A SOLUTION ADAPTIVE GRID METHOD FOR CALCULATION OF HIGH SPEED FLOWS AROUND BLUNT BODY CONFIGURATIONS

A.R Jahangirian
Aerospace Engineering Department
Amirkabir University of Technology
Hafez Avenue, Tehran, Iran
Email: ajahan@cic.aku.ac.ir

Abstract

Extension of a solution adaptive grid method to hypersonic flows is presented. Fully unstructured triangular grids are generated in an automatic manner and further refined by adapting to the flow features. Two-dimensional inviscid flow is simulated by numerical solution of Euler equations. A cell-centered finite-volume scheme is used and the steady state solution is obtained using an explicit multi-stage procedure, with standard convergence acceleration techniques such as local time stepping and residual smoothing.

The method is applied for two standard blunt body configurations at Mach numbers of 8.15 and 15 and the results show the efficiency and accuracy of the procedure in predicting strong flow discontinuities compared with unadapted grid results.

1 Introduction

Computational Fluid Dynamics (CFD) has attracted much interest in the last few years for simulation of high-speed flows [1-3]. This is mainly due to the difficulties in producing data from experimental facilities. Such flows are characterised by a strong detached shock wave around the blunt body, which is designed to reduce the heat transfer, thus preventing the nose to be burned at very high speeds. This phenomena presents either a discontinuity or strong gradients of the flow variables whose correct resolution has major impact on the quality and stability of the solution.

Several numerical schemes have been presented basically for transonic and low supersonic flows [4,5,6]. However, extension of these methods to high-speed flows has faced different problems [7,8]. Some reported difficulties include unexpected negative values of flow variables, spurious oscillations of the numerical solution near shock wave and slow rates of convergence [2].

Additional difficulty are associated with the generation of a suitable computational grid which must also contain sufficient, well-placed grid-points so as to resolve all relevant flow features (e.g. strong shock waves). The position of this flow feature, however, is not known a priori. Thus, grid adaptation is required to alter the grid based on the flow solution in an automatic manner.

Unstructured grid methods provide a natural environment for flow adaptation, so as to ensure the most efficient distribution and use of the grid points. Most of the previous works, however, utilised the structured grids. Therefore, the main objective of the current work is to present the application of a solution adaptive unstructured grid method to high-speed flows.

Results are presented for two blunt body configurations and the efficiency of grid adaptation in terms of accuracy and computational time is described.

2 Flow Solution Algorithm

Two-dimensional inviscid compressible flow is simulated by numerical solution of Euler equations in conservative form. A cell-centered finite-volume scheme is used and the steady state solution is obtained using an explicit, multi-stage procedure, with standard convergence acceleration techniques such as local time stepping and implicit residual smoothing. Further details of the procedure can be found in [9]. Some modifications are made to the original algorithm in order to be applied for hypersonic flows. These include the increase of the second and fourth order dissipation factors and decrease of the time step.

As mentioned by Kroll et al. [7] the central difference schemes, although may suffer from the shock wave resolution, are very reliable in terms of convergence and smoothness of the results. In contrast, some high-resolution upwind schemes showed oscillatory behaviour near shock waves.

3 Unstructured Grid Generation

Fully unstructured grids are generated in an automatic manner [10]. The method is based on a combination of grid enrichment techniques whereby new grid points and point connectivities are created simultaneously. A typical starting (coarse) grid is generated that discretise the flow domain by only a few cells. Therefore, an initial distribution of grid points within the flow domain is not required. Also, unlike the majority of existing methods, which start with a well-refined discretisation of the geometry, the present approach generates the boundary and field grids simultaneously.

Once the starting grid is constructed, the computational cells are subdivided using a directional refinement procedure. This method together with grid smoothing and diagonal swapping ensures the generation of isotropic cells that are most efficient for inviscid flow calculations. Introducing point and line sources within the flow domain, as appropriate performs preliminary clustering of cells around the geometry. More details about the grid generation procedure are mentioned in [11].

