Journal of Visualization, Vol. 8, No. 1 (2005)65-72

Visualization of Shock Waves around Hypersonic Spiked Blunt Cones Using Electric Discharge

Viren, M.*[1], Jagadeesh, G.*[2], Reddy, K. P. J.*[2], Sun, M.*[1] and Takayama, K.*[1]

*1 Interdisciplinary Shock Wave Research Center, Institute of Fluid Science, Tohoku University, Sendai, 980-8577,
Japan. E-mail: viren@rainbow.ifs.tohoku.ac.jp
*2 Department of Aerospace Engineering, Indian Institute of Science, Bangalore, 560-012, India.

Received 19 August 2003
Revised 24 May 2004

Abstract : The hypersonic flow fields around a 120^0 apex angle blunt cone fitted with spikes have been visualized using electric discharge technique at Mach 5.75 in a hypersonic shock tunnel. The blunt cone has been fitted with a flat tipped spike and a flat aerodisc tipped spike assembly. The length of the aerodisc tipped spike assembly has been kept equal to the model base diameter (L/D=1) and the length of the flat tipped spike has been kept at 12mm. The flow field pictures reveal all the salient features of the flow fields around the spiked bodies. Numerical simulations carried out to complement the experiments agree well with the experimental pictures.

Keywords : Visualization, Electric discharge, Spiked blunt cone, Hypersonic shock tunnel.

1. Introduction

Large angle blunt cones (semi-apex angle of $60^0/70^0$) are usually employed in interplanetary aero-assisted space missions. These configurations are preferred since they not only withstand the severe aerodynamic heating during re-entry with appropriate thermal protection but also provide sufficient body drag for slow and smooth descent in planetary atmospheres. But on the other hand the large wave drag experienced by the blunt body during ascent stage is undesirable and has to be minimized. Use of retractable aerospike (Hutt and Howe, 1989) ahead of the blunt body is one of the most useful and attractive techniques for minimizing the drag during the ascent stage in aero-assisted space maneuvers. The introduction of spike ahead of the blunt body alters the aerodynamics of the flow field and it is possible to completely push the blunt body shock away from the cone surface using appropriate spike shape and length. In order to investigate the feasibility of using a retractable nose spike concept for wave drag reduction on a 120^0 apex angle blunt cone at a hypersonic Mach number, an experimental programme on aerodynamic force measurements had been initiated. Exhaustive force measurements were carried out on the 120^0 apex angle blunt cone with various nose spike geometries, using an accelerometer force balance at Mach 5.75 in hypersonic shock tunnel HST2 at the Indian Institute of Science (Viren et al., 2002). Further, surface convective heat transfer measurements were carried out using platinum thin film sensors in order to find out the consequence of using the spikes on convective heating rates of the blunt body and also to study the extent of flow separation on the body (Viren, 2003). Illustrative numerical simulations of the flow fields around the spiked body have also been carried out using a commercial CFD code

CFX-TASCflow to have an insight into the experimental results.

As a prelude to the above experiments, a few flow visualization experiments were carried out on the 120^0 apex angle blunt cone, with and without nose spikes, using electric discharge technique (Jagadeesh et al., 1996), to understand the salient features of the flow fields around the spiked body. These studies were carried out at the initial stage in order to optimize both the size and geometry of the spike as well as to design the model configurations for drag and heat transfer measurements in the shock tunnel. Some of the visualized flow field results have been used to validate a commercial code CFX-TASCflow that has been used for numerical simulations in the present study. The visualized flow field pictures also ascertain the quality of the hypersonic flow in the shock tunnel test section.

The important research goals of the present study have been as follows:

1. Visualization of the shock structure around spiked blunt bodies using electric discharge technique.
2. Identification of the location of flow reattachment point on the blunt cone surface using visualization and surface heat transfer measurements for different spike geometries.
3. Computational study of the flow field around the blunt cone with spikes for complementing the visualization experiments.

In this paper, we present the details of the flow visualization experiment and some of the experimental and CFD results on the flow fields around large angle spiked blunt cones.

