Simulation and Analysis of Protrusions on Flat Plate at Hypersonic Speeds

M. Aathira Nair* and S. R. Nagaraja

Department of Mechanical Engineering Amrita School of Engineering, Bengaluru Amrita Vishwa Vidyapeetham Amrita University, India; aathinair8@gmail.com, sr_nagaraja@blr.amrita.edu

Abstract

This paper presents investigation into the complex phenomena that occur in the vicinity of three dimensional forward facing steps mounted on a flat plate in hypersonic flow of Mach number 8. The dependence of the flow field on geometry parameters like height of protrusion and deflection angle is also studied. Vortices generated at the separation location seemed to wrap around the protuberance, causing high surface heating in the separated region, with the hotspot at the foot of the shock. For lower deflection angle model, the protuberance did not have much effect upstream, but turbulence was observed to the side of the protuberance. The flow remained more or less attached, suggesting nominal heating in the front of the protrusion. As the deflection angle increased, separation phenomena was more visible and hence the recirculation zone. For un-separated flow cases, the protrusion height had less effect on the surface heating. But when the flow was separated, the extent of flow separation region was large, indicating higher surface heating.

Keywords: Deflection Angle, Hypersonic , Protuberance, Re-Circulation, Surface Heating, Vortices

1. Introduction

Aerodynamic vehicle surfaces are often characterized with surface discontinuities that originate from fabrication tolerances, sensor installations, piping and so forth located on the exterior portion of the vehicles. Further, there shall also be discontinuities like sheet joints, control surfaces etc, producing protuberances partially or completely immersed in the vehicle boundary layer. In the practical scenario, these discontinuities are most often small in dimension, and are comparable to the local boundary layer thickness in case of low speed flows. In high speed flows the boundary layer is much thinner and the height of protrusions (h) may not be very small compared to the local boundary layer thickness δ . Hence the presence of the protrusions could have severe effects on the flow. The flow features are dependent on several factors like size and geometry of protrusion, flow velocity, Reynolds number and type of the boundary layer. The interference invariably causes high rates of surface heat flux and therefore requires adequate shielding. The interference caused to the boundary layer by the flow causes the generation of local hot spots in the vicinity of the protuberances, which if not properly mapped, could prove fatal.

Owing to their potential for significant enhancement of local heat and pressure loads in the vicinity, interference interactions due to such surface discontinuities have been the subject of several investigations [1-9] for several decades. Presence of a protrusion in the flow field generates a separation bubble ahead of it, which is bounded by a separation shock. This separation shock causes the boundary layer to lift off the surface of the plate, the upper part of which gets deflected over the protuberance, whereas the lower part meets the protrusion, creates a separation zone of high vorticity, and then escapes to one side of the protuberance. The re-circulating flow sets up a vortex that wraps around protuberance causing increased heat flux in the front as well as at the sides. The vortices generated at this location eventually wrap around the protuberance and attain a horseshoe orientation. The interference interaction protrusions can be classified as subcritical and supercritical interactions based on the angle of the protuberance (α). For a given height of protuberance, there exists a critical lip deflection angle below which the interactions are subcritical, where no substantial recirculation zone is generated upstream of the protrusion. Supercritical interactions, where the deflection angle is greater than the critical value, are characterized by a distinct recirculation region immediately upstream of the protuberance. In the former, the hottest point generally appears to the side of the protuberance, whereas in the latter, the hot spot is exhibited within the separated region, immediately upstream of the protuberance, close to its foot.

B. Burbank et al¹ have investigated heat transfer distributions in partially and fully immersed turbulent boundary layer in the vicinity of two dimensional surface projections. The extent of the separations caused by these projections, both upstream and downstream as well as the magnitude of the resultant interference heat transfer coefficient were found dependent on the size and cross-sectional shape of the projection, Mach number, Reynolds number and the local boundary layer thickness. They also observed that the upstream heat transfer reduced with decreasing the inclination of the front face and the windward interference heating rates increase with decreasing boundary layer thickness. The effects of Mach number and Reynolds number were confined to the immediate vicinity of the windward face of the projection. The lambda footed shock formed at the foot of the projections generate a localized region of high heating, the location of which is dependent upon Mach number and boundary layer thickness. The size of this interaction region also increases with increase in the frontal area of the protrusion.

