AIAA 94-2329

A 2-D Compressible Navier-Stokes Algorithm Using an Adaptive Unstructured Grid

Hin-Fan Lau and Doyle D. Knight
Rutgers University
Piscataway, New Jersey

25th AIAA Fluid Dynamics Conference
June 20-23, 1994 / Colorado Springs, CO
A 2-D COMPRESSIBLE NAVIER-STOKES ALGORITHM USING AN ADAPTIVE UNSTRUCTURED GRID

Hin-Fan Lau*
and
Doyle D. Knight†
Department of Mechanical and Aerospace Engineering
Rutgers University - The State University of New Jersey
PO Box 909 - Piscataway NJ 08855-0909

ABSTRACT

The results of a two-dimensional, compressible, Navier-Stokes solver using an unstructured grid with adaptive remeshing are presented in this paper. Roe's flux-difference splitting method and Gauss's Theorem were used correspondingly to represent the inviscid and viscous terms of the Navier-Stokes equation. Temporal integration is performed with a modified Runge-Kutta method. Mesh adaptation is accomplished through five simple error indicators. They are the undivided difference of density, velocity and energy, and the Laplacian of velocity and temperature. After the adaptive process, a global remeshing of the domain was performed using an efficient mesh generator. Two problems were analyzed by the adaptive algorithm. The first was a supersonic flat plate boundary layer at a free stream Mach number $M_\infty = 2.0$ and Reynolds number (based on the plate length) $Re_{plate} = 6.5 \times 10^4$. The results were compared with the theoretical Blasius solution. The second case was viscous flow past a NACA 0012 airfoil at zero angle of attack. The free stream Mach number $M_\infty = 0.2$ and the Reynolds number (based on the airfoil chord) $Re_c = 10^4$. The numerical solutions were compared to computational results obtained by the Beam-Warming scheme. Good agreement was observed for both the boundary layer and airfoil computations.

*Graduate Student, Member AIAA
†Professor, Associate Fellow AIAA
Copyright © 1994 by Hin-Fan Lau and Doyle D. Knight. Published by the American Institute of Aeronautics and Astronautics, Inc. with permission.

INTRODUCTION

During the past few years, much interest has been focused on the applications of unstructured grids in the area of computational fluid dynamics. With the development of the various upwinding algorithms such as Godunov [6], Roe [23] and van Leer [26], the use of polygons as mesh elements has been very successful. A basic advantage of the unstructured mesh is its ability to discretize complex domains without the need for mesh transformation. Consequently, solutions for practical engineering problems can be more easily obtained with the use of unstructured mesh.

Another benefit provided by the unstructured grid is its inherent capability for local mesh modifications. Since the locations of structures such as shock waves, slip lines, expansion fans, and stagnation regions are not known a priori in most engineering problems, it is desirable to modify the mesh during a computation so that flow features will be appropriately resolved. Unstructured grids are very promising in the ability to capture flow structures since the mesh can be changed locally with the addition or subtraction of nodes. The robustness of the unstructured mesh's ability to resolve flow structures has been demonstrated by various researchers, for example Löhner [15], Yang et al [29], Trépanier et al [25], and Baum and Löhner [2].

For the process of grid adaptation, there are two major issues, namely, the selection of criteria to identify cells for adaptation and the approach for mesh modification. These two areas are of concern because, depending on their application
and usage, non-unique solutions can be obtained as suggested by Warren et al [27] and Hassan et al [7]. The common categories for error indicators (e.g., undivided difference, gradients and second derivatives) are discussed by Warren et al. The three approaches for grid adaptation are node movement, node redistribution and node enrichment. These are examined by Hassan et al.

For the present research, five simple error indicators are selected to define the areas where refinement and coarsening should be performed. They are the undivided difference of density [20], velocity magnitude [11] and energy, and the Laplacian of velocity magnitude and temperature. These criteria are selected to identify different physical aspects of flow. Additionally, since the quantities required for the evaluation of the criteria are readily available from the flow solver, the values of the error indicators can be obtained with minimum computation.

Once the cells have been identified for modification, the method of mesh enrichment is used to refine the regions where flow features exist. After the modification of the mesh is completed, the subsequent grid is obtained by performing a global remeshing of the computational domain with an unstructured mesh generator. With the increasing interest with unstructured grids, many researchers, for example Holmes and Snyder [8], Nelson [19], Merriam [18], Baker [1], Lee and Drysdale [14] and Spargle et al [24], have developed efficient triangular mesh generators to produce the best possible cells for a given node distribution. For the current work, the mesh generator developed by Merriam [18] which uses an Advancing Front technique is employed for the remeshing process.