2. Experiments

The electric discharge technique is based on the principle that the local gas density dictates the amount of spontaneous light emission from a discharge zone (Nishio, 1990). Hence when the discharge takes place across a shock wave, the position of the shock wave can be clearly seen, as the intensity of light from the shock wave is different from that of the free stream and the shock layer. In the experiments described here, the electric discharge is generated between a point electrode fixed to the roof of the test section and a line electrode embedded in the model, which are exposed to hypersonic flow. The light emitted is captured using an SLR camera (Cannon F1.4) operated in the bulb exposure mode using ASA 1600 speed film.

The schematic diagram of the electrode arrangement on a 120^0 apex angle blunt cone with 2.6 mm diameter and 12mm long flat tipped spike, used for flow visualization studies is shown in Fig. 1. A 1mm dia. stainless steel needle suspended from the roof of the test section is used as a point electrode while a 0.5mm thick copper strip embedded in the blunt cone acts as a line electrode. In order to extend the discharge sheet to the spike region the copper strip and the nose spike are electrically connected. The electrodes are kept about 50mm apart and operating voltage and current across the electrodes are fixed at 1.8KV and 1A, respectively. A Bakelite hylem blunt cone model is used in the experiments to ensure insulation and also to avoid multiple reflections from the model surface. The application of high DC voltage across the electrodes is controlled by a switching transistor, which in turn is controlled by a triggering pulse from a pulse delay unit. In all the experiments, duration of the electric discharge between the electrodes is maintained for 2μs after the free stream in the tunnel test section attains steady state. For details on the electric discharge generation and control unit, refer Jagadeesh (2001). The nominal test conditions during the present set of experiments are shown in Table 1.

3. Results and discussion

Initially, the flow around the large angle blunt cone without spike is visualized. The blunt body shock is seen very clearly from the photograph shown in Fig. 2. The measured thickness of the shock layer along the stagnation streamline is 3.57±0.17 mm in air and 3.29±0.26 mm in carbon- dioxide for the blunt cone without spike (Jagadeesh et al., 2001). Then the flow fields around the blunt cone with a simple flat tipped spike of 12mm length and 2.6mm dia. have been visualized. The visualized flow fields around this spike are shown in Figs. 3 and 4, for different specific enthalpies. The spike flow separation shock, re-attachment point/shock, transmitted shock at the re-attachment point on the body can clearly be seen in these pictures. The measured flow separation shock angle is 43^0 (Fig. 3A). By comparing the pictures in Figs. 3A and 3B that are taken for identical flow conditions, a difference in the shape of the shock structures can be seen. This is an indication of a probable oscillation

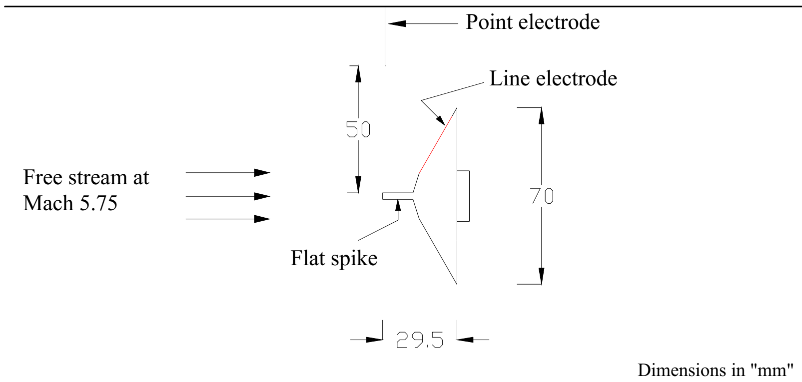


Fig. 1. Schematic of the Flow Visualization Model indicating Positions of the Electrodes.

Table 1. Nominal Flow Conditions for the Experiments.

Condition	Driver gas	Test gas	P_0 kPa	T_0 K	P_∞ kPa	T_∞ K	V_∞ m/s	ρ_∞ kg/m^3	M_∞	Re m^{-1}	Steady flow (test) duration µs
1	He	Air	500	1200	0.425	140	1400	0.01	5.75	1.5×10^6	500
2	N_2	Air	210	710	0.170	95	1100	0.0065	5.75	1.1×10^6	800

of the flow separation bubble on the spike surface. This unsteady bubble oscillation may cause shift in the point of flow re-attachment and may also lead to variations in the body (re-attachment) shock standoff distance. The green spot seen at the point of re-attachment in Fig. 4B indicates excessive heat in that region.