Estruch et al² observed that the local interferenceinteractions are dominated by the incipient separation angle separated by the protuberance. The local heating of the vehicle surface depends on whether the boundary layer separates ahead of the protuberance or not. The dependence of maximum heating on the incoming boundary layer state was negligible in both unseparated and fully separated interactions. Investigations conducted by K. P Reddy et al³⁻⁴ around such supercritical protrusions with deflection angles of 60°, 90°, and 120° also reported highest heat flux within the recirculation region and enhanced heat flux in comparison to the undisturbed value, to the side of the protuberance, close to it. This hot spot formation was accounted to the generation of vortices immediately upstream of the protuberance that wraps around it in a horse-shoe fashion. C.S. Kumar and K.P. Reddy⁵, in their study of flow dynamics in the vicinity of three-dimensional forward facing steps mounted on a sharp cone, experimentally confirmed this hot-test spot around the protuberance at its foot, within the recirculation region upstream of it. This was the location where the vortex action caused the flow to reattach on the cone surface. Rudolph A. King et al⁶ et al have experimentally studied the effect of shape of the protuberances on hypersonic boundary layer. The experiments are conducted on shuttle orbiter models at Mach 6. Frank K. Lu^Z has studied the flow over sub boundary layer protuberances at Mach 2.5. He has visualized the boundary layer interactions for flow over micro vortex generators. They have found that separated flow from the micro vortex generators forms weak horseshoe vortex. Ahmet Selim Durna et al⁸ have done computational study of shock and boundary layer interactions over double wedges in Mach 7 flow. They have observed a strong interaction between the deformation of boundary layer and the bow shock and transmitted shock. Hussain H. Al-Kayiem et al⁹ have developed finite difference based CFD methods solving compressible flow over missiles. They have used MacCormack's explicit method for solving the finite difference equations and have found that the temperature and pressure gradients are very large at the leading edge of the missile.

This paper presents an investigation into the complex phenomena that occur in the vicinity of three-dimensional forward facing steps mounted on a flat plate in hypersonic flow (M=8). Using ANSYS-FLUENT, the flow field around a single step protrusion model, and its dependence on the geometry parameters like height of protrusion and deflection angle, are studied.

2. Methodology

In order to understand flow physics over a rectangular protrusion on a flat plate in hypersonic flow (Mach number =8) computational studies were conducted. The investigation was carried out using the commercial computational fluid dynamics (CFD) software ANSYS-FLUENT. Due to relative simplicity of implementation and reasonability of predictions with easy convergence, k- ω model is widely used for high turbulence flows. For capturing boundary layers under high adverse pressure gradient, realizable k- ε model is used. This model has same turbulent kinetic energy equation as that of the standard k- ε model but has an improved equation for ε .

2.1 Model

To enhance the computational efficacy, only a symmetric half of each model was modeled as shown in Figure 1. The base model was flat-plate with a protrusion of height h= 20mm and angle θ =90°. An enclosure with inlet length 100mm, exit length 50mm, height 100 mm and width 100 mm was created around the flat-plate surface, with XY and XZ as the symmetry planes (of the enclosure). In order to study the effect of the protrusion height on the flow field, models with different protrusion heights were simulated (h= 10mm, 15mm and 20 mm). To study the effect of protrusion angle models with θ =30°, 45° and 90° were created.



Figure 1. Model 20mm protrusion height and $\theta = 90^{\circ}$

2.2 Mesh

The domain chosen over the model surface was meshed throughout with coarse mesh, limiting the maximum element size 250mm Figure 2. As the analysis focuses on the effect of the protrusion on the surface of the flat-plate, surface sizing (element size 0.002m) was provided for the entire model surface in order to capture the flow phenomena better. To resolve the features in the boundary layer accurately, the mesh was made denser (0.002m-0.003m) to a height of 45mm over and near the protrusion surface. A surface inflation could have been provided on the surface to capture the boundary layer phenomena more precisely, but this resulted in stepped mesh creation at the location and had to be avoided. Due to technical limitations, further refinement of the mesh was not possible and hence the total number of mesh elements had to be limited to 10, 00,000 -12, 00,000 elements.



