PAPER • **OPEN ACCESS**

Transient Analysis of a Blunt Body with Cylindrical Protrusion in Hypersonic Flow

To cite this article: N Akshay and S. R Nagaraja 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **577** 012178

IOP Conf. Series: Materials Science and Engineering **577** (2019) 012178 doi:10.1088/1757-899X/577/1/012178

Transient Analysis of a Blunt Body with Cylindrical Protrusion in Hypersonic Flow

Akshay.N[1], Nagaraja S. R[2]

[1,2]Department of Mechanical Engineering, Amrita School of Engineering, Bengaluru, Amrita Vishwa Vidyapeetham, India.
Email: sr_nagaraja@blr.amrita.edu

Abstract. Numerical analysis of a blunt body with protrusion in hypersonic flow having Mach number 8 is analysed using ANSYS-FLUENT. Transient analysis gives a better understanding of the flow field, separation of the flow from body, time taken for achieving steady state. 3D analysis has done. The variations of the pressure with respect to time at different locations on the body are obtained. The peak pressure at the separation point is 38% more than the steady state pressure. Pressure fluctuates inside the separation region.

1. Introduction

Protrusions exist on the surfaces of almost all aerodynamic bodies. The protrusions can be due to fuel lines, electrical cables, control surfaces, rivets etc. Such protrusions induce aerodynamic and heating loads on the surface of the aerodynamic bodies. The hot spots were form in the vicinity of these protrusions and proper thermal shielding needs to done on the surfaces. Experimental and numerical studies can identify these hot spots and their behaviour. The pressure distribution over the protrusion gives a better idea of the hypersonic flow. Li, S.-X et al. [1] conducted an experimental investigation of Mach 5 flow over a flat plate with rectangular protrusion. The value of H/D is contributing a major part in the flow field features if H/D value is less than 1.5, whereas if H/D is more than 2 then flow field features are independent of the parameter H/D. Between the above cases, the flow field features are in a transition region, i.e. changing with H/D. In addition, the peak value of the pressure is somewhere at the top of the protrusion which is just above the triple point. C.S Kumar et al. [2] flow separation in the vicinity of protrusions on rectangular plates and cones. The separation distance in the case of a flat plate observed to be between 10.5 to 12 times of the protrusion height. For the cone, it was 9 to 10.5 times. The separation distance rises with increase in protuberance height. However, the pressure variation in both flat plate and cone is same for experimental and computational results.C.S Kumar et al. [3] have done heat flux measurement around a protrusion in a flat plate for two different enthalpy vales and three different deflection angles.It was observed that the maximum heat flux value is at the corner of the protrusion foot. In three different deflection angles 60^{o}, 90 ^{o}, 120^{o} the higher value of the hot sport is for 120^{o}. Aathira Nair et al. [4] have studied the flow field over rectangular protrusions over flat plate by varying different parameters like Mach number, distance from leading edge, height of the protrusion, and deflection angle of protrusion. The flow will re-circulate in the downstream of the separated shock and there was hot spot on the surface due to the shock. The magnitude is lesser for smaller deflection angles of the protrusion.

There are some advantages of protrusions, like using protrusions, we can create pitching moment on the body and able to control the body with adjusting the height of this protrusion.Salimuddin Zahir [5] did a detailed computational study on aerodynamic flow field for hypersonic free stream interaction with cone geometries installed with lateral pin-protuberance. The steerage capability obtained by, changing the height of the protrusion creates a blockage in the flow. R.Sriram et al. [6] mainly concentrated on the cooling phenomena on the hypersonic flight by introducing array of jets at the leading edge of the body. This makes an effective reduction in the heating of the body. BallaVenukumar et al. [7] have experimentally studied drag reduction using jet at the leading edge. In the both works, the blunt body geometry is same as the geometry used in this work. Another important parameter is the shockwave boundary layer interaction, controlling of shockwave boundary layer interaction effects could do with the help of the protrusion. Frank K. Lu [8] used a micro vortex generator that can be considering as a protuberance in the flow field. The visualization of the flow gives a clear idea of how the protrusion effects the interaction of shock wave and boundary layer using two types of vortices. In this paper variation of flow parameters with time, over a blunt cone with protrusions in hypersonic flow presented.

2. Simulation methodology

The computational fluid dynamics software ANSYS Fluent used for the simulation. As the flow is compressible and hence density based solver is used. The turbulence is modelled using two-equation k-ω model. The model predicts turbulence by two partial differential equations for two variables, k and ω, which are the turbulence kinetic energy and specific rate of dissipation respectively.