4 Grid Adaptation

The grid generation procedure outlined above results in a computational grid that is adequate for an initial flow solution. However, the grid may not yet be sufficiently well refined to give the required resolution of flow features such as shock wave and stagnation point. Further refining the initial grid, by adapting it to an initial flow solution can fulfill this requirement most efficiently. In the present approach, the grid is adapted to the flow solution by subdividing the cells based on the magnitude of flow gradients obtained from initial flow solution. Mach number has been found to be the appropriate choice for detecting inviscid flow features. Following Jahangirian and Johnston [11] the flow feature detector is computed for each cell-edge, by taking the first difference of the relevant flow property along that edge, made non-dimensional by the cell center value. Thus, the actual error indicator for

$$e_k = \frac{\sum_{i=1}^{3} \delta ui}{uref} \times \delta s^{\beta}$$

each cell ***k*** will be determined in the form;

Where δui is the gradient of detection parameter along the edge *I*, *uref* is the mean value of the detector variable, β is an empirically determined coefficient (typically between 0.0 and 1.0) and δs is a measure of the local length scale calculated by dividing the cell area by the sum of the cell-edge lengths.

Finally a cell is tagged for refinement if the error is larger than the average plus a fraction of the standard deviation of the error.

5 Results

Initial results contain hypersonic solution around a standard Double ellipse configuration at Mach number 8.15 and 30 degrees incidence angle [12]. Figure 1a illustrates the initial computational grid generated around the body. The grid contains 1364 cells, 762 points and 125 surface edges, while the grid points are initially clustered near leading edge (stagnation point) and close to the surface of the solid body, where flow gradients are supposed to occur. The flow-field iso-Mach contours obtained from this grid are plotted in figure 1b. As sketched, the strong bow shock is relatively truncated on this initial grid mainly due to insufficient grid points in these areas.

The grid is then adapted to the shock wave by refining the grid in high gradient regions. Figure 2a shows the final grid, after four application of grid refinement, which contains 3402 cells, 1801 points and 148 surface edges. As illustrated, more grid points are introduced in the vicinity of the shock wave, while the rest of the domain remains almost unchanged. The final solution on this grid is shown in figure 2b presenting remarkable improvement in the representation of the shock wave compared with the initial unadapted solution.

The second case is a hypersonic flow with the free-stream Mach number of 15 over a blunt body configuration (figure 3a). This flowfield was also used by Drikakis and Tsangaris [2] using a generalised upwind method on structured grid and is chosen here in order to demonstrate the efficiency and accuracy of the adapted results.

Figure 3a shows the adapted grid after 4 refinement cycles which contains 2229 cells. The iso-pressure lines are shown in figure 3b. A regular (51x21) unstructured grid is generated which contains 2000 cells (figure 4a). As expected the regular grid presents smoother iso-pressure contours near the shock wave (figure 4b). This is mainly due to the alignment of grid lines with the shock wave as also reported by Kroll et al [7]. Despite that, the resolution and position of the predicted shock by adapted grid is quite reasonable.

The efficiency of the adapted grid is further demonstrated in figure 5. This figure compares the convergence history of adapted grid to the regular unstructured grid. As illustrated the adapted grid needs 25% less computational work to achieve a certain level of convergence compared with the regular unstructured grid.

In figure 6 the temperature distribution along the symmetry line is plotted. The adapted and regular unstructured grid results are compared with the corresponding results of Drikakis [2] who used a structured (81x41) grid. The jump in the value of the temperature across the shock wave for all results are comparable, while the adapted grid and Reference [2] results present sharper shock resolution than the regular grid. The calculated stand-off distance of the shock wave is in good agreement with the corresponding value $d=0.197R$ obtained by Billing [13].

In figure 7 the pressure and density distributions along the body surface are shown. Again the adapted grid and results of Ref. [2] are almost identical, where regular unstructured grid results show some deficiencies.

6 Conclusions

Extension of a solution adaptive grid method to hypersonic flows was presented. Fully unstructured triangular grids were generated in an automatic manner and further refined by adapting to the flow features. Two-dimensional inviscid flow was simulated by numerical solution of Euler equations. A cell-centered finite-volume scheme was used and the steady state solution was obtained using an explicit multi-stage procedure.