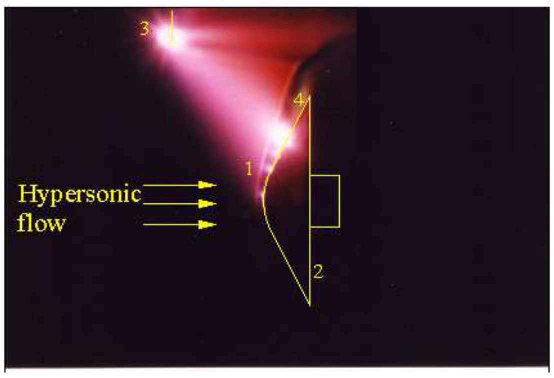


Fig. 2. Visualized Flow Field around the Large Angle Blunt Cone without Aerospike.
(Figure caption: 1: Bow shock, 2:Blunt cone, 3: Point electrode, 4: Line electrode.)

In order to complement the experiments, illustrative numerical simulations are carried out using commercial CFD code CFX-TASCflow. Axisymmetric steady N-S equations have been solved with appropriate boundary conditions based on the nominal flow conditions in the shock tunnel test section. The simulated pressure field around the blunt cone with 12mm long flat tipped aerospike is shown in Fig. 5. The visualized shock angle around the spike agrees well with the numerically simulated value of 45^0.

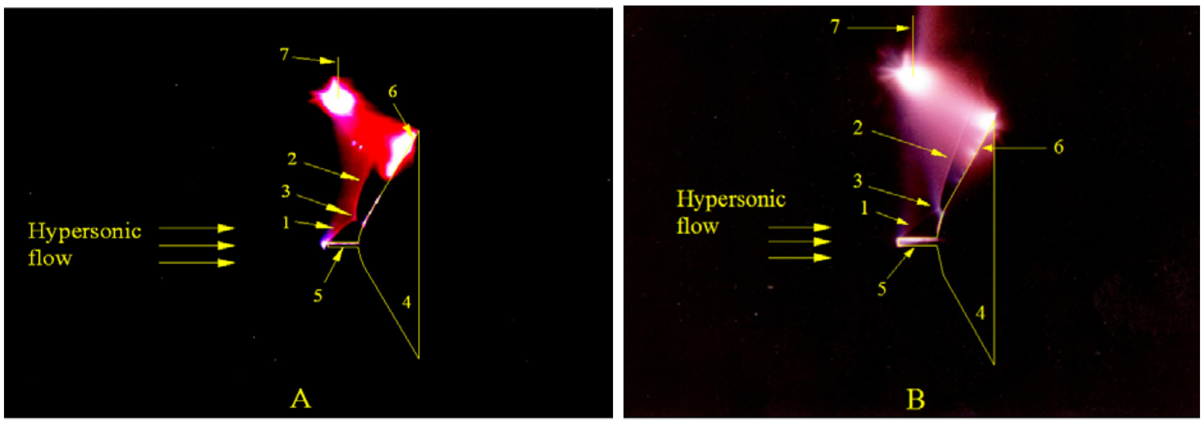


Fig. 3. Visualized Flow Fields around the Large Angle Blunt Cone with Flat Aerospike at Stagnation Enthalpy H_0 =1.2 MJ/kg; A: Run 1, B: Run 2. *(Figure caption: 1: Flow separation shock, 2: Re-attachment shock, 3: Re-attachment point, 4: Blunt cone model, 5: Flat spike, 6: Line electrode, 7: Point electrode.)*