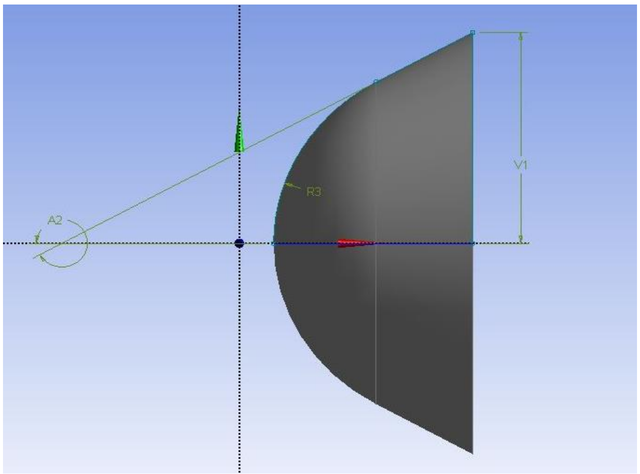

Figure 1: Geometry of the Blunt Body

2.1. Geometry

The geometry used for the simulation is similar to the realistic space vehicle's nose part. The blunt cone model has 80 mm base diameter and cone radius of 35 mm. The apex angle of the blunt body is 58^0. The disturbance, which is a pin protuberance, is at 25 mm from the leading edge. The height and diameter of the protrusion are same and is 2.5mm.The enclosure for flow simulation over the body is a user defined one which having the similar shape as that of the blunt body. This reduces the number of elements in the mesh. The enclosure is having base diameter of 160 mm, blunt cone radius of 70 mm and the total length of the enclosure from leading edge to trailing edge is around 120 mm. The parts of the geometry have been named as inlet, outlet and blunt body. Inlet is the front section of the blunt enclosure used to define the inlet conditions and outlet is the backside of the enclosure used to define outlet conditions. The geometry of the object is as shown in figure 1.

2.2. Meshing of the geometry

Triangular mesh with an element size of 1mm is used. At the surface of the body, an inflation of 0.1 mm for 40 layers is used. The total number of elements is 3536255 in the entire domain. The convergence used is criterion is 10^{-3}.

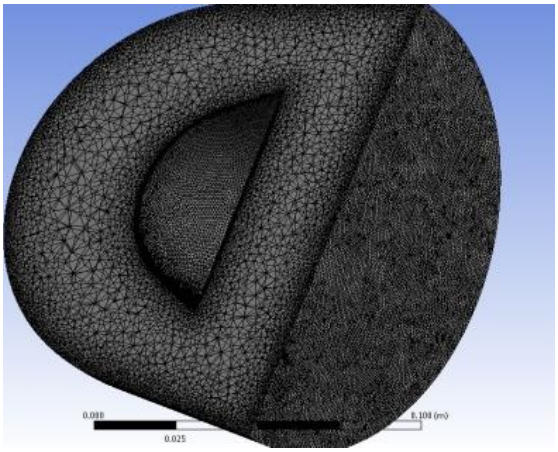

Figure 2: Meshed Geometry

2.3. Boundary condition and initialization

While solving double precision is used because the geometry itself very small and some places of the body having very small mesh size. However, the problem is that this double precision uses up double the system memory and the solving time.

Table 1: Boundary Conditions

Face name	Type of boundary
Inlet	Pressure far field
Outlet	Pressure far field
Blunt body	Wall

The above table gives the boundary conditions provided for each face of the geometry. The inlet and outlet gives pressure far field having,

Pressure = 10kPa
Mach number = 8
Temperature = 100 K

IOP Conf. Series: Materials Science and Engineering **577** (2019) 012178 doi:10.1088/1757-899X/577/1/012178

The surface of the blunt body considered as wall with constant wall temperature of 300K. The fluid is air, and obeys ideal gas equation. Sutherland's law adopted for finding the viscosity, which is a three co-efficient viscosity model. The numerical solution method used for the simulation is least square upwind method. The formulation is on the implicit scheme with a second order upwind method.

3. Results and discussions

The flow over the blunt body at a hypersonic Mach number creates a bow shock wave in front of the body. The flow properties behind the bow shock are different from the free stream properties. The flow approaches the protrusion with reduced Mach number than the free stream. This is unlike in the case of rectangular protrusions over flat plates where the free stream flow conditions and flow conditions over the protrusion are same. The flow properties over the protrusion depend on the location of the protrusion in case of blunt body. Flow separates in front of the protrusion with a separation shock and lip shock on the protrusion. The separation distance from the protrusion depends on the height of the protrusion. A recirculation or dead air region formed in front of the protrusion. In the present analysis protrusion, height equals with the diameter is used.

