

Validation Studies of Shock Wave Boundary Layer Interactions in Hypersonic flow over Double Cone Geometries

Vayala Siva^a, R K. Peetala^{b,*}

^a*M.Tech Student, Department of Mechanical Engineering, VNIT Nagpur-440010, Maharashtra, India*

^b*Assistant Professor, Department of Mechanical Engineering, VNIT Nagpur-440010, Maharashtra, India*

*Corresponding author Email: rkpeetala@mec.vnit.ac.in

Abstract:

Validation studies of laminar shock wave boundary layer interactions in hypersonic flows of Mach number 11.30 over a 25° – 55° sharp edge double cone geometry conducted numerically. An open source CFD tool OpenFOAM based on the finite volume method used for the current study. Central-upwind schemes of Kurganov-Tadmor used for the flux calculations with second-order reconstruction schemes at cell faces. A proper grid independence study has done to obtain an accurate solution. The numerical solution captures all important features of the flow accurately and the obtained numerical results with optimum mesh are following the corresponding trend and matching with experimental results. Two different slope limiters -van Leer and superbee have been used to check the influence of slope limiters on the solution. It is found that superbee limiter is more accurate than vanLeer but inducing spurious spatial oscillations which is also consistent with previous works and these spatial oscillations diminish with mesh refinement.

Keywords: Hypersonic flows, Double Cone configurations, OpenFOAM, superbee limiter

1. Introduction

The future possibilities of hypersonic flow applications in civilian transport, defense sector as well as in space technology kept researchers carrying their work in the field of compressible flows. Hypersonic flows with the speed five times more than the sound speed has its own significance in the compressible flows from supersonic, transonic and subsonic flows. It involves numerous complex phenomena that made it as a special category of supersonic flows. These include shock-shock interactions, shock wave boundary layer interactions (SWBLI), high temperature effects such as vibrational excitation, dissociation and ionization. The presence of these phenomena in numerous applications of external as well internal flows gives a lot of importance to hypersonic flows. Out of many complex processes that hypersonic flows involved, shock – shock interactions and shock wave boundary layer interactions grabbing the attention of researchers. Presence these processes may induce massive thermal as well as mechanical loads on the surface of the space vehicles. The inevitable presence of boundary layer and shocks in hypersonic flows give chances to SWBLI which induces adverse pressure gradient inside the boundary layer. This results separated flows in the boundary layer accompanied by two or more additional shocks. Separation starts with separation shock and ends with reattachment shock. Flow passes through the reattachment shock will experience a large rise in temperature, which gives large thermal loads to the vehicle surface. A conscious study has to be carried out to understand and analyze these phenomena in the design of the thermal protection system of the space vehicles.

Many researchers have done experimental and computational studies on the SWBLI over many canonical geometries. Out of those, double cone configuration got importance due to its sensitivity to hypersonic flows and its ability to produce most of the possible complexities which are useful for design aspects of space vehicles. J. Olejniczak [1] has studied different shock interactions - type IV,

V, V and a new interaction on double wedge geometries computationally and stated the very fine mesh is required to capture these interactions accurately. Holden M. S. [2] has done preliminary studies of hypersonic flows over double cone configurations through experimental works which are still considered as a reference data for validation of computational results. G. V. Candler [3] conducted experimental studies on double cone geometries to provide the benchmark dataset and to mitigate the pre-existing differences between experimental and computational results. Datta V. Gaitonde [4] carried out numerical studies on the double cone hypersonic flows and concluded small changes in flow parameters and numerical aspects gives moderate changes in the flow properties. Later G. V. Candler [5] extended his studies through computational works in which he stated possible reasons for the discrepancies between experimental and computational results. In this study it is concluded that providing the incorrect free stream conditions are the main reason for these deviations of computational results from experimental results. Marie-Claude Druguet [6] analyzed a steady hypersonic flow over double cone configurations numerically and provided the insight of different numerical schemes and slope limiters that can be used for compressible flow simulations. It is concluded that less dissipation schemes and limiters provide the accurate solution and it is also shown that solution with very fine mesh will be independent of the changes in the numeric.

The present study focuses on the validation aspects of a shock wave boundary layer interactions over a sharp edge $25^\circ - 55^\circ$ double cone geometry at Mach number 11.30 using OpenFOAM.

2. Numerical Methodology

An open source C++ based CFD toolbox OpenFOAM is used for numerical studies based on the finite volume method (FVM). A density based solver named “*rhoCentralFoam*” based on central-upwind schemes of Kurganov and Tadmor for flux calculations at cell faces has been used for the current study. This solver solves fundamental governing equations in two steps– predictor and corrector. In prediction step, inviscid and convective terms of conservative variables ($\rho, \rho u, \rho e$) are treated explicitly from which primitive variables (ρ, u, T) are calculated. In corrector step earlier predicted primitive variables are corrected implicitly by adding viscous and diffusion terms. Courant number is chosen 0.5 and the time step is chosen according to the stability criteria.

