Abstract

Recent advances in computing power have attracted much attention to the development of different new schemes especially in high speed compressible flow. Special attention is required for accurate simulation of flows with strong shock waves and capturing complex flow phenomena like shock wave reflection, diffraction, shock-shock and shock-vortex interactions. With a view to compute high speed compressible flow over different kind of geometries a finite volume method is developed for the solution of two dimensional Euler and Navier-Stokes equations on structured and unstructured grids. The driving algorithm is an upwind biased cell centered finite volume method. *AUSM+* method is used to split the convective fluxes. The split flux implementation with appropriate limiting has been discussed in details. The diffusion fluxes in viscous flow are computed using central difference scheme.

Several fundamental problems of high-speed compressible flows involving reflection and diffraction of shock waves, vorticity produced by shock wave diffraction and shock-shock interactions are studied to assess the numerical methodologies. The shock-wave diffraction process– result of two sub processes (shock-wave reflection and shock induced flow deflection) has been well captured by developed structured unsteady solver. Present numerical results obtained for shock diffraction over a $90°$ corner is well validated with the available results. On the shock-vortex interaction in Schardin problem the numerical results are validated with the experimental interferograms. The flow field is turbulent and contains sharp gradients due to the presence of shock waves. Without considering any turbulence model, using closely clustered grid points in conjunction with a fifth order *MUSCL* scheme these gradients and fine scale shear layer vortices are captured. Detailed simulation of steady inviscid shock interactions on double-wedge geometry is successfully performed on unstructured mesh. Two types of shock interactions, Type VI and Type V, are studied and discussed in details. The transition between the two types is identified. The resolution required introduces a length scale that determines the accuracy of capturing the shock interaction. In this respect, present unstructured Euler solver is robust enough to simulate the problem with a comparatively coarser mesh. The unstructured Euler solver has finally been modified to solve hypersonic flow over simple geometries taking the equilibrium air chemistry effects in to account.

Keywords: Compressible Navier-Stokes solver, finite volume method, *AUSM+*, *MUSCL*, limiter, supersonic, hypersonic, shock diffraction, shock-shock interaction, equilibrium air chemistry.