The method was applied for two standard blunt body configurations. The results were compared with the results of Ref [2] and those achieved using a regular unstructured grid. It was shown that the adapted grid could produce the accurate results with less computational time than the other two methods.

References

[1] Kordulla W. Computational techniques for hypersonic flows. AGARD 761, 1988.

[2] Drikakis D and Tsangaris S. On the accuracy and efficiency of CFD methods in real gas hypersonics. Int. J. for Numerical Methods in Fluids, Vol. 16, pp 759-775, 1993.

[3] Yee H.C, Klopfer G.H and Montagne J.L. High resolution shock-capturing schemes for inviscid and viscous hypersonic flows. NASA TR-100097, Apr. 1988.

[4] Jameson A, Schmidt W and Turkel E. Numerical solutions of the Euler equations by a Finite Volume method using Runge-Kutta time stepping scheme. AIAA Paper 81-1259, 1981.

[5] Roe P.L, A survey of upwind differencing techniques. Proc 11th Int Conference on Numerical Methods in Fluid Dynamics, 1988.

[6] Steger J.L and Warming R.F. Flux vector splitting of the inviscid gasdynamic equations with application to Finite Difference methods. Journal of Computational Physics, Vol 40, pp 263-293, 1981.

[7] Kroll N, Gaitonde D and Aftosmis M. A systematic comparative study of several high resolution schemes for complex problems in high speed flows. AIAA Paper 91-0636, 1991.

[8] Muller B. Simple improvements of an upwind scheme for hypersonic flows. AIAA Paper 89-1977, 1989.

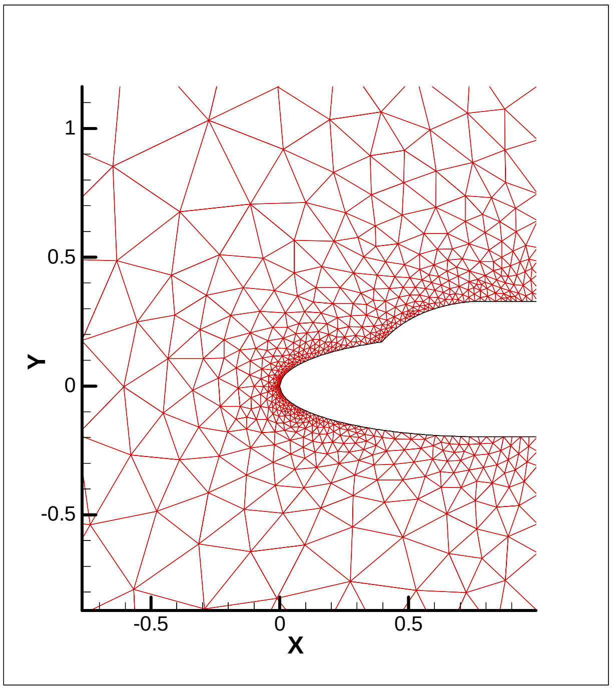
[9] Stolcis L and Johnston L. Solution of the Euler equations on unstructured grids for two-dimensional compressible flows, Aeronautical Journal, Vol 94, No 936, pp 181-195, 1990.

[10] Jahangirin A and Johnston L. Unstructured grid generation and flow adaptation for external aerodynamic flows. Numerical Methods for Fluid Dynamics V, Oxford, pp 439-446, 1995.