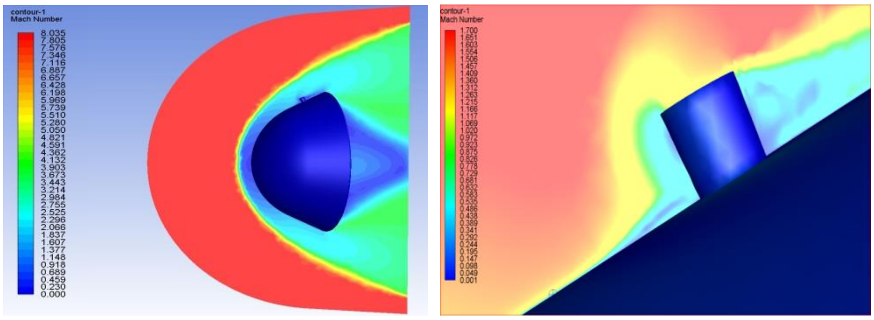

Figure 3: Mach contour

Transient analysis will help for better understanding of the flow field dynamics, separation of the flow from body, time taken to achieve steady state. In addition, this analysis helps to visualize the flow field variation in the dead air region. The complete recirculation flow with respect to each time interval shown in figure 4, 5 and 6.The flow is analysed for 0.5 milliseconds. At the beginning of flow, there is sudden increase of pressure and heat flux over the protrusion.