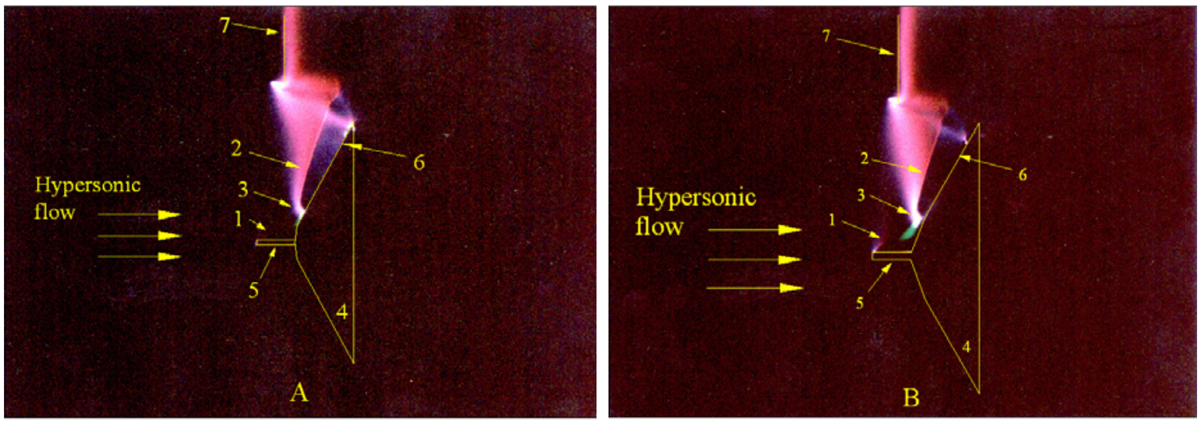


Fig. 4. Visualized Flow Fields around the Large Angle Blunt Cone with Flat Aerospike at Stagnation Enthalpy H_0 =0.7 MJ/kg; A: Run 1, B: Run 2. *(Figure caption: 1: Flow separation shock, 2: Re-attachment shock, 3: Re-attachment point, 4: Blunt cone model, 5: Flat spike, 6: Line electrode, 7: Point electrode.).*

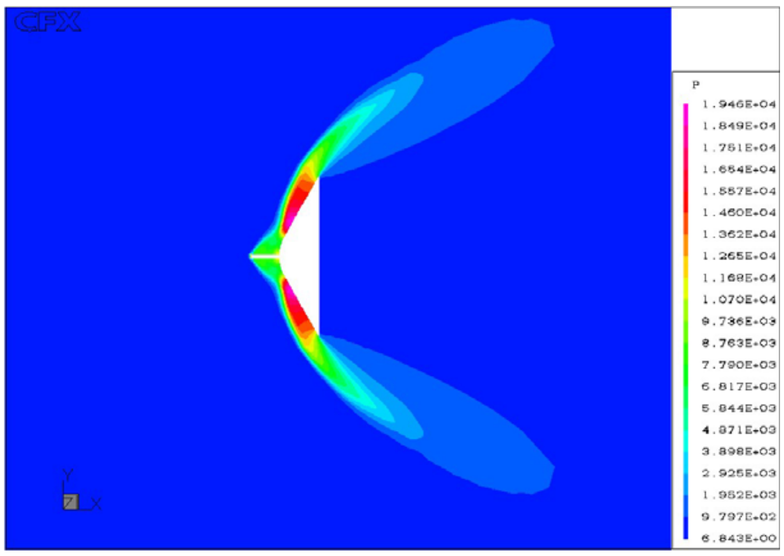


Fig. 5. Computed Pressure Field around the Blunt Cone with Flat Aerospike at Mach 5.75 & H_0 =1.2MJ/kg.

From the above flow field pictures, it can be seen that the 12mm long flat tipped spike causes the separated flow to re-attach at a point very close to the nose of the blunt cone. The flow re-attachment point is a region of very high temperature and pressure (refer Fig. 5), and even beyond this flow re-attachment point, the pressure on the body is high due to the close-standing body shock (re-attachment shock). Though this spike brings about a small flow separation near the nose of the blunt cone, the contribution of the spike to drag reduction is estimated to be negligible. Hence, to bring about maximum drag reduction on the body, the spike configuration should be such that the separated flow re-attachment point should be pushed off the body surface, engulfing the majority of the body in the low pressure separated flow.

In view of the above, a new spike configuration has been designed, with a flat aerodisc at its tip, and again the flow visualization experiments are carried out on the blunt cone with the new spike at a flow Mach number of 5.75 and a stagnation enthalpy of 1.2 MJ/kg. This time, the spike length to model base diameter ratio has been maintained at unity. The flat aerodisc diameter has been kept at $1/4^{th}$ the model base dia. and a spike of 2.6mm diameter has been used in the assembly. Figure 6 shows the flow field pictures around the flat aerodisc tipped spike for two identical flow (free stream) conditions. As seen from the visualized pictures, the disc oblique shock does not touch the blunt cone fore-body and the flow re-attachment point is pushed to the tip of the cone, creating flow separation all over the model fore-body. This brings about a substantial reduction in the blunt body wave drag and also a reduction in the surface convective heat transfer rates in the separated region. Figure 7 shows the experimental (measured) results on drag and heat transfer rates on the body, to which the pictures shown in Fig. 6 are complementary. There is an additional shock seen on the spike ahead of the blunt cone in Fig. 6, and this is expected to be the flow separation shock. Such a flow field around a spiked body generally exhibits some kind of unsteady behavior.

