectiles at supersonic speeds”. Journal of Engineering Science and Technology, 2(03), p. 278 292.

[9] Gates, R., 1998. “Aerodynamic test and analysis of a slender generic missile configuration”. Air Force Ar-mament Laboratory, 32, p. 010.

[10] Coe, C., 1981. “Crossflow effects on steady and fluc-tuating pressures on an ogive-cylinder cone-frustum model in supersonic separated flow”. NASA Technical Paper, 40.

[11] Maitra, S., 2000. “Aerodynamic heating of ballistic missile including the effects of gravity”. International Journal of Heat and Fluid Flow, 25(03), p. 36.