GOVERNING EQUATIONS

The two dimensional compressible laminar Navier-Stokes equations for an arbitrary control volume \( V \) and surface \( \partial V \) can be written in non-dimensional form as

\[
\frac{d}{dt} \int_V Q \, dx \, dy + \int_{\partial V} (F \, dy - G \, dz) = 0
\]  

with \( Q \) being the vector of dependent variables

\[
Q = \begin{pmatrix}
\rho \\
\rho u \\
\rho v \\
\rho e
\end{pmatrix}
\]  

where \( \rho \) is the density, \( u \) and \( v \) are the velocity components in the \( x \)- and \( y \)-directions, and \( e \) is the total energy per unit mass,

\[
e = \frac{T}{\gamma(\gamma - 1)} + \frac{1}{2}(u^2 + v^2)
\]

\( T \) is the static temperature and \( \gamma \) is the ratio of specific heats. The flow variables are non-dimensionalized using the characteristic length \( L \), the free stream density \( \rho_\infty \), the speed of sound \( a_\infty \), and the free stream static temperature \( T_\infty \). The flux vectors in equation (1) are

\[
F = \begin{bmatrix}
\rho u \\
\rho u^2 + p - \tau_{xx} \\
\rho uv - \tau_{xy} \\
\rho eu + pu + \beta_x
\end{bmatrix}
\]

\[
G = \begin{bmatrix}
\rho v \\
\rho vu - \tau_{xy} \\
\rho v^2 + p - \tau_{yy} \\
\rho ev + pv + \beta_y
\end{bmatrix}
\]

For the implementation of Roe's method, it is convenient to decompose these vectors into their inviscid and viscous contributions,

\[
F_{inv} = \begin{bmatrix}
\rho u \\
\rho u^2 + p \\
\rho uv \\
\rho eu + pu
\end{bmatrix}
\]

\[
F_{vis} = \begin{bmatrix}
0 \\
-\tau_{xx} \\
-\tau_{xy} \\
\beta_x
\end{bmatrix}
\]

\[
G_{inv} = \begin{bmatrix}
\rho v \\
\rho vu \\
\rho v^2 + p \\
\rho ev + pv
\end{bmatrix}
\]

\[
G_{vis} = \begin{bmatrix}
0 \\
-\tau_{xy} \\
-\tau_{yy} \\
\beta_y
\end{bmatrix}
\]

Equation (1) can be rewritten with the decomposition of the flux vector as

\[
\frac{d}{dt} \int_V Q \, dx \, dy + \int_{\partial V} (F_{inv} \, dy - G_{inv} \, dz) + \int_{\partial V} (F_{vis} \, dy - G_{vis} \, dz) = 0
\]
The components of the viscous stress tensor are
\[\tau_{xx} = \frac{M_\infty}{Re_\infty} \mu \left( \frac{4 \partial u}{3 \partial x} - \frac{2 \partial v}{3 \partial y} \right)\]
\[\tau_{xy} = \frac{M_\infty}{Re_\infty} \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)\]
\[\tau_{yy} = \frac{M_\infty}{Re_\infty} \mu \left( \frac{4 \partial v}{3 \partial y} - \frac{2 \partial u}{3 \partial x} \right)\]

where \(M_\infty\) is the reference Mach number, \(Re_\infty\) is the reference Reynolds number, and \(\mu\) is the molecular viscosity normalized by a suitable reference value \(\mu_\infty\). The static pressure \(p\) is normalized by \(\rho_\infty a_\infty^2\) and satisfies the Ideal Gas Equation
\[p = \frac{\rho T}{\gamma}\]

Furthermore,
\[\beta_x = q_x - \tau_{xx}u - \tau_{xy}v\]
\[\beta_y = q_y - \tau_{xy}u - \tau_{yy}v\]

where the heat fluxes are
\[q_x = -\mu \frac{M_\infty}{Re_\infty} \frac{1}{Pr(\gamma - 1)} \frac{\partial T}{\partial x}\]
\[q_y = -\mu \frac{M_\infty}{Re_\infty} \frac{1}{Pr(\gamma - 1)} \frac{\partial T}{\partial y}\]

with \(Pr\) being the Prandtl number.