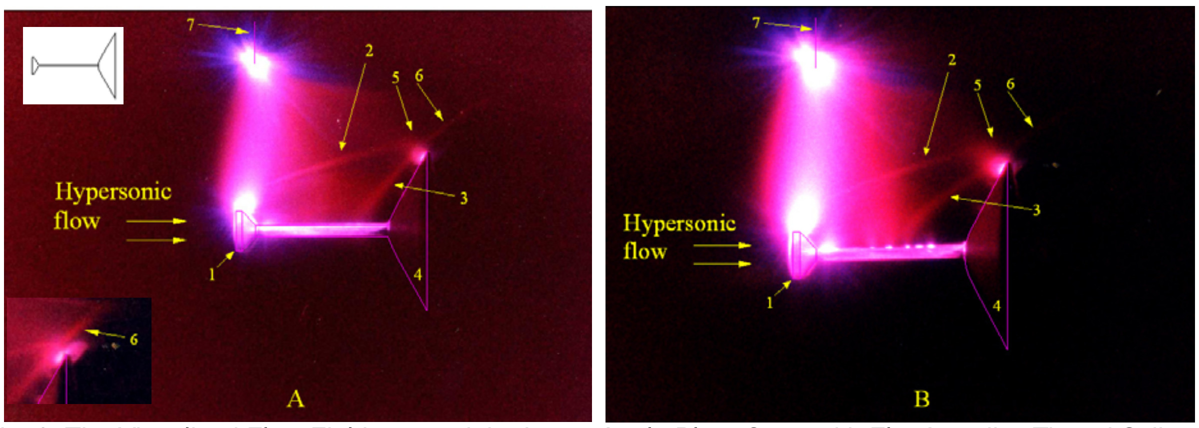
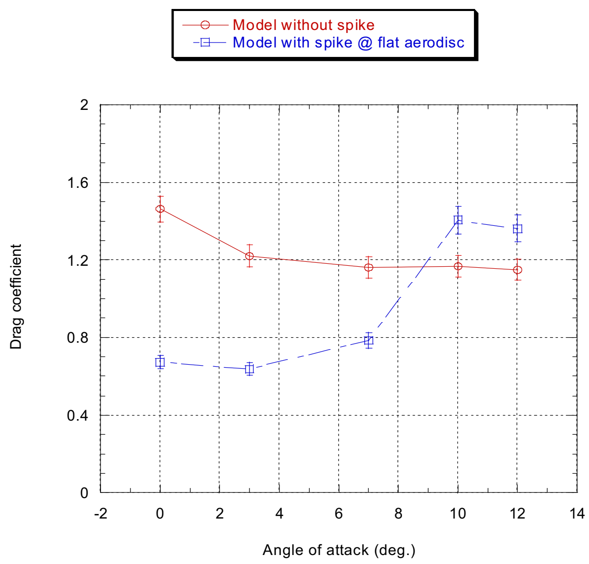


Fig. 6. The Visualized Flow Fields around the Large Angle Blunt Cone with Flat Aerodisc Tipped Spike at Stagnation Enthalpy H_0 =1.2 MJ/kg; A: Run 1, B: Run 2. *(Figure caption: 1: Aerodisc assembly, 2: Disc oblique shock, 3: Flow separation shock, 4: Blunt cone model, 5: Flow re-attachment point, 6: Re-attachment shock, 7: Point electrode.)*

Flow fields around the blunt cone with the flat aerodisc tipped spike are also simulated using CFX-TASCflow. The code has been run under steady state based on appropriate boundary conditions since the pressure based N-S solver could not be run as unsteady owing to very low density levels involved in the boundary conditions (based on experiments). One of the simulated flow fields is shown in Fig. 8. Though the experimental and simulated (steady state) flow field pictures (Fig. 3 &Fig. 5) agree well for the body with short 12mm long spike, a substantial difference is noticed in the experimental and simulated pictures for the body with aerodisc, with the flow separation shock missing in the simulated picture in Fig. 8, while the shape, angle of the disc oblique shock and the position of the flow reattachment point in Fig. 8 agree with the experimental results.