The governing equations are given below.

$$\text{Conservation of mass} \quad \left(\frac{\partial \rho}{\partial t} \right) + \nabla \cdot (\rho u) = 0 \quad (1)$$

$$\text{Conservation of momentum} \quad \left(\frac{\partial (\rho u)}{\partial t} \right) + \nabla \cdot [u(\rho u)] + \nabla p + \nabla \cdot T = 0 \quad (2)$$

$$\text{Conservation of total energy} \quad \left(\frac{\partial (\rho E)}{\partial t} \right) + \nabla \cdot [u(\rho E)] + \nabla \cdot [up] + \nabla \cdot (T \cdot u) + \nabla \cdot j = 0 \quad (3)$$

Where, the total energy density $E = e + \frac{1}{2}U^2$, specific internal energy $e = c_v T$, diffusive heat flux $j = -k \nabla T$ (Fourier’s law of heat conduction) and T is the viscous stress tensor, assumed positive in compression. The detailed algorithm is provided by C. J. Greenshields [7].

3. Results and Discussion

3.1. Validation Studies

The present study focuses on the shock wave boundary layer interactions in hypersonic flows over double cone configuration. The geometrical details and sample mesh structure used for simulation are shown in Fig. 1. The axisymmetric boundary condition is applied to the axis of the cone to facilitate 2D simulation of the double cone. Nitrogen is used as test gas and the free stream conditions are the same as that of Marie-Claude Druguet [7]. Kurganov – Tadmor flux scheme with superbee interpolation has been used for the simulations. Separation bubble size is taken as a parameter for the validation due to its sensitivity to the changes in the flow. This separation region can be measured from the variations of surface properties like surface pressure, surface heat flux, skin friction coefficient, etc. due to its strong influence on the surface properties. In surface pressure variation sudden rise on first cone and peak value on second cone represents starting and endings of the separation zone. The numerical results of present studies are validated with well-established experimental results of Holden M.S [2]. The simulation results are shown in Fig. 2. are following the corresponding trend and well in agreement with experimental results.