**FLOW SOLVER**

In this section, only a brief summary of the flow solver is discussed. A detailed discussion of the numerical algorithm is provided in Knight [12].

The cell-averaged value of the vector of dependent variables \(Q\) is defined by
\[Q_i = \frac{1}{V_i} \int_V Q dx dy\]

where \(i\) is the cell index and \(V_i\) is the volume of the cell. The governing equations (7) are therefore
\[\frac{d}{dt} (Q, V_i) + C_i = 0\]

where \(C_i\) is the net flux across the cell faces
\[C_i = \int_{\partial V_i} (F_{in,x} dy - G_{in,x} dx) + \int_{\partial V_i} (F_{vis} dy - G_{vis} dx)\]

The inviscid fluxes in \(C_i\) are obtained using Roe’s [23] flux-difference splitting method
\[\int_{\partial V_i} (F_{in,x} dy - G_{in,x} dx) = \sum_{k=1}^{k=3} T^{-1} H s\]

where \(k\) denotes the three faces of a cell, \(T^{-1}\Delta s\) is the rotation matrix, and \(H\) is the rotated inviscid flux vector for the cell faces. The value of \(H\) is computed with the expression
\[H = \frac{1}{2} \left( H_l + H_r + \frac{1}{2} [\delta s] \chi^{-1} \Delta R \right)\]

where \(H_l\) and \(H_r\) denote the “left” and “right” reconstructed values of the flux vector corresponding to the inside surface of the triangle and outside surface of face, respectively and the third term of the expression represents the contribution to the flux vector made by waves originating within the adjacent cells. This third contribution to the flux vector \(H\) is calculated from a second order accurate linear reconstruction. For the viscous fluxes and heat transfer components of \(C_i\), Gauss’s Theorem [16] is applied to a quadrilateral formed by the centroid of the two cells sharing the face and the face endpoints (see Fig. 1). Using the quadrilateral, an arbitrary function, \(f(x, y)\), can be represented by
\[\frac{\partial f}{\partial x} = \frac{1}{V} \int_{\partial V} f n_x dA\]
\[\frac{\partial f}{\partial y} = \frac{1}{V} \int_{\partial V} f n_y dA\]
ADAPTATION

Mesh adaptation is performed in four major steps. They are the identification of cells for adaptation, the modification of mesh, reconnection between the new set of nodes, and interpolation of flow variables. There are many techniques to achieve the results for these stages. For the present work, five simple criteria are employed to mark the necessary cells. A mesh enrichment approach refines the cell while nodes are allowed to be removed from regions where features do not exist. A global remeshing is performed afterwards to obtain optimum triangles from the new distribution of nodes. Finally, a linear interpolation scheme recomputes the cell centroid values after the remeshing is completed.

Criteria

During the recent studies for adaptive grids, many different criteria have been used as error indicators to aid in the selection of elements to change. Some of the criteria are based on intuitive mathematical arguments [4], others are based on flow properties such as the change of pressure or density [27], and the compressibility and vorticity [20]. However, whichever criterion is chosen, the equidistribution of the error in the computational domain is the objective. In this research, five physical criteria were used. The first three are the undivided difference of density, velocity magnitude and energy. The values for these criteria are computed for cell $i$ by

$$E_i = \sum_{k=1}^{3} |Q_i - Q_{adj}|$$  \hspace{1cm} (18)

where $E_i$ is the criterion value for the cell, $Q$ represents the values of the density, velocity magnitude and energy evaluated at the cell centroid, and $adj$ denotes the cell neighboring cell $i$ and face $k$. The other two selected criteria are the Laplacian of the velocity magnitude and temperature. Using the divergence theorem to integrate over the cell volume, the two error indicators are obtained from

velocity:

$$E_i = \sum_{k=1}^{3} |A - B|$$  \hspace{1cm} (19)

where

$$A = \left( u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) dy$$

$$B = \left( u \frac{\partial v}{\partial y} + v \frac{\partial v}{\partial y} \right) dx$$

temperature:

$$E_i = \sum_{k=1}^{3} \left| \left( \frac{\partial T}{\partial x} \right) dy - \left( \frac{\partial T}{\partial y} \right) dx \right|$$  \hspace{1cm} (20)
where all the quantities between the absolute value signs are evaluated at the cell faces.