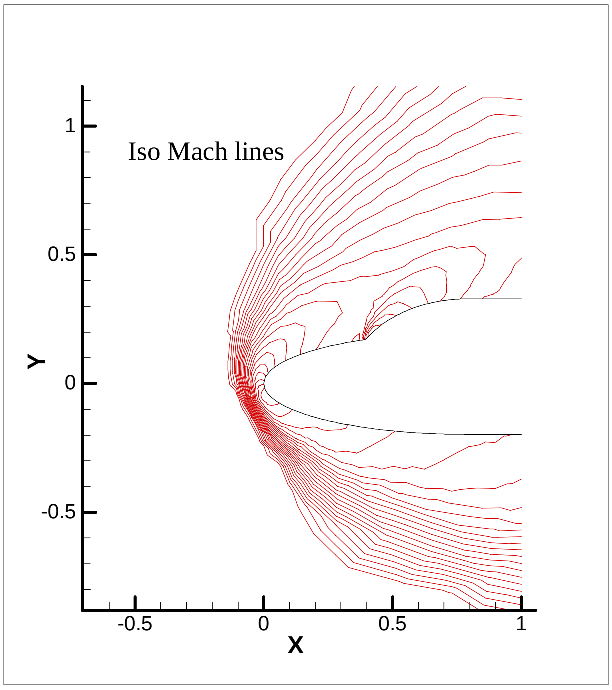
[11] Jahangirin A and Johnston L. Calculation of high-lift aerodynamics on adaptive unstructured grids. ICAS Congress, ICAS-96-1.10.5, Italy, 1996.

[12] Desideri J.A Glowinski R and Periaux J. Hypersonic flows for reentry problems, 1990.

[13] Billing F.S. Shock wave shapes around spherical and cylindrical nosed bodies. Journal of Spacecraft and Rockets, No 4, pp 822-823, 1967.

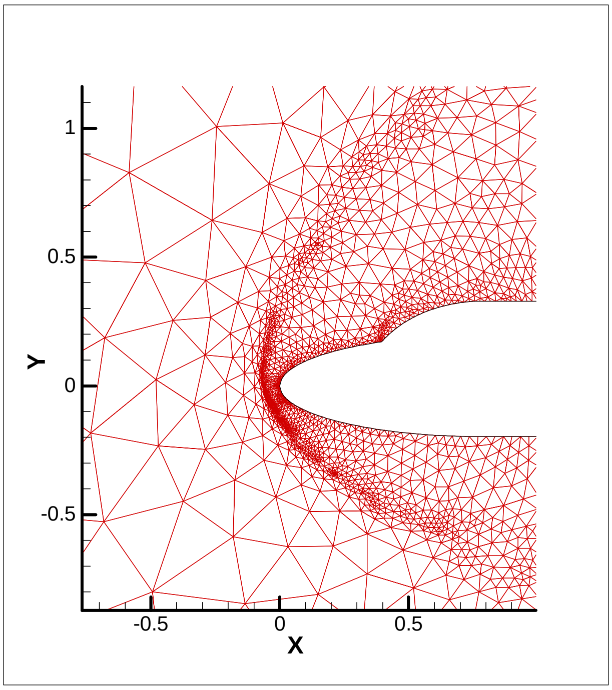


a)

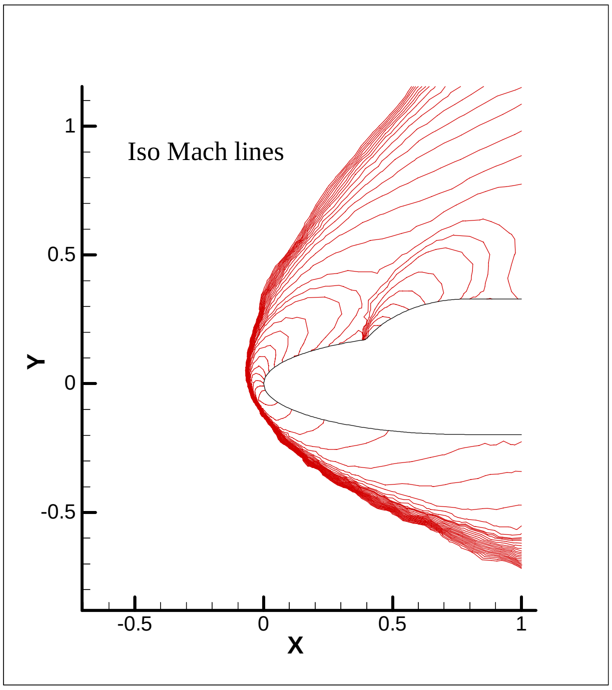


b)