Figure 2. 3D Model before and after meshing

2.3 Solver

Since the geometry had very small dimensions for the protrusion compared to rest of the domain and very small mesh sizes at certain locations, to capture these, double precision was used while solving. Taking up to 16 decimal places, double precision uses up double the system memory and takes twice the solving time. To accommodate for the compressible effects at high speeds, a steady, density-based solver with absolute velocity formulation was chosen. Simulations were carried out on the meshed model using a realizable k- ε model, with air as the test fluid. In order to accommodate the viscous heating that occurs, the energy equation was also included. As complicated near-wall effects were expected, enhanced wall functions with pressure gradients and thermal gradients were chosen as they perform better for adverse pressure gradients and cope better with flow separation and reattachment. Though this would require 10-20 layers of boundary layer mesh, due to step mesh creation at the location, mesh was limited to 5-6 layers. Each of the six faces was allotted a boundary condition as listed in Table1.

The inlet was given pressure far-field condition as to resemble the actual flight conditions, with 50,000Pa and Mach Number 8. The pressure outlet had pressure 10 Pa, turbulence intensity 5% and turbulent viscosity ratio 10. The model surface including the protrusion was tagged as the lower wall, with no slip condition pertaining to the shear forces acting. It was also assigned an isothermal wall condition at room temperature. Right side of the model is the symmetry plane, which is a wall with zero shear. The top and left sides were assigned pressure far-field conditions. Both the top and left surfaces were confirmed to be far enough away from the interference interaction region that their effects were not felt at the protrusion station. A cell based least squares method was used for spatial discretization and an implicit second order upwind methodology was assumed for calculations.

Face name	Туре
Inlet	Pressure far-field
Outlet	Pressure outlet
Тор	Pressure far-field
Bottom	Wall
Left side	Pressure far-field

 Table 1.
 Boundary condition for faces

The inlet was given pressure far-field condition as to resemble the actual flight conditions, with 50,000Pa and Mach Number 8. The pressure outlet had pressure 10 Pa, turbulence intensity 5% and turbulent viscosity ratio 10. The model surface including the protrusion was tagged as the lower wall, with no slip condition pertaining to the shear forces acting. It was also assigned an isothermal wall condition at room temperature. Right side of the model is the symmetry plane, which is a wall with zero shear. The top and left sides were assigned pressure far-field conditions. Both the top and left surfaces were confirmed to be far enough away from the interference interaction region that their effects were not felt at the protrusion station. A cell based least squares method was used for spatial discretization and an implicit second order upwind methodology was assumed for calculations.

3. Results and Discussions

3.1 Model 1: Protrusion Height 20mm, $\theta = 90^{\circ}$

The flow field captured has a separation shock in front of the protrusion, extending upstream for a separation distance 5-6 times that of the protrusion height Figure 3. The region bordered by the shock exhibits flow re-circulation, with the vortices wrapping around the protrusion and extending downstream. The Mach number plots showed similar trend, with least velocities in the dead air region, right in front of the protrusion.



Figure 3. Pressure contours and mach number contours of model 1

3.2 Model 2: Protrusion Height 10mm, θ = 90 Degree

The captured flow field has a separation shock in front of the protrusion, as well as a leading-edge shock. The extent of separation region upstream of the protrusion is comparatively smaller than 20 mm model Fig ure 4. A lip shock off the protrusion edge exists, though not very clearly captured. The recirculation zone is also small suggesting higher surface heating confined to only this region.



Figure 4. Temperature and density contours for Model 2

3.3 Model 3: Protrusion Height 15mm, θ = 90 Degree

The flow field is more disturbed than in case of a 10mm protrusion, as shown by a larger separation region Figure 5a. The lip shock was not captured but the velocity vectors depict the recirculation of flow occurring in the vicinity of the protrusion. Higher surface heating is suggestive in this region with a hot spot right at the foot of the protrusion Figure 5b. The path lines display how the flow interacts with the protuberance, creating vortices in front of it which later wraps around and moves downstream.