When we carried out heat transfer measurements on the surface of the blunt cone with disc spike, we observed unsteadiness in the temperature signals in the separation and re-attachment zones. Even a considerable scatter has been observed in the heat flux data in these zones for very identical free stream conditions in the shock tunnel. We wanted to probe into this unsteady nature of the flow and since the flow visualization carried out in the present case is not a time resolved one, an attempt has been made to steady the unsteady flow using a laminar axisymmetric unsteady Navier-Stokes solver, based on finite volume method on a solution-adaptive unstructured quadrilateral grid (Sun, 1999; Sun and Takayama, 1999). The MUSCL-Hancock scheme is used to determine the inviscid flux through interfaces. The boundary conditions used for the simulation are: Isothermal boundary condition at the wall & No slip condition at the wall. The gas has been assumed to be perfect and initial conditions for the simulations are based on experimental free stream values. Results of this time resolved simulation are presented in Fig. 9. The simulated results indicate that the flow field around the blunt cone with flat aerodisc tipped spike is a little unsteady, and this unsteadiness seems to be caused by the shedding vortices behind the aerodisc. Except for vortex shedding in the separated region behind the disc, the simulated results of both the codes show similar features, but the flow separation shock captured by the experimental pictures is not seen in the simulated flow fields.



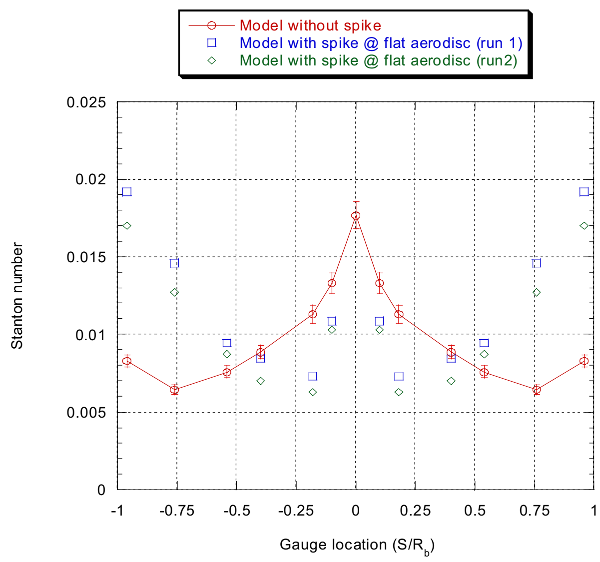


Fig. 7. Measured Drag and Heat Transfer Rate Values (non-dimensional) on the Blunt Cone with and without the Flat Aerodisc Tipped Spike at Mach 5.75 and Stagnation Enthalpy 1.2MJ/kg.

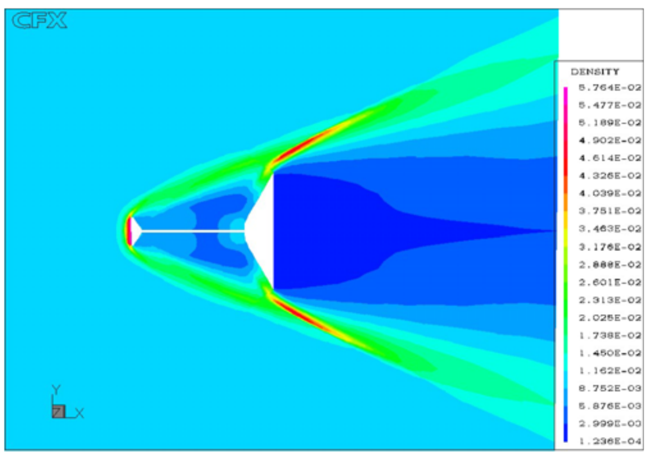


Fig. 8. Computed Density Field around the 120^0 Blunt Cone with Flat Aerodisc at Mach 5.75 and Stagnation Enthalpy H_0=1.2MJ/kg.