After the criterion value for each cell have been computed, the mean and standard deviation for the distributions are calculated.

\[ \text{Mean} = \frac{\sum_{i=1}^{m} E_i}{m} \]  
\[ \delta = \sqrt{\frac{\sum_{i=1}^{m} (E_i - \text{Mean})^2}{m}} \]

To identify a cell for adaptation, the error indicator of the cells is compared to the mean plus a factor of the standard deviation for the corresponding distribution. Thus, a cell is marked for refinement if one or more of its five criteria values is above a set threshold

**Flag cell for refinement if:**

\[ E_i \geq \text{Mean} + \alpha \delta \]  \hspace{1cm} (23)

where \( \alpha \) is a user defined factor to control the threshold levels. Similarly, a node can be identified for removal if all five criteria values for its surrounding cells are below the specified levels

**Flag node for coarsening if:**

\[ \sum_{j=1}^{\text{valence}_j} (E_{\text{cell}(jv)} < \text{Mean} - \beta \delta) = \text{valence}_j \]  \hspace{1cm} (24)

where \( \beta \), like \( \alpha \), is a user input for the algorithm, \( \text{cell}(jv) \) are the cells that share the common node \( j \) and \( \text{valence}_j \) is the number of cells that surround the node.

Although these criteria generally work well, problems such as those suggested by Warren et al [27] have been observed. Due to the method from which the criteria are computed, the "refinement magnets" [20] phenomenon was detected during the refinement process. For shock waves the selected error indicators remain approximately constant as the cells are refined. Consequently, shock waves will always be targeted for refinement while zones containing weaker flow interactions may be coarsened. To alleviate this problem, upper and lower limits were placed on the cell volumes.

**Mesh Modification**

For the refinement or coarsening of the mesh, four important processes must be considered. They are

- Creation of a buffer zone
- Refinement of the boundary
- Coarsening of the domain
- Refinement of the domain

Although these procedures can be performed in any order, the order shown minimizes the memory requirement.

The creation of a buffer zone [9] provides a transition region between the coarse and fine cells. The buffer zone is created by imposing two constraints. First, the neighboring cells of those identified for refinement are marked for refinement as well. Secondly, the parameters \( \alpha \) and \( \beta \) are selected so that a region will exist where neither refinement nor coarsening will occur.

The next mesh modification is performed at the domain boundary. Since an accurate representation of the physical domain is desired during the refinement, the added nodes should be located on the exact physical boundary instead of the numerical approximation. For rectilinear boundaries, the numerical and physical edges coincide. However, for curved physical boundaries, a series of linear edges are used to approximate the curvature of the boundary. Thus, as illustrated in Fig. 2, the added nodes should lie on the physical boundary rather than the previous numerical representation.

To save memory, the coarsening of the domain is completed before refinement. A node is removed only if all of its surrounding cells are marked for
coarsening. However, an additional constraint is imposed so that large sections would not be coarsened during one modification step. Thus, neighboring nodes of a removed node are not allowed to be deleted, thereby preventing formation of large cavities. This restriction is especially important at a boundary since it would be undesirable to have the boundary completely removed during coarsening.

The final part of the mesh modification is the enrichment of the regions where flow structures exist. Fig. 3 illustrates several possible locations where new nodes can be added. All of these locations have been used by various researchers but preference for a particular location has not been shown. The addition of multiple nodes would of course be more memory intensive than the use of a single node. Thus, for the current research, a node is added to the centroid of the cells which have been identified for adaptation.

Remeshing

Once the mesh has been enhanced, the reconnection between the previous nodes and the newly adjusted ones has to be performed. There are two approaches. The first is local reconnection which is shown in Fig. 3. Local remeshing can be performed very quickly, but a disadvantage may be the obtuse triangular cells that are sometimes formed. Parameters have been developed by Zhang et al [30] and Parhasarathy et al [21] to test the quality of a cell. Once a "bad" cell is determined, several techniques (e.g., diagonal swapping, grid cure, grid smoothing, and mesh relaxation) are performed to adjust the mesh so that better cells are obtained over all.

