COMPUTATION OF HIGH SPEED CHEMICALLY REACTING VISCOUS FLOWS WITH CARTESIAN MESH ON A GPU BASED PARALLEL SYSTEM

A Thesis submitted
in partial fulfillment for the Degree of

Doctor of Philosophy

by

V.ASHOK

Department of Aerospace Engineering

INDIAN INSTITUTE OF SPACE SCIENCE AND TECHNOLOGY

THIRUVANANTHAPURAM

MARCH 2013

ABSTRACT

The computation of high speed chemically reacting flows during reentry of a vehicle from outer atmosphere and Scramjet propulsion involving high speed turbulent combustion of hydrogen and air are some of the important technologies for low-cost access to space. The solutions to such problems using Cartesian mesh framework have a tremendous advantage in terms of very fast turnaround time from geometry to solution because of completely automated grid generation. However the Cartesian mesh has a very serious limitation in terms of handling the near wall viscous resolution and hence requires some special treatment near the wall. With regard to solving complex flow problems pertaining to Scramjet combustion the turn- around time can be reduced in a very cost effective way through parallel computing with latest high performance computing technology engaging cluster of multi core processors with Graphic Processing Unit called GPU which accelerates the computation. The present work addresses all the above three issues namely, the near wall resolution of hypersonic viscous flows with a Cartesian mesh based system and computations of finite rate chemically reacting turbulent flow in Scramjet engines with Cartesian mesh and performing the Scramjet computations on a GPU based parallel system. Thus the work carried out has three main objectives. The first objective is to obtain the solution of high speed laminar viscous flows both non-reacting and reacting for reentry type problems with a hybrid approach of unstructured prism layer near the wall and Cartesian mesh way from the wall The second objective is to develop a turbulent finite rate chemically reacting code with hydrogen air combustion for Scramjet computations involving complex geometries with Cartesian mesh from an existing perfect gas Cartesian mesh turbulent flow code which uses a wall function approach. The third objective is to develop parallel computing algorithms and necessary code for GPU based parallel computing to perform tip-to-tail simulation for a typical Scramjet vehicle with combustion on a cluster of machines with GPU accelerators.

The Cartesian mesh based viscous laminar flow solution is achieved by creating an unstructured prism layer near the wall by the normal projection of Cartesian mesh panels and stitching with the outer Cartesian mesh and performing a hybrid solution having a combination of unstructured prism layer solution near the wall and Cartesian mesh solution away from the wall. As for the numerical scheme, the inviscid fluxes are computed using Advective Upstream Splitting Method and linear reconstruction of primitive variables with limiter is employed. The viscous fluxes are evaluated from gradients estimated using standard Green-Gauss procedure. The solution is fully explicit and marched using backward Euler time marching mode with local time stepping for convergence acceleration. The developed code is first validated for perfect gas conditions against available experimental results for typical sphere-cone-cylinder-flare geometry at hypersonic Mach number for zero angle of attack. For three dimensional cases with angle of attack, the prism layers extruded from the Cartesian mesh from surface panels which are of 3 sides to 6 sides are not stitched with the outer Cartesian

mesh. Hence in this approach, first an Euler solution is obtained for the Cartesian mesh and this solution is mapped on to the hybrid unstructured prism layer near the wall and the laminar Navier-Stokes solution carried out for the unstructured prism layer alone with the Euler solution data as the boundary condition for the outermost unstructured prism layer. This solution procedure was validated against available experimental heat flux data at angle of attack for hypersonic Mach number. For hypersonic chemically reacting flows the hybrid solution methodology with 7 species finite rate air chemistry model (Park-87) is used. Results of species mass fractions, temperature profiles, wall heat flux and shock stand-off distance from the present code are validated for standard test cases like chemically reacting flow over wedge and Lobb sphere by comparing with the reported results of other CFD code solutions with structured mesh and with limited experimental results.

The solution of turbulent flow in Scramjet engines with finite rate Hydrogen-air chemistry was achieved by developing a code starting from an existing Cartesian mesh perfect gas turbulent code with wall function. 7 species 7 reaction ONERA chemistry model was used to obtain the species production rates from Hydrogen-air reactions. The developed code was validated against available experimental data on pressure and total temperature from ground test results of a typical Scramjet combustor in connected pipe mode conditions. Since the ground test conditions are not the same as flight conditions, numerical experiments were performed to bring out the effects of inlet pressure and vitiation on the Scramjet combustor performance.

The finite-rate chemically reacting flow in Scramjet engines involve highly compute intensive operations on a very large mesh which typically demands use of high performance computing platforms. In this regard, the utility of latest GPU based computing platforms has been explored for such applications. To obtain good performance from GPU accelerators with Cartesian mesh solvers was a real challenge as the rectangular adaptive Cartesian mesh with hanging node is not inherently data parallel. To achieve good parallel computing performance with GPU accelerators, suitable data parallel algorithms as applicable to adaptive Cartesian mesh and good memory management techniques were developed. Data parallelism was achieved by grouping the Cartesian mesh cells into eight different cell groups with each group having almost identical computational flow and group-wise computation is launched in the GPU kernel one after another. Parallel computing performance on cluster of GPU machines and factors affecting the performance are brought out.

Tip-to-tail computation with combustion for a representative Scramjet vehicle with a cone cylinder fore body and two Scramjet engines mounted was carried out with the developed Cartesian mesh solver on a cluster of GPU machines. The performance of the vehicle in terms of pressure, combustion efficiency and thrust was evaluated for a typical flight condition for two air fuel equivalence ratios. The parallel computing performance on the GPU cluster for such a large size problem is also brought out.

In this thesis, Cartesian mesh based solution to laminar hypersonic flows both non-reacting and chemically reacting flow as in re-entry type vehicles and Scramjet engine turbulent flows with Hydrogen-air combustion which are the two critical technologies for low-cost access to space have been addressed Also development of suitable data parallel computing algorithms has been done to enable the use of adaptive

Cartesian mesh solver on a cluster of GPU based machines to reduce the turnaround time for Scramjet engine solution which is very essential for a faster design cycle.