a **b**

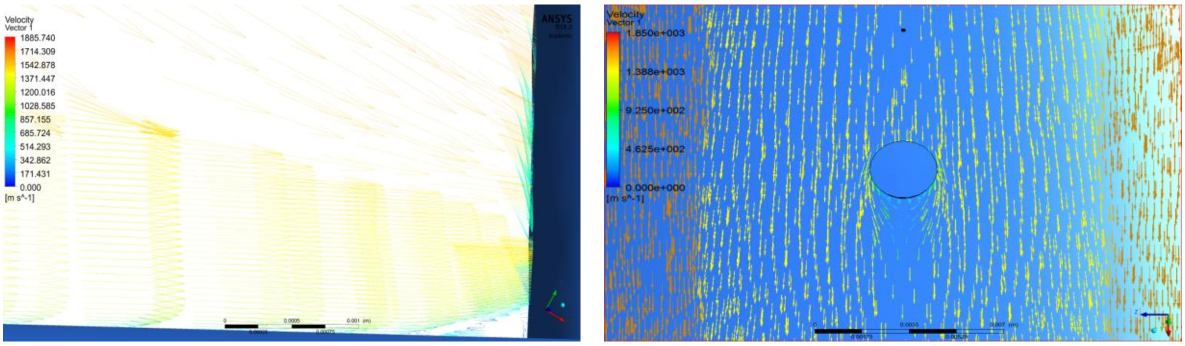

Figure 4: At 10 microseconds **a** side view, **b.** top view

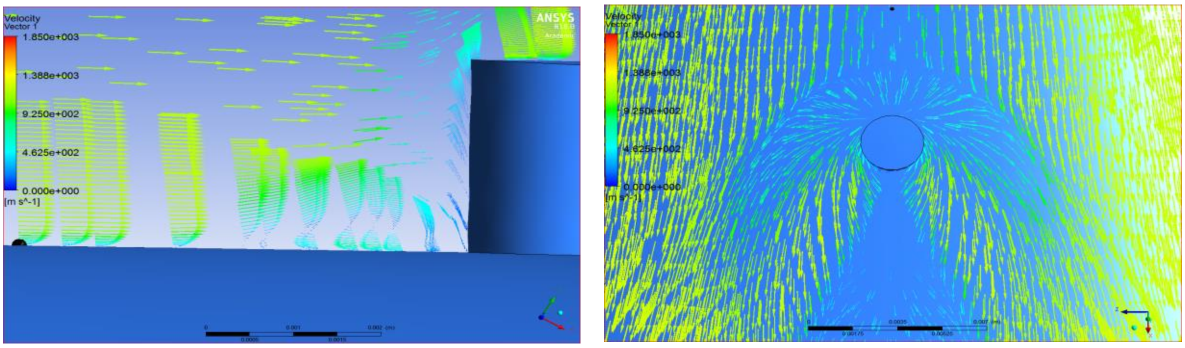

Figure 5: At 60 microseconds **a** side view, **b.** top view

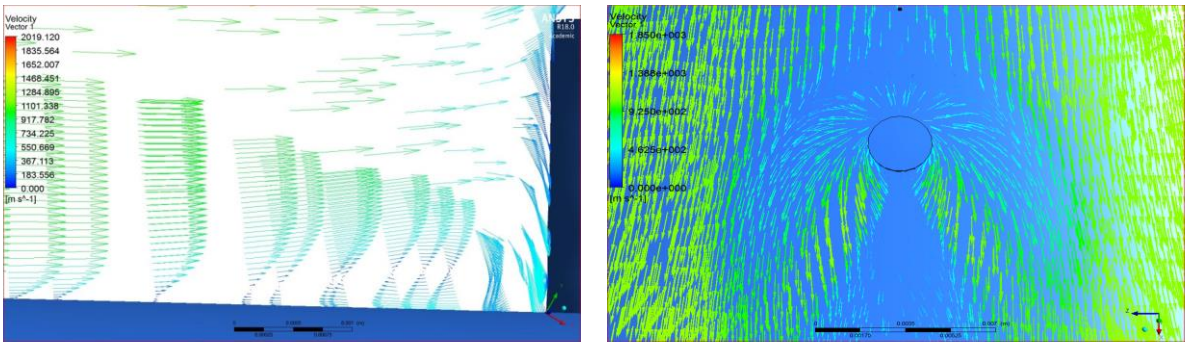

Figure 6: At 120 microseconds **a** side view, **b.** top view

The variation in the size of the dead air region with respect to time can be clearly seen in the velocity vectors of both views. The thickness of this region is reducing with respect to increase in the height. The figures b is top views of planes at the bottom of the protrusion at different period of time. As the height increases, the flow will attached with the sides of the protrusion. At the top, the flow will completely have attached and wrap around the pin protuberance smoothly. The variations of the pressure and heat flux with respect to time at three points on the protrusion are in the figures 7 and 8 respectively. Initially there is huge value of the pressure and heat flux at these three points then it drastically drops and attains a constant value. This happens in a very small interval of time. Among the three points, the maximum value is at the top most point in the protrusion.

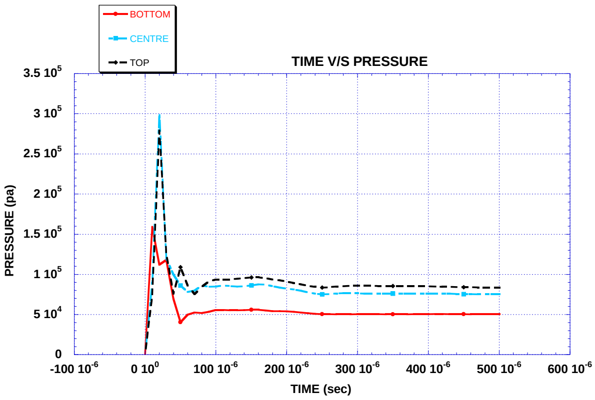

Figure 7: Time v/s Pressure

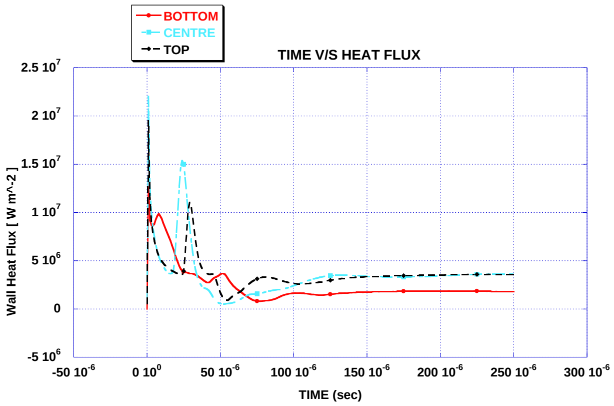

Figure 8: Time v/s Heat Flux

The flow separation point is at a distance of 5.1 mm from the foot of the protrusion. This value obtained after the flow become steady, before that, we cannot predict the separation point using pressure values over the body. The figure 9 shows the variation of the pressure at the separation point with respect to time. It can be seen that the value of pressure increases at the initial stage then it reduces. The pressure fluctuates and attains a steady value after 150 microseconds. But the variation of pressure inside the dead air region is difficult to predict. Figure 10 shows the pressure variation of a point inside the dead air region. The pressure is varying continuously, but with decreasing amplitude. From this plot, it is clear that the pressure at the dead air region is not attaining steady state even after the external flow attains steady state.