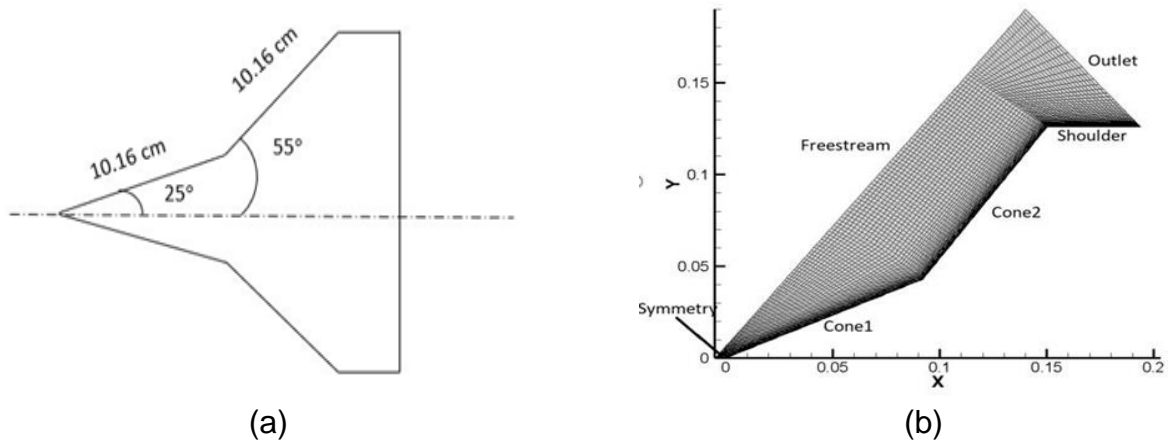


Fig. 1. Schematic of (a) Geometrical configuration (b) Sample structured mesh

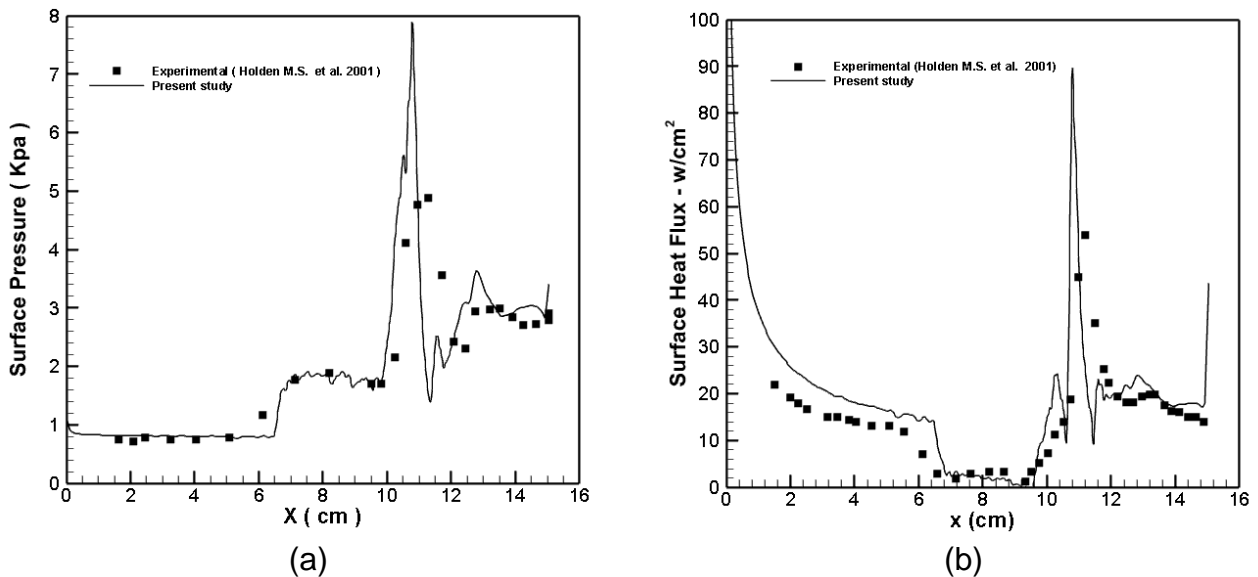


Fig. 2. Variation of surface properties along streamwise direction (a) Surface pressure (b) Surface heat flux

3.2. Effect of slope limiter

From the earlier studies, it is concluded that selection of limiter has a significant effect on accuracy of the solution. van Leer limiter is recommended for all kinds of compressible flow simulations due to its less dissipation characteristics. Despite being more accurate than vanLeer limiter, superbee is not used as much as vanLeer due to its nature of inducing spatial oscillations in the flow properties. The present study investigated the ability of both the limiters vanLeer and superbee in providing an accurate solution for the same grid. It is found that for sufficiently fine mesh 512×256 with $5 \times 10^{-7} m$ near wall spacing, superbee is significantly more accurate than vanLeer with negligible oscillations in surface properties which are shown in Fig. 3. It is also observed that superbee oscillations are grid dependent and diminishes with grid refinement. To investigate the dependency of superbee limiter on the grid in producing oscillations, three different meshes 128×64 , 256×128 , 512×256 with appropriate near wall spacings have been considered. Out of three meshes chosen 256×128 and 512×256 are showing the same separation size and similar trends of surface properties. But oscillations with 512×256 mesh are in negligible magnitude than 256×128 mesh. So 512×256 mesh is chosen as an optimum mesh from grid independent studies and used for further studies. Grid independent study results are shown in Fig. 4.