Figure 5a. Path lines for model 3



Figure 5b. Temperature contour for model 3

3.4 Model 4: Protrusion Height 20mm, Deflection Angle 30°

The captured flow field shows the lip shock and the leading edge shock Figure 6. The flow remains more or less attached to the surface as the deflection angle is only slightly above the critical angle. Surface heating is also limited due to the reduced separation effect. The path lines show the flow to be sticking on the surface, only deflected upwards by the 30 degree protrusion.



Figure 6. Velocity vectors of Model 4

3.5 Model 5: Protrusion Height 20mm, Deflection Angle 45°

Density contours reveal a lip shock and the leading edge separation shock Figure 7. The flow does not stick on to the surface as much as it did for the 30 degree case. Here, there is a clear separation region, ahead of the protrusion region. Increased surface heating is also suggested owing to the presence of vortices and re-circulation.



Figure 7. Density and Temperature contours for Model 4

4. Conclusions

Supercritical interference interactions in the vicinity of short surface protuberances were studied on 5 models using commercial CFD software, ANSYS 15. Three different protrusion heights were examined to understand the effect of 'h/ δ ' on the flow field. Three configurations with deflection angles 30°, 45° and 90° were analyzed to study the effect of 'a' on the flow field.

A complicated re-circulation flow was observed in the front of the protuberance, accompanied by a separation shock. Vortices generated at this location seem to wrap around the protuberance, causing high surface heating in the separated region, with the hotspot at the foot of the shock. For lower deflection angle model, the protuberance does not have much effect upstream, but turbulence was observed to the side of the protuberance. The flow remains more or less attached, suggesting nominal heating in the front of the protrusion. As the deflection angle increases, separation phenomena is more visible and hence the re-circulation zone. These cases would have high heating zones right in front of the protrusion, apart from having increasing heating to the sides. For unseparated flow cases, the protrusion height seems to have less effect on the surface heating. But when the flow is separated, the extent of flow separation region is large, indicating higher surface heating.

5. References

- 1. Paige B Burbank, Robert A Newlander, and Ida K Collin., Heat Transfer and Pressure Measurements on a flat plate surface and Heat Transfer Measurements on attached protuberances in a Supersonic turbulent boundary layer at Mach Numbers 2.65, 3.51 and 4.44.December 1962; NASA Technical Note. TN D-1372.
- 2. Estruch D, MacManus D G, Stollery J L, Lawson N J, Garry K P, Hypersonic interference heating in the vicinity of surface protuberances. Exp. Fluids, 2010; **49**:683-699.
- 3. Kumar CS, Reddy KPJ. Experimental Investigation of Heat Fluxes in the Vicinity of Protuberances on a Flat Plate at Hypersonic Speeds. J. of Heat Transfer. 2013;135(12): 121701-9.
- 4. Kumar CS, Singh T, Reddy KPJ., Investigation of the separated region ahead of three-dimensional protuberances on plates and cones in hypersonic flaws with laminar boundary layers. Physics of Fluids. 2014;**26**:126101-1.
- Kumar CS and Reddy KPJ., Hypersonic interference heating on cones with short three-dimensional protuberances. J. of Experimental Thermal and Fluid Science. 2014; 55(2014): 29-41.
- Rudolph A. King, Scott A. Berry, Michael A. Kegerise., Effect of protuberance shape and orientation on Space Shuttle Orbiter boundary-Layer Transition., NASA Technical Memorandum. TM-215103. 2008.
- Frank K. Lu., Visualization of supersonic flow around a sharp-edged sub-boundary-layer protuberance., Journal of Visualization.2015; 18(4):619–29

- 8. Ahmet Selim Durna, Mohamad El Hajj Ali Barada, and Bayram Celik. Shock interaction mechanisms on a double wedge at Mach 7., Physics of Fluids., 2016; **28:** 096101. doi: 10.1063/1.4961571.
- 9. Hussain H. Al-Kayiem, Ahmed K. Hussein, Jalal M. Jaleel and Salam H. Hussain., Numerical Computation

of 3-DimensionalSupersonic Flow Field over Seamless Missiles, Indian Journal of Science and Technology., 2014; 7(10): 1563–72.