Proceedings of International Conference on Astronomy and Space Science
Celebrating 20th Anniversary of Korea Space Science Society, Jeju, 2004

SHOCK WAVE BOUNDARY LAYER INTERACTION STUDIES IN CORNER FLOWS

Hee-Joon Lee[1]†, Jan B. Vos[2]
[1]Control Systems Group, Korea Aerospace Research Institute, Daejeon 305-333, Korea
[2]Department of Mechanical Engineering, Ecole Polytechnique Federale de Lausanne, Lausanne CH1015, Switzerland
E-mail: joellee@kari.re.kr

(Received September 25, 2004; Accepted October 1, 2004)

ABSTRACT

Shock wave boundary layer interactions can make flows around a vehicle be very high pressure and temperature due to pass shock waves in small areas of the hypersonic vehicle. These phenomena can affect a critical problem in the design of hypersonic vehicles. To research the effect of shock wave boundary layer interactions, corner flows were studied in this paper using numerical studies with the NSMB (Navier-Stokes Multi Block) solver and then comparing corresponding numerical results with experimental data of the Huston High Speed Flow Field Workshop II. The mach number of flows is 12.3 in corner flows. The comparison with the computational result is presented based on diverse numerical schemes. Good agreement is obtained.

Keywords: aerodynamics, shock wave boundary layer interaction, hypersonic flows

1. INTRODUCTION

Shock wave boundary layer interactions are a critical problem in the design of vehicles flying at supersonics and hypersonic speeds, since it can induce locally high pressure, temperature and heat transfer rate gradients on the wall. It occurs when a shock wave impinges on a boundary layer, causing the boundary layer to separate due to the large pressure gradient across the shock wave. Concretely speaking, the hypersonic flow has a large kinetic energy that can be transferred to another kind of energy, almost internal energy near the wall where the velocity is zero. If the temperature is increased in the boundary layer, the density must be decreased on the assumption that the pressure is constant in the normal direction. The boundary layer should be increased because the decreased density makes the volume increase under the conservation of mass. We can define this phenomena as *viscous interaction*. There are two types of viscous interaction. One is pressure interaction caused by the exceptionally thick boundary layers on the surface under some hypersonic conditions, the other is shock wave boundary layer interaction by the impingement of a strong shock wave on a boundary layer.

This separation in general takes place upstream of the impingement point. The separated boundary layer induces a second shock wave will emanate from the reattachment point. At the reattachment point, the boundary layer is relatively thin and the heat transfer reaches a local maximum here.

†corresponding author

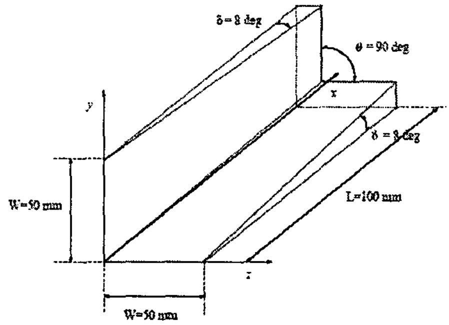


Figure 1. Corner test case geometry.

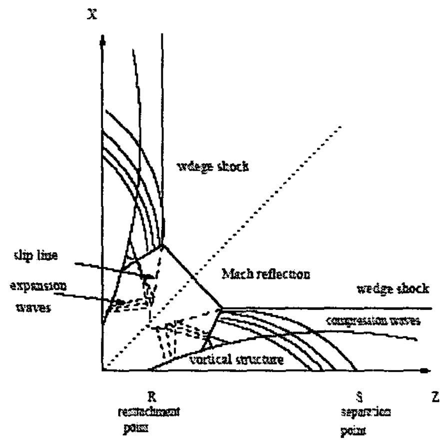


Figure 2. Flowfield configuration.

(a) Pressure contours (b) Mach contours (c) Temperature contours

Figure 3. Computational results at $x = 0.09$ m.

Shock wave boundary layer interactions were studied using corner flows (Marini 1997) named T4 case in the Huston High Speed Flow Field Workshop II in this paper. Moreover, the NSMB (Navier-Stokes Multi Block) solver (Vos et al. 1998) was adopted to solve corner flows with various central and godunov schemes. The NSMB solver was initially developed at EPFL (Ecole Polytechnique Federale de Lausanne). The solver was further developed in a joint research project between two universities (IMHEF/EPFL, Lausanne, Switzerland and KTH, Stockholm, Sweden), one research establishment (CERFACS, Toulouse, France) and two industrial partners (Aerospatiale, Toulouse, France and SAAB, Linkoping, Sweden).

Section 2 of this paper describes a calculating geometry and flow condition. In section 3 computational conditions are given. The results for the corner flows are discussed in sections 4 and 5 respectively.

2. CORNER FLOWS DESCRIPTION

The T4 case is composed of a 3 dimensional body that has 2 wedges at a corner angle of 90° with unswept leading edges orthogonal to the freestream.