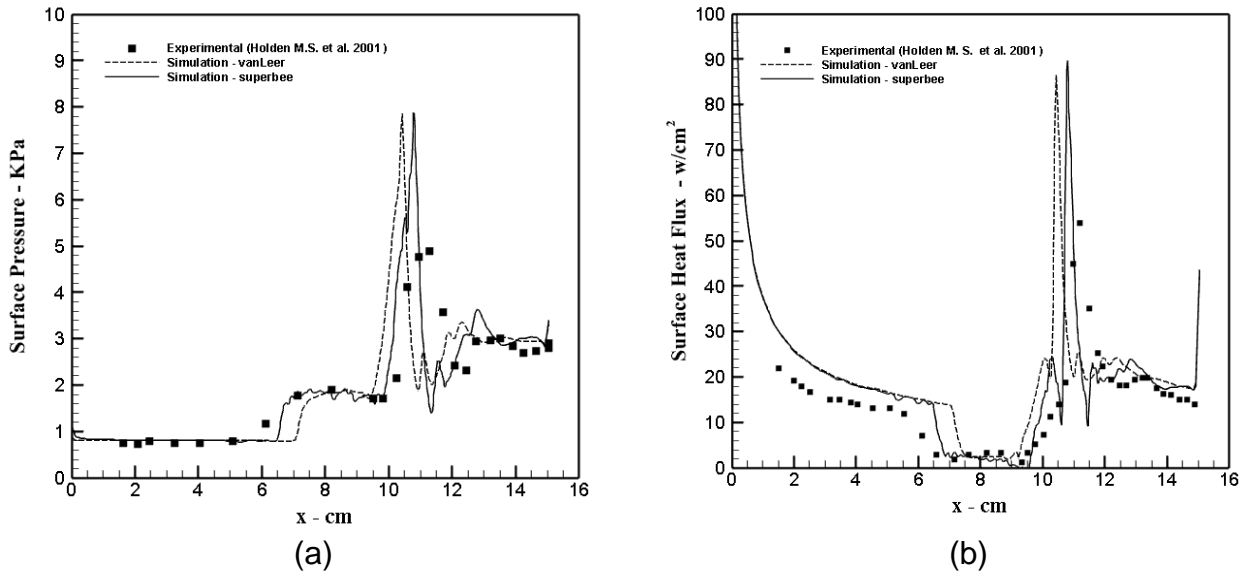


Fig. 3. Effect of slope limiter on surface properties (a) Surface pressure (b) Surface heat flux

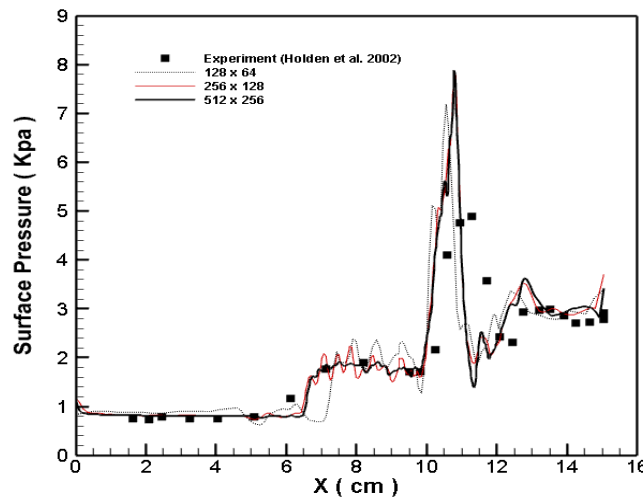


Fig. 4. Grid independence studies of superbee limiter

4. Conclusion

The numerical analysis of laminar shock wave boundary layer interactions over a double cone configuration has been carried out using OpenFOAM. The results obtained from the computational studies show the corresponding trend and are in good agreement with the experimental results. Effect of slope limiters was also studied to check the accuracy of superbee and vanLeer limiters in providing an accurate solution and found that for same grid structure superbee is considerably more accurate than the van Leer. It is also observed superbee limiter's ability to produce spurious oscillations is strongly dependent on the grid and diminishes with grid refinement.

References

- [1] J. Olejniczak, M. J. Wright, G. V. Candler, "Numerical Study of Shock Interactions on Double Wedge Geometries", AIAA Paper, Jan. 1996.
- [2] Holden, M. S., and Wadhams, T., "Code Validation Study of Laminar Shock/Boundary Layer and Shock/Shock Interactions in Hypersonic Flow. Part A: Experimental Measurements", AIAA Paper 2001-1031A, Jan. 2001.
- [3] G. V. Candler, I. Nompelis, M. S. Holden, "Computational Analysis of Hypersonic Laminar Viscous – Inviscid Interactions", 38th AIAA Aerospace Sciences Meeting and Exhibit, Jan. 10-13, 2000.
- [4] Datta V. Gaintode, Patrick W. Canupp, "Heat Transfer Predictions in a Laminar Hypersonic Viscous/ Inviscid Interaction", Journal of Thermophysics and Heat transfer, Vol. 16, No. 4, Oct-Dec 2002.
- [5] G. V. Candler, I. Nompelis, M.-C. Druguet, M. S. Holden, T. P. Wadhams, I. D. Boyd, Wen-Lan Wang, "CFD Validation for Hypersonic Flight: Hypersonic Double – Cone Flow Simulations", 40th AIAA Aerospace Sciences Meeting and Exhibit, Jan. 14-17, 2002.
- [6] Marie-Claude Druguet, G. V. Candler, I. Nompelis, "Effect of Numerics on Navier-Stokes Computations of Hypersonic Double-Cone Flows", AIAA Journal, Vol. 43, No. 3, March 2005.
- [7] C. J. Greenshields, H. G. Weller, L. Gasparini, J. M. Reese, "Implementation of Semi-discrete, Non- staggered Central Schemes in a Colocated, Polyhedral, Finite Volume Framework, for High-Speed Viscous Flows", International Journal for Numerical Methods in Fluids. Vol. 63, No. 1, pp. 1-21.