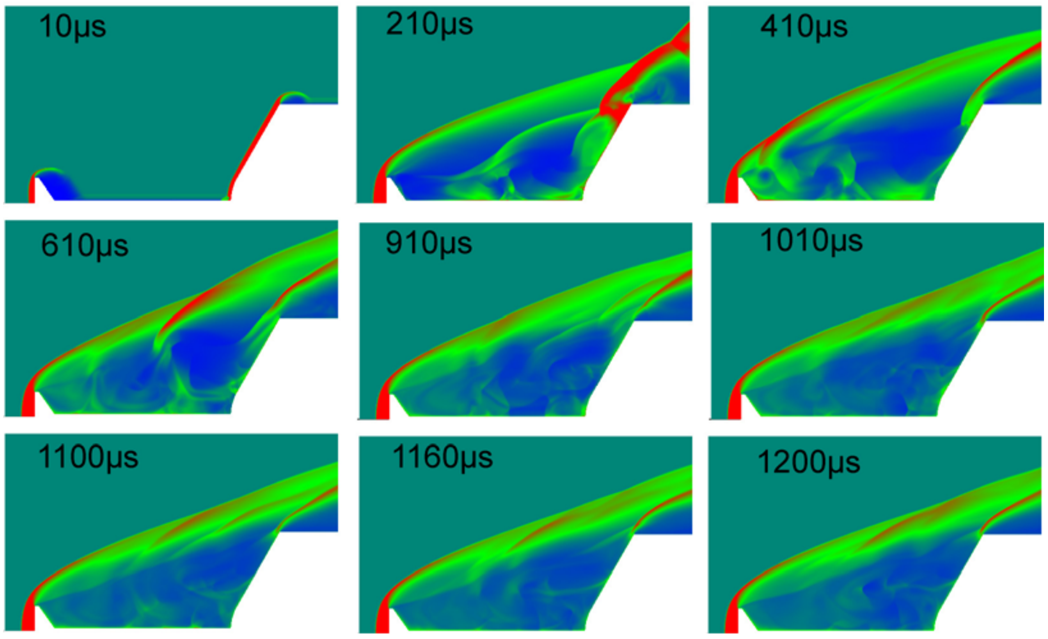


Fig. 9. Simulated Time Resolved Density Field Pictures around the Blunt Cone with Flat Aerodisc at Mach 5.75 & Stagnation Enthalpy H_0 =1.2MJ/kg.

4. Conclusion

The flow fields around a 120^0 apex angle blunt cone without any spike, with a short flat tipped spike and a flat aerodisc tipped spike are experimentally visualized in the test section of a shock tunnel at Mach 5.75 using electric discharge method. A clear bow shock wave is seen in the visualized flow field picture around the blunt cone model without any spike. Visualized flow field pictures around the 12mm long flat tipped spike clearly indicate all the salient features of the spiked body flow fields with a good match to the simulated results. Flow field pictures around the model with flat aerodisc tipped spike indicate that the entire model surface is engulfed in the aerodynamic shadow of the aerodisc, having flow separation almost all over the body surface with the flow re-attachment point pushed to the edge of the cone, which is the principal reason for the drag reduction associated with this spike configuration. The unsteady flow simulations carried out on the body with disc spike show some element of unsteadiness in the separated flow filed due to vortex shedding behind the aerodisc.

References

Hutt, G. R. and Howe, A. J., Forward Facing Spike Effects on Bodies of Different Cross Section in Supersonic Flow, The Aeronautical Journal, 93 (1989), 229-234.

Jagadeesh, G., Srinivasa Rao, B. R., Nagashetty, K., Reddy, N. M. and Reddy, K. P. J., A New Technique for Visualization of Shock Shapes in Hypersonic Shock Tunnel, Current Science, 71-2 (1996), 128-130.

Jagadeesh, G., Sun, M., Nagashetty, K. and Reddy, K. P. J., Experimental Investigation of the Test Gas Effects on Hypersonic Flow Field around Large Angle Blunt Cone in Hypersonic Shock Tunnel, AIAA Paper 2001-0305, (2001).

Nishio, M., A New Method for Visualizing Three-Dimensional Shock Shapes around Hypersonic Vehicles using an Electrical Discharge, AIAA Journal, 28-12 (1990), 2085-2091.