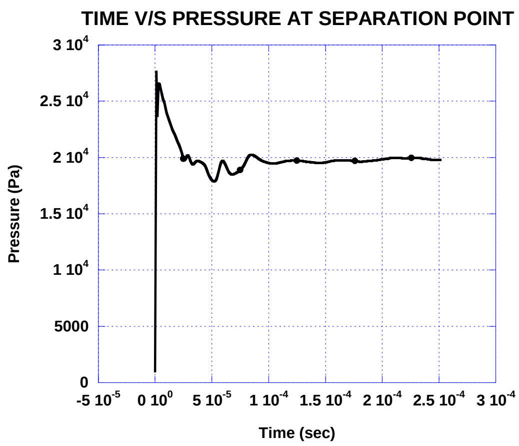

Figure 9: Time v/s Pressure at separation point

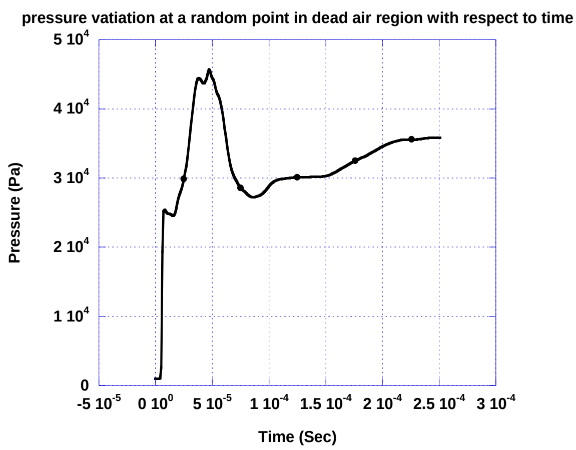

Figure 10: Time v/s Pressure at a random point in dead air region

4. Conclusion

The transient analysis gives the clear visualization of the recirculation and the vortex generation. The value of pressure and heat flux at the initial time was very high then it reduces. The comparison of three different points on the protrusion gives a clear idea of the position of the highest hot spot and highest pressure point. The separation point found at 5.12 mm from the protrusion but the position of separation varies until the flow field become steady. Moreover, the pressure value in the dead air region continuously varies even after the flow field becomes in steady, due to the strong vortex formation.

5. Reference

[1]. Li, S.-X., Chen, Y-K, Ni, Z.-Y2001 “Flow field Features on Hypersonic Flow over Rectangular Obstacles”. *Journal of Visualization*.**4**73-79
[2]. Kumar C S, Singh T, Reddy K P J 2014 “Investigation of the separated region ahead of three dimensional protuberances on plates and cones in hypersonic flaws with laminar boundary layers” *Physics of fluids*.**26**, 126101.
[3]. Kumar C S, Reddy K P J 2013 “Experimental Investigation of Heat Fluxes in the Vicinity of Protuberances on a Flat Plate at Hypersonic Speeds” *Journal of Heat Transfer*. **135**, 121701-7
[4]. Ms. Aathira Nair and S. R. Nagaraja 2016 “Simulation and analysis of protrusions on flat plate at hypersonic speeds” *Indian journal of science and technology*, **9** (45)
[5]. Mr. SalimuddinZahir 2014 “Computational aerodynamic interaction of hypersonic flow with pin-protuberance on blunted cones” *Aarab. J Sci. Eng*. **39**7275–7288.
[6]. R. Sriram, G. Jagadeesh2009 “Film cooling at hypersonic Mach numbers using forward facing array of micro-jets”. *International journal of heat and mass transfer*, **52** 3654–3664.
[7]. BallaVenukumar and K P J Reddy2007 “Experimental investigation of drag reduction by forward facing high speed gas jet for a large angle blunt cone at Mach 8”*Sadhana***32** 123-131.
[8]. Frank k. Lu, 2015 “Visualization of supersonic flow around a sharp-edged, sub-boundary-layer protuberance” *Journal Visualisation* **18**619–629
[9]. A.P. Roychowdhury and C. Unnikrishnan2016 “Numerical simulation of hypersonic flow over blunt bodies” *International journal of innovative research in technology & science***2**2321-1156.
[10]. David s. Dolling 2001 “Fifty years of shock-wave/boundary-layer interaction research: what next?” *AIAA journal* **39**.