The other approach is to perform a global reconnection of the domain. Since the Delaunay triangulation can be applied, the final mesh that is obtained will be optimized, i.e. for the given node distribution the maximum angle of each cell has been minimized. The selected method for the current work uses Merriam's unstructured grid generator, UNSTRUCT [17], which employs an Advancing Front technique. Since the scheme is also used for the initial generation of the mesh,
Table 1: Controlling parameters used in the adaptation process for the flat plate boundary layer and NACA 0012 airfoil computations

<table>
<thead>
<tr>
<th>Criterion</th>
<th>Boundary Layer</th>
<th>NACA 0012 Airfoil</th>
</tr>
</thead>
<tbody>
<tr>
<td>Difference of density</td>
<td>0.65 -0.60</td>
<td>1.90 -1.85</td>
</tr>
<tr>
<td>Difference of velocity</td>
<td>1.40 -1.33</td>
<td>-0.34 0.35</td>
</tr>
<tr>
<td>Difference of energy</td>
<td>3.05 -2.70</td>
<td>0.00 0.06</td>
</tr>
<tr>
<td>Laplacian of velocity</td>
<td>0.10 -0.07</td>
<td>-0.44 0.45</td>
</tr>
<tr>
<td>Laplacian of temperature</td>
<td>0.00 0.01</td>
<td>0.60 -0.55</td>
</tr>
<tr>
<td>Maximum cell volume</td>
<td>7.86 \times 10^{-4}</td>
<td>9.00 \times 10^{-1}</td>
</tr>
<tr>
<td>Minimum cell volume</td>
<td>1.50 \times 10^{-5}</td>
<td>8.10 \times 10^{-7}</td>
</tr>
</tbody>
</table>

RESULTS

The adaptive grid algorithm has been applied in the analysis of supersonic flat plate boundary layer and steady, viscous flow past a NACA 0012 airfoil. These two cases provide many different flow structures to test the accuracy of the code. These features include a weak shock wave, a viscous boundary layer, a stagnation region and a wake. The results from these two calculations will be compared with theoretical solutions and other computational works.

Criteria Parameters

Before the results of the computations are presented, the selection of the controlling parameters for the adaptation process are discussed. In Table 1, the refinement, \( \alpha \), and coarsening, \( \beta \), parameters for the five error indicators are shown. Negative values for the parameters indicate relative position of the threshold level to its mean. (See definitions of threshold, Eq. (23) and (24).) The other controlling parameters shown in Table 1 are the upper and lower limits for cell volumes. (The characteristic length scale for the boundary layer is the plate length and the chord length for the airfoil.) The cell volume limits are used to prevent the “refinement magnets” phenomenon from occurring, as discussed in the previous section. The upper limit is obtained by increasing the largest cell volume of the initial
mesh by a factor of three and the lower limit is about one half the smallest cell volume.

**Flat Plate Boundary Layer**

The supersonic flat plate boundary layer with free stream Mach number $M_{\infty} = 2.0$ and the Reynolds number based on the plate length $Re_{\text{plate}} = 6.5 \times 10^4$ is used to examine the accuracy of the adaptive code. Two computations were performed. The first computation uses an unadapted unstructured grid. The other employed the adaptive scheme with the result from the first computation as its initial condition. The two meshes are presented in Fig. 4 and 5 and the details for the meshes are provided in Table 2. The increase of nodes and cells is about a factor of 2 between the adapted and unadapted meshes. An examination of Fig. 4 and 5 show the regions where nodes have been added. It is noticed that the regions for the shock, expansion fan (behind shock) and boundary layer have all been refined while the area in front of the shock has been coarsened. The density contours from the two calculations are illustrated in Fig. 6 and 7 for the unadapted and adapted cases, respectively. From these figures, it can be seen that the shock wave and expansion fan have been better resolved with the adaptive computation. A comparison of the density distribution along two horizontal lines, presented in Fig. 8, better illustrates the increased clarity of the shock wave with adaptation. The boundary layer is virtually the same for both analyses. This is observed because the initial grid provides adequate resolution for the boundary layer. A closer examination of the boundary layer can be obtained by considering the velocity and temperature profiles at $z = 1.0$ (corresponding to $Re_{x} = 3.95 \times 10^4$), illustrated in Fig. 9 and 10, respectively, along with the comparison to the Blasius solution [28]. Both of the computational results are in good agreement with the theoretical results.

It is interesting to observe the adaptive solution to be slightly less accurate near the boundary layer edge than the unadapted result. Earlier adaptation attempts using only the undivided difference criteria, equation (18), resulted in poorer agreement for this region. Since the small gradients that exist at the outer edge of the boundary layer were expected to be the cause of the poor agreement, the two Laplacian criteria, equations (19) and (20), were introduced. Because the Laplacian identifies regions with large curvature, the additional two criteria did improve the numerical results. Nevertheless, the present adaptive results indicate that further research is necessary to develop more efficient criteria for regions near the boundary layer edge.