**Figure 1. a) Computational initial grid (1364 cells) ,
b) Iso-Mach lines , Mach 8.15, Incidence 30 deg.**

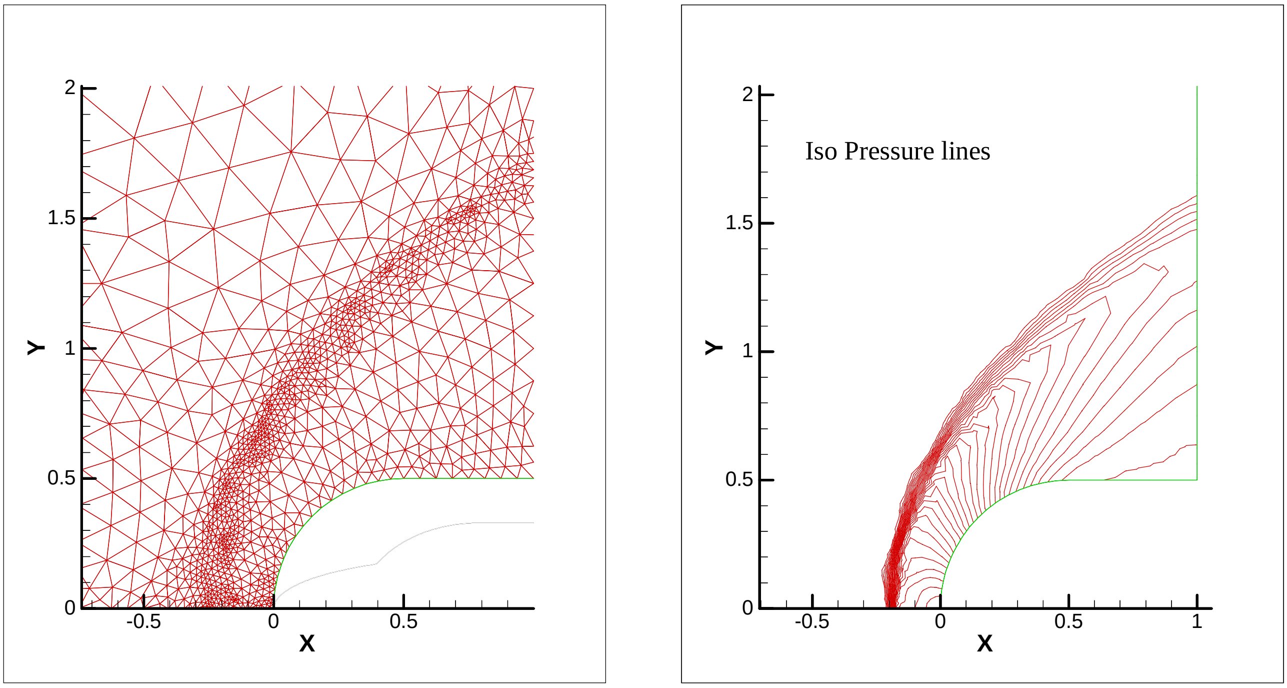


a)



