Research Article
Simulation and Experimental Validation of Hypersonic Shock Wave Interaction

Li Jing, Xiao Hong and Wu Di
School of Power and Energy, Northwestern Polytechnical University, Xi'an 710072, China

Abstract: The present paper examines the relevance of grid and simulation accuracy of hypersonic CFD in terms of hypersonic sharp double-cone flow. The flow grid and normal grid each adopted 250×100, 500×100, 1000×100, 500×200, 1000×200, 1000×400 and so on grids. When the normal grid was 100, the wall pressure and heat flux distribution obtained from flow grid 500 and 1000 were consistent, indicating that the solution of flow grid convergence was obtained. However, some difference was observed when the separation zone was compared with the experimental data. In increasing the normal grid number and adopting grid 500×200, the position of the separation point, wall pressure and heat flux peak was shown to be consistent with the experiment. When the grid was further encrypted, the calculation using grid 1000×200 and 1000×400 was equal to that using grid 500×200. The simulation of hypersonic sharp double-cone flow also showed that when the separation zone of the simulation was less than the experimental measurement, the wall pressure and heat flux peak moved forward. This is because the backwardness of the intersection of the separation shock and the first shock resulted in the forwardness of the intersection of the first shock and the second shock after interference, making the work region of the induction shock and boundary layer move forward. The key challenge in achieving the correct simulation of the hypersonic sharp double-cone flow is explained as follows: the algorithm can not only capture shock wave strength correctly and give the adverse pressure gradient formed by the interfering shock wave near the wall accurately. It can also prevent the numerical dissipation of the algorithm from affecting the simulation accuracy of the viscous boundary layer to ensure the correct prediction of the size of the separation zone.

Keywords: Grid relevance, jet interaction flow, limiter, sharp double-cone

INTRODUCTION

The simulation of flow hypersonic CFD plays an important role in the development of hypersonic vehicle. Shock/shock and shock/boundary layer interactions can seriously degrade the performance of a hypersonic vehicle. The complexity and the design implications of these phenomena require their quantitative assessment. The precise calculation of supersonic and hypersonic flows put conflicting demands on the formulation of inviscid numerical flux functions. A high speed numerical scheme has to possess enough dissipation to capture strong shocks without developing overshoots and oscillations in the vicinity of the discontinuity and the scheme must also possess numerical dissipation that is much smaller than the physical viscosity to accurately compute boundary layers (Husain, 2012; Brown, 2009). Investigation shows that simulation accuracy is directly depended on the grid algorithm (Holden, 2001). However, several conclusions in previous research are not entirely consistent (Benay, 2006; Gao, 2005), consequently making the development of an accurate simulation system of hypersonic CFD difficult. The present paper examines the effects of grids and the interpolation
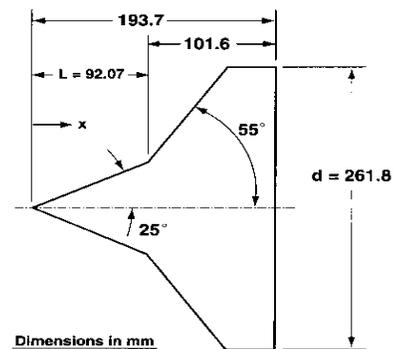

Fig. 1: Dimension diagram

limiter numerical method on simulation accuracy. Experimental data on hypersonic sharp double-cone flow are used in the aim to provide a basis for hypersonic simulation CFD software.

Introduction of sharp double-cone flow: The present work involves two typical non-linear interactions, namely, shock-shock and shock-boundary layer interaction. In examining the problem, assessing accurately the efficiency and thermal environment of a

Corresponding Author: Xiao Hong, School of Power and Energy, Northwestern Polytechnical University, Xi'an 710072, China

Table 1: Experimental condition of sharp double cone flow (28th)

Velocity of far filed	Density of far filed	Temperature of far filed	Temperature of wall
$V_\infty = 2664$m/s	$\rho_\infty = 0.0006546$ kg/m^3	$T_\infty = 185.6$K	$T_w = 293.3$K

Table 2: Simulation condition of sharp double cone flow (28th)