**NACA 0012 Airfoil**

The second case is steady flow past a NACA 0012 airfoil at zero angle of attack. The free stream Mach number $M_{\infty} = 0.2$ and the Reynolds number (based on the airfoil chord) $Re_{c} = 10^4$. Similar to the flat plate boundary layer case, a converged solution for the airfoil is first obtained with an unadapted unstructured mesh. Using the converged solution, the adaptation process is applied to the grid. The results after four refinement steps are compared with that of the unadapted analysis and the solution performed by Ghosh Choudhuri [5].

Before the comparisons of the three solutions are made, it is of interest to see which regions are selected by the individual error indicators. Since the flow field is everywhere subsonic [13], the main structures of interest are expected to be the stagnation region, boundary layer, and the wake behind the airfoil. The areas identified by the five individual criteria are presented in Fig. 11, 12, 13, 14 and 15 for the differences of density, velocity and energy and the Laplacian of velocity and temperature, respectively. In addi-

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Node</th>
<th>Cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>Non-adapted</td>
<td>2990</td>
<td>5760</td>
</tr>
<tr>
<td>Adapted</td>
<td>5604</td>
<td>11038</td>
</tr>
</tbody>
</table>
Table 3: Grid information for the initial and final meshes

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Nodes</th>
<th>Cells</th>
<th>Nodes on Boundary</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>Airfoil</td>
</tr>
<tr>
<td>Initial</td>
<td>13319</td>
<td>26278</td>
<td>180</td>
</tr>
<tr>
<td>Final</td>
<td>26752</td>
<td>52867</td>
<td>591</td>
</tr>
</tbody>
</table>

Table 4: Comparison of lift and drag coefficient for the non-adapted, adapted and Ghosh Choudhuri computation

<table>
<thead>
<tr>
<th>Computation</th>
<th>$C_d$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Non-adapted</td>
<td>0.0402</td>
</tr>
<tr>
<td>Adapted</td>
<td>0.0394</td>
</tr>
<tr>
<td>Ghosh Choudhuri</td>
<td>0.0401</td>
</tr>
</tbody>
</table>

...
of about 2% between the three analyses. Since the estimated numerical uncertainty of Ghosh Choudhuri's result is approximately 2%, the observed difference is within computed errors. The cause of the variation may be related to the coarser cells that are formed in the trailing edge region. The surface pressure coefficients, Fig. 30, predicted by the three computations are also in close agreement to each other. A more detailed comparison for the surface pressure coefficients at the leading edge is illustrated in Fig. 31. Again, a 2% difference between the adapted solution and Ghosh Choudhuri’s result is observed for the pressure coefficients.

ACKNOWLEDGMENTS

The authors would like to express their appreciation to Dr. Marshal Merriam, NASA Ames Research Center, for providing the unstructured mesh generator, UNSTRUCT. The development of the code was supported by supercomputer grants from the National Center for Supercomputer Applications (NCSA).

REFERENCES


Figure 4: Unadapted mesh for supersonic boundary layer

Figure 5: Final adapted mesh for supersonic boundary layer
Figure 6: Plot of density contours without adaptation

Figure 7: Plot of density contours with adaptation
Figure 8: Density distribution along the plate at two y locations (y = 0.1 and y = 0.3) (line 1 is for the unadapted results and line 2 is for the adapted results)
Figure 9: Comparison of velocity profile obtained with and without adaptation to theoretical solution

Figure 10: Comparison of temperature profile obtained with and without adaptation to theoretical solution
Figure 11: Regions selected for refinement by the difference of density

Figure 12: Regions selected for refinement by the difference of velocity
Figure 13: Regions selected for refinement by the difference of total energy

Figure 14: Regions selected for refinement by the Laplacian of velocity
Figure 15: Regions selected for refinement by the Laplacian of temperature

Figure 16: Regions for refinement with the combination of all criterias
Figure 17: Plot of instantaneous streamlines with the adaptive region
Figure 18: Initial mesh for the NACA 0012 airfoil analysis, entire domain

Figure 19: Final mesh for the NACA 0012 airfoil analysis, entire domain
Figure 21: Final mesh for the NACA 0012 airfoil analysis, near airfoil
Figure 22: Initial mesh for the NACA 0012 airfoil analysis, at leading edge

Figure 23: Final mesh for the NACA 0012 airfoil analysis, at leading edge
Figure 24: Initial mesh for the NACA