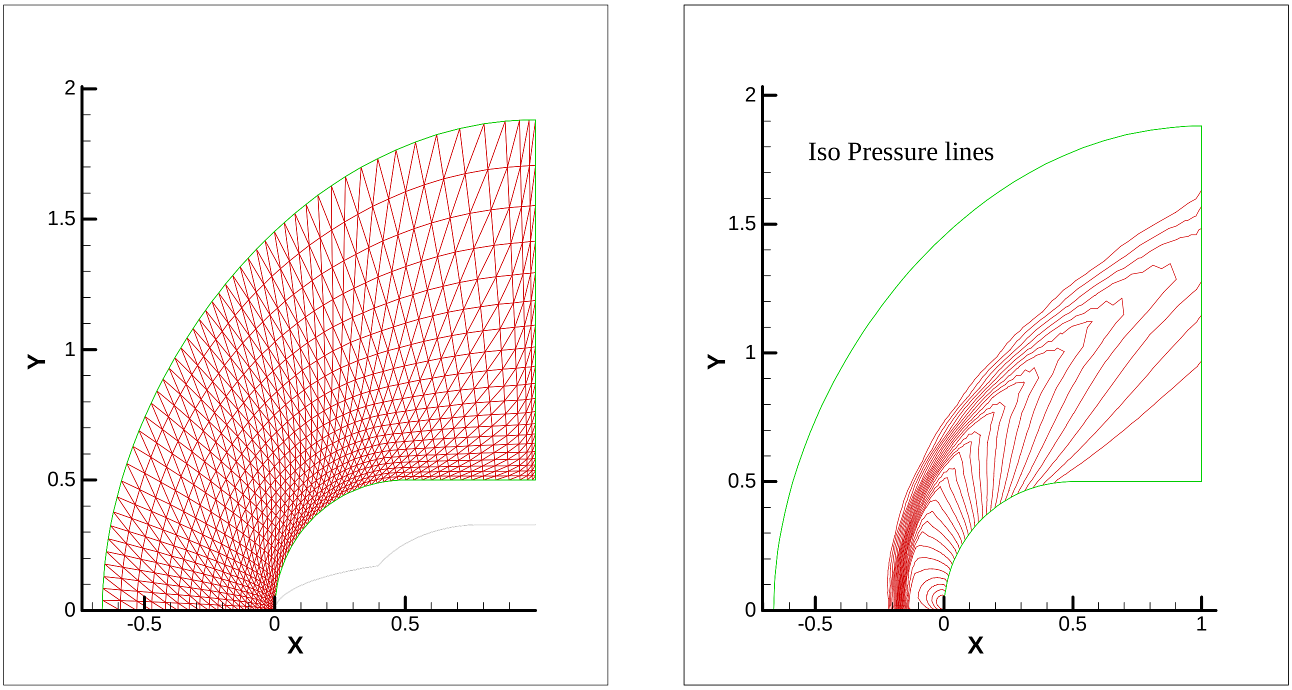
b)

**Figure 2. a) Adapted grid (3402 cells) ,
b) Iso-Mach lines, Mach 8.15, incidence 30 deg.**



a) b)

Figure 3. a) Computational adapted grid (2229 cells) , b) Iso Pressure lines , M=15



a) b)

Figure 4. a) Regular unstructured grid (2000 cells), b) Iso Pressure lines , M=15