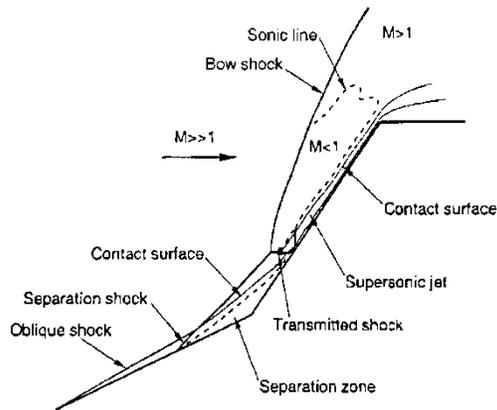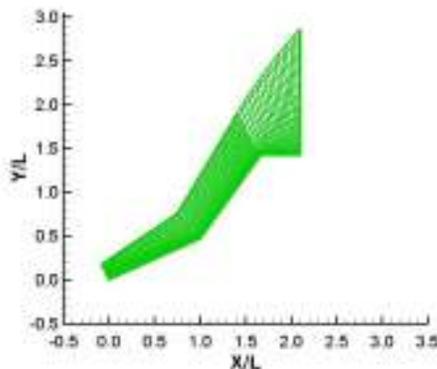| Mach number	Angle of attack	Flow temperature	Wall temperature	Reynolds number
$M_\infty = 9.59$	$\alpha = 0°$	$T_\infty = 185.6$K	$Tw = 293.3$K	$Re_\infty = 13253$/m

Fig. 2: Structure of flow field

Fig. 3: Grid diagram

space aircraft control plane at a large tilt angle is significant.

The Calspan University at Buffalo Research Center completed measurements of wall pressure and heat flux for various forms of compression corner to confirm the simulation accuracy of a numerical simulation software for laminar flow hypersonic separation flow (Moss, 2001; Gnoffo, 2001). The sharp double cone given in Fig. 1 is one of the models and the experimental condition of the 28th draught is as follows. The experimental cases selected as benchmark are shown in Table 1.

Figure 2 shows the diagram (Suzen, 1999) of the flow field structure. The first cone, which angle is relatively small, compresses flow and generates the attached shock wave. The Mach number decreases after the first shock wave, but the flow is still supersonic. The second shock wave is formed after flow compression by the second cone. Adverse pressure

gradient is formed in the second shock wave, inducing the boundary layer flow's separation at the first cone, finally forming the separation shock wave. These three shock waves interact and form two trigeminal points. The first trigeminal point is the intersection point of the shock wave formed in the boundary layer flow separation and the first shock wave, with the shock wave leaving the object surface after intersecting, inducing the first shear layer at intersection. The second trigeminal point is the intersection point of the first cone shock wave interfered by the separation shock wave and the second cone shock wave. It induces the second shear layer and shock wave, inducing shock wave as well through the wall which moves down in the form of multiple reflections at the wall and the second shear layer. Obviously, the simulation of flow field was quite difficult. Thus far, several software programs have made numerical simulations on this flow. The relevance of the simulation results (separation zone length, wall pressure and heart flow distribution) and experimental measurements was closely associated with the grid (Dilley, 2001; Mingsheng, 2007). To investigate the effect of the grid on CFD precision, some simulation cases of hypersonic double cone flow were conducted in current study. And, comparisons with experiment were also preformed to validate CFD precision.

Condition and method of calculation: The simulation condition is shown in Table 2.

Calculation method: KFVS flux splitting method and Vanleer interpolation limiter, discrete grid template as five-point using the original variable NND implicit scheme, laminar flow NS equation as the governing equation, finite volume as the discretion method and discrete equation using the LU decomposition technique.

Calculation grid: The flow direction and normal direction grid adopted the following six sets of grids: 250×100, 500×100, 1000×100, 500×200, 1000×200和 and 1000×400. Figure 3 shows the diagram of the grid.

RESULT OF THE SIMULATION

Figure 4 shows the sonic line, Fig. 5 gives the pressure contours and Fig. 6 presents a detailed view of the flow field. The flow field in Fig. 4 to 6 was in qualitative agreement with that in Fig. 2. As the inducted shock wave interacted with the boundary
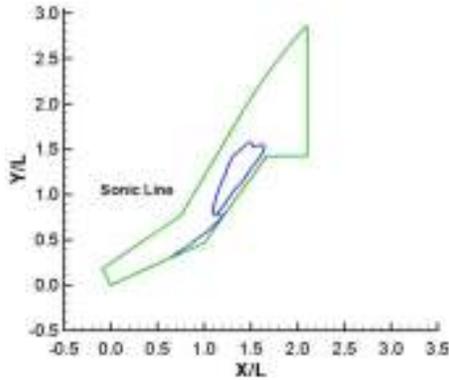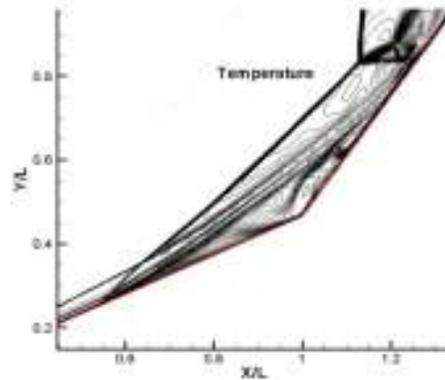
Fig. 4: Velocity line of flow field
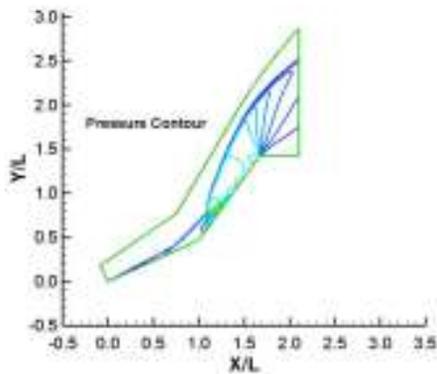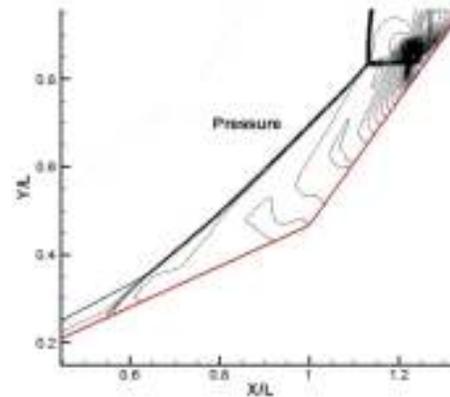

(a) Temperature contour

Fig. 5: Pressure contour

(b) Pressure contour

layer, a high-temperature and high-pressure zone was formed. A high temperature leads to a high wall heat flux.

Figure 7 and 8 show the wall pressure and heat flux distributions of the previous four sets of grids. When the normal grid was 100, the wall pressure and heat flux distributions obtained in the two sets of grids, namely, flow grid numbers 500 and 1000, were consistent. This result indicates that the convergence solution of the flow grid was reached. However, comparing the separation zone with the experimental data, some differences still exist. Therefore, increasing the normal grid number and adopting grid 500×200 resulted in a consistency of the position of the separation point, wall pressure peak and heat flux peak with the experiment. When the grid was encrypted further, the calculation using grid 1000×200 and 1000×400 was equal to the result of the calculation using grid 500×200. Figure 7 and 8 also show that when the separation zone of the simulation was less than the experimental measurement, the wall pressure and heat flux peak moved forward. This is because the backwardness of the intersection of the separation shock wave and the first shock resulted in the forwardness of the intersection of the first shock and the second shock after interference. This result made the work region of the induction shock and the boundary layer move
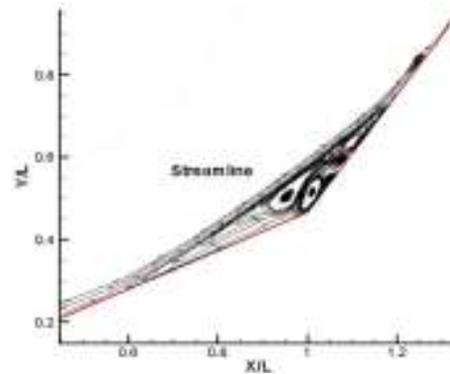
(c) Flow line

Fig. 6: Detail view of flow field structure

forward. Therefore, the key problem in correct simulation is as follows: the algorithm can not only capture the shock wave strength correctly and give the adverse pressure gradient formed by the interfering shock wave near the wall accurately. It may also prevent numerical dissipation of the algorithm from affecting the simulation accuracy of the viscous boundary layer to ensure the correct prediction of the size of the separation zone.
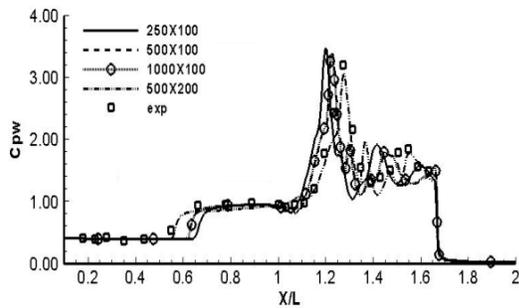
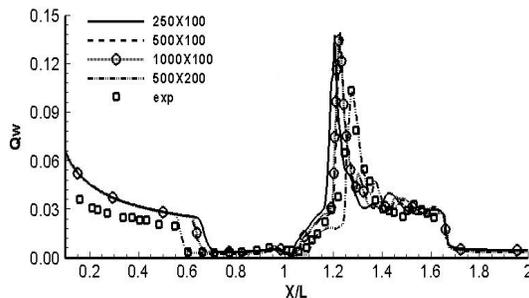Fig. 7: Affection of grid on wall pressure distribution

Fig. 8: Affection of grid on wall heat flow distribution

Therefore, after the grid achieved a certain condition, the hypersonic CFD software platform can stimulate effectively the complex shock/shock and shock/ wave-boundary interaction.

CONCLUSION

Through numerical simulation and analysis on the use of hypersonic CFD software for sharp double-cone shock wave interaction and jet interaction flow field, the following conclusions are obtained:

- On the simulation of hypersonic CFD software for shock wave interaction and jet interaction flow field, the flow topology was clear and the qualitative results were verified to be consistent with those in the literature
- By choosing the appropriate calculation method and examining grid convergence, the hypersonic CFD software platform can stimulate sharp double-cone corner flow accurately. The key problem in the simulation of the correct hypersonic sharp double-cone flow is that the algorithm can not only capture shock wave strength correctly and give the adverse pressure gradient formed by the interference shock wave near the wall accurately. It also shows the inability of the numerical

dissipation of the algorithm to affect the simulation accuracy of the viscous boundary layer in order to ensure the correct prediction of the size of the separation zone.

REFERENCES

Benay, R., B. Chanetz, B. Mangin, L. Vandomme and P. Jean, 2006. Shock wave/transitional boundary-layer interactions in hypersonic flow. AIAA J., 44(6): 1243-1254.

Brown, L.M., R.R. Boyce, N. Mudford and S.O. Byrne, 2009. Intrinsic three dimensionality of laminar hypersonic shock wave/boundary layer interactions. Proceeding of the 16th AIAA/DLR/DGLR International Space Planes and Hypersonic Systems and Technologies Conference, Bremen, Germany, 1: 17.

Dilley, A.D., 2001. Evaluation of CFD Turbulent Heating Prediction Techniques and Comparison with Hypersonic Experimental Data. NASA/CR-2001-210837.

Gao, X.C., 2005. Validations of hypersonic aerodynamic software. MS Thesis, Northwestern Polytechnical University, Xian, China.

Gnoffo, P.A., 2001. CFD Validation Studies for Hypersonic Flow Prediction. AIAA-2001-1025.

Holden, M.S., T.P. Wadhams, J.K. Harvey and G.V. Candler, 2001. Comparisons between Measurements in Regions of Laminar Shock Wave Boundary Layer Interaction in Hypersonic Flows with Navier-Stokes and DSMC Solutions. RTO-TR-AVT-007-V3.

Husain, M. and C.H. Lee., 2012 Numerical simulation of complex hypersonic flows. J. Space Technol., 1(1).

Mingsheng, M., 2007. Numerical investigation of supersonic jet interactions for tactical bodies. Acta Aerodyn. Sinica, 25(4).

Moss, J.N., 2001. Hypersonic Flows about a 25° Sharp Cone. NASA/TM-2001 -211253.

Suzen, Y.B. and K.A. Hoffmann, 1999. Application of Several Turbulence Models for High Speed Shear Layer Flows. AIAA-99-0933.