The wedge angle δ is the same for both wedges, and is equal to 8 degrees. three dimensional

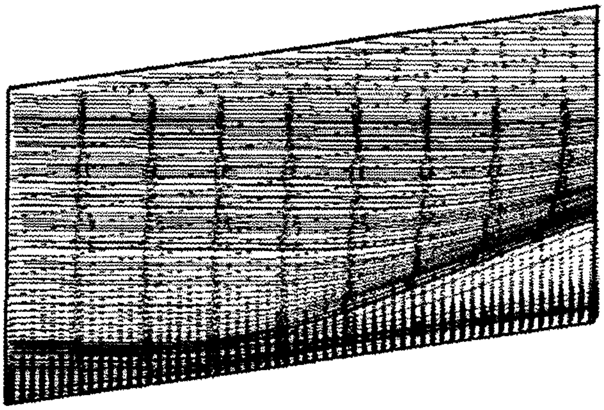
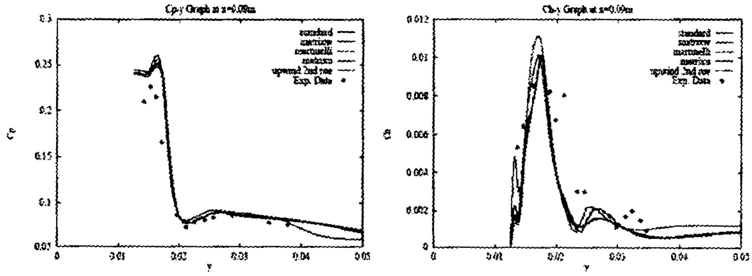


Figure 4. Streamlines at the wall.



(a) C_p (b) C_h

Figure 5. C_p and C_h near walls at $x = 0.09$ m.

as Figure 1, fully laminar flow as assumption and laminar viscosity by means of Sutherland's law. Flow conditions are mach 12.3 at Reynolds number 5×10^6. The region that has important problems is surely the corner region. As each wedge generates a shock wave, the intersection of these two shock waves can produce two different three dimensional shock configurations as shown Figure 2: the regular reflection configuration and the irregular reflection called mach disk. The difference between the two configurations is that for the Mach disk.

3. COMPUTATION

Figure 1 shows the geometry for this test case. The following boundary conditions were imposed:

• no-slip conditions at solid walss, with a surface temperature equal to ($T_{wall} = 300K$).

• at the inflow the conservative flow vector is specified and the values are imposed to the known upstream values.

• at the outflow boundary, the top boundary and the side boundary a zero gradient extrapolation for the conservative variables is applied.

Calculations were mare on both the coarse and fine grid, using the following space discretization schemes: central scheme with I) the standard (anisotropic) dissipation, ii) the artificial dissipation proposed by Martinelli, and iii) matrix dissipation. Calculations with the upwind scheme were impossible to start from free stream conditions. A result was obtained with the second order Roe scheme starting from an initial solution computed with the central scheme. For all computations, the implicit LU-SGS scheme was used to integrate the equations in time.

4. RESULTS

The computational results at $x = 0.09$ m are shown in Figure 3. It is possible to see the mach reflection and the impingement of the reflected shock waves on the viscous layer which generated a compression fan. Zones of very high pressure are detected near both walls and seem to correspond to the location of the reattachment of the main vortex. The reattachment region of the primary vortex at $x = 0.09$ m is located far away from the corner. In Figure 4, the oblique shock wave and the separated boundary layer are clearly visible. If the freestream mach number is increased, the boundary layer will grow and merge with the oblique shock wave that has a smaller angle. Consequently we can see shock wave boundary layer interaction based on the more and more separated boundary layer in hypersonic flows.

Experimental data are available in the plane $x = 0.09$ m. The pressure coefficient and heat flux coefficient at the wall are available in Figure 5. In Figure 5a, the experimental data and numerical data are almost consistent, but there exists some differences between 0.01 m and 0.02 m. In Figure 5b, the value of the peak heat flux for the secondary separation is not correctly predicted using these two methods. However, both the numerical and experimental results are almost similar. In Figure 3c, the static freestream temperature is 45.3 K, but increased until 520K. Thus we can confirm very high heat transfer gradient rate at shock wave boundary layer interactions.

5. CONCLUSIONS

Shock wave boundary layer interactions produced by a corner configuration have been successfully computed. The essential features of the interactions have been predicted. The resultant flowfield is similar to the flow field in experimental studies. Comparisons with experimental results are globally in good agreement. The main discrepancies with respect to the experimental results are the location of the separation and reattachment points. A parametric study would be necessary to clarify the influence of these conditions on the location of the separation and reattachment points.

REFERENCES

Marini, M. 1997, First europe-US high speed flow field data base workshop part II, Description of test cases, Technical Report CIRA-TR-97-048

Vos, J. B., Rizzi, A. W., Corjon, A., Chaput, E., & Soinne, E. 1998, Recent Advances in aerodynamics inside the NSMB (Navier-Stokes Multi Block) Consortium, AIAA 98-0225

Vos, J. B., Duquesne, N., & H. J. Lee 1998, Shock wave boundary layer interaction studies using the NSMB flow solver", Proceedings of the 3rd European Symposium on Aerothermodynamics for Space Vehicles, Noordwijk, pp.229-238, European Space Agency SP-426