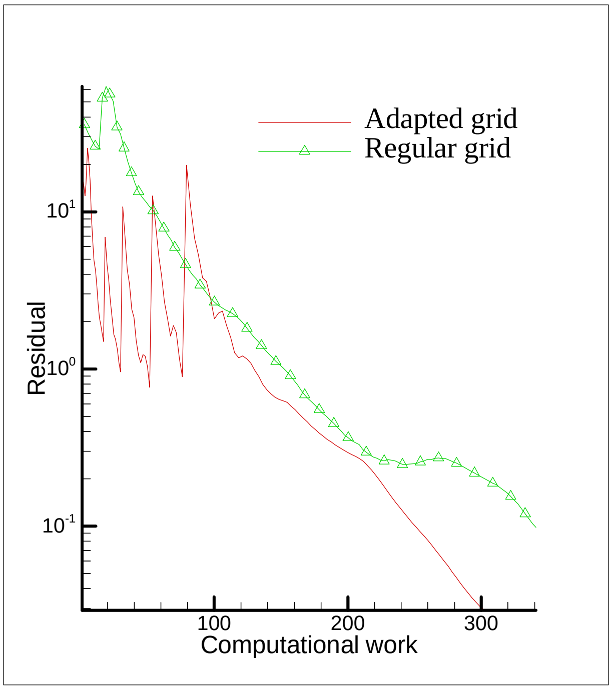


Figure 5- Convergence history, Mach 15

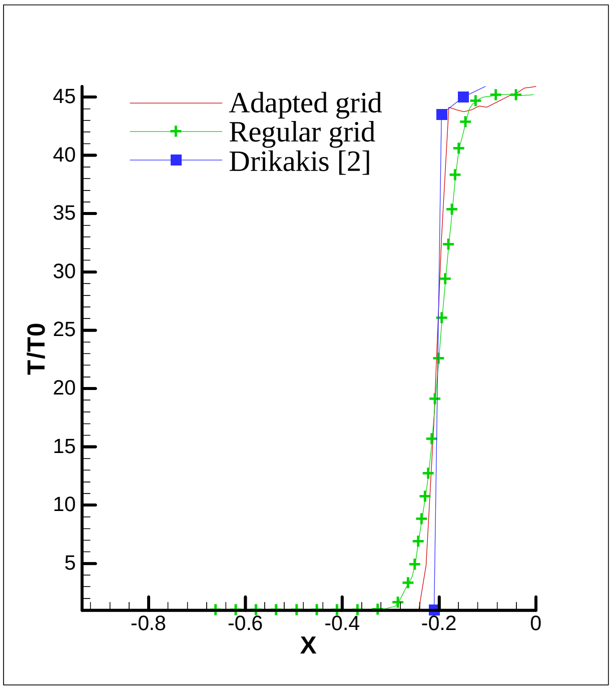
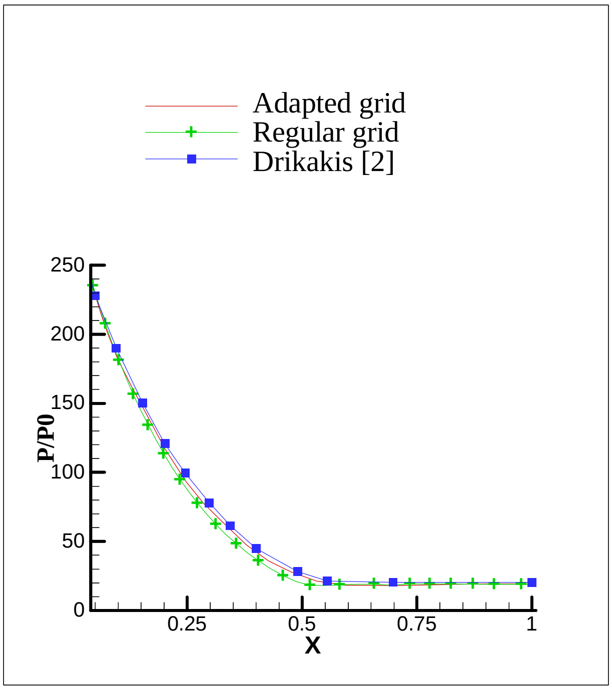
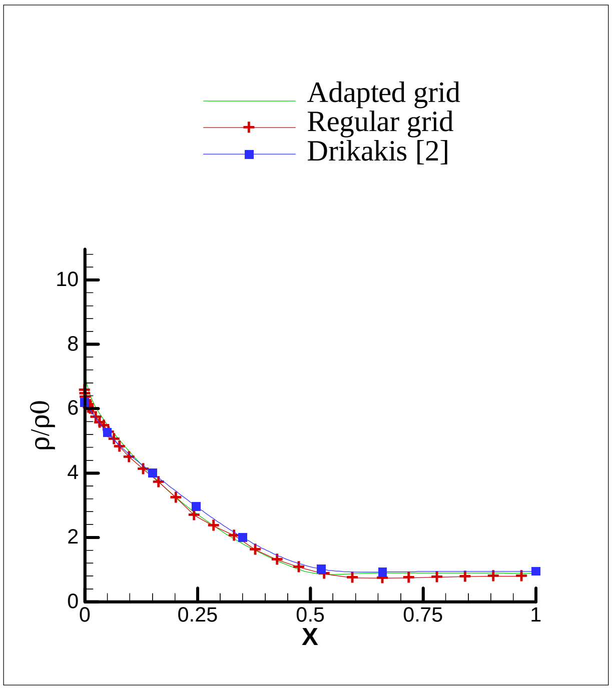


Figure 6- Temperature distribution along symmetry line, Mach 15



a)



b)

Figure 7- Distributions along body surface, a) Pressure, b) Density, Mach 15