Sun, M., Numerical & Experimental Studies of Shock Wave Interaction with Bodies, Ph.D. Thesis, Tohoku University, Japan, (1999).

Sun, M. and Takayama, K., Conservative Smoothing on an Adaptive Quadrilateral Grid, Journal of. Computational Physics, 150 (1999), 143-180.

Viren, M., Saravanan, S., Jagadeesh, G. and Reddy, K, P, J., Aerodynamic Drag Reduction using Aerospikes for Large Angle Blunt Cone Flying at Hypersonic Mach Number, AIAA Paper 2002-2709, (2002),

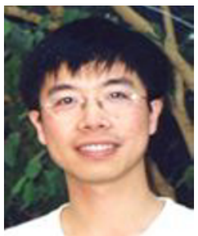
Viren, M., Investigation of Aerospike Induced Flow Field Modifications around a Large Angle Blunt Cone Flying at Hypersonic Mach Number, Ph.D. Thesis, Aerospace Engg. Dept., Indian Institute of Science, Bangalore, (2003).

Author Profile

Viren Menezes: He received his B.E. (Eng.) degree in Mechanical Engineering in 1994 and M.Tech. degree in Industrial Pollution Control Engineering in 1996 from Mangalore University, India. After obtaining M.Tech degree he worked as an assistant manager in an engineering firm till 1999. He received his Ph.D. degree in Aerospace Engineering in 2003 from Indian Institute of Science, Bangalore, India. Currently he is working as a JSPS post-doctoral fellow at the Institute of Fluid Science, Tohoku University, Sendai, Japan. His research interests include hypersonic aerodynamics, measurement and flow visualization techniques for shock/wind tunnels, high-speed unsteady flows and medical applications of shock waves.

Gopalan Jagadeesh: He received his B.E. (Eng.) degree in Mechanical Engineering in 1989 from Bangalore University, India, M.E. (Heat Power Eng.) degree in 1992 from Birla Institute of Technology, Mesra, Ranchi, India, and Ph.D. degree in Aerospace Engineering in 1998 from Indian Institute of Science, Bangalore, India. After receiving Ph.D. degree he worked as a visiting lecturer at the Shock Wave Research Center, Tohoku University, Sendai, Japan before taking up his current position as an assistant professor in the Department of Aerospace Engineering, Indian Institute of Science, Bangalore, India. His research interests include hypersonic aerothermodynamics and industrial & biological applications of shock waves.

K. P. Jagannatha *Reddy*: He received his B.Sc. degree in 1973 and M.Sc. degree in Physics in 1975 from Mysore University, India, and Ph.D. degree in Laser Technology in 1981 from Birla Institute of Technology, Ranchi, India. In 2002, he was promoted as a professor of Aerospace Engineering Department at Indian Institute of Science, Bangalore, India. His research interests include quantum electronics, optoelectronics, hypersonic aerodynamics, gasdynamic lasers and applications of lasers.



Mingyu Sun: He received his B.Sc. degree in 1993 and M.Eng. degree in 1995 from the Department of Mechanics, University of Science and Technology, China, and Ph.D. degree in Aeronautics and Space Engineering in 1998 from Tohoku University, Sendai, Japan. After obtaining Ph.D. degree he worked as a research associate at the Shock Wave Research Center at Institute of Fluid Science, Tohoku University. In 2004, he has been promoted as an associate professor of Institute of Fluid Science at Tohoku University. His primary research interest is basic wave phenomenon. He has developed a solution-adaptive code using unstructured quadrilaterals, which accurately and efficiently captures propagation and interaction of shock waves.

Kazuyoshi Takayama: He received his B.Sc. (Eng.) degree in Mechanical Engineering in 1963 from Nagoya Institute of Technology, Japan, his M.Sc (Eng.) degree in 1965 and Ph.D. degree in 1973 in Mechanical Engineering from Tohoku University, Sendai, Japan. In 1986, he was promoted as a professor of Tohoku University, and in 1988 he became the director of the Shock Wave Research Center at Institute of Fluid Science, Tohoku University. His primary interests are in the basic research of shock wave dynamics. He has intensively studied double exposure holographic interferometry for use in shock wave research. The results of his shock wave research have been successfully applied to various fields of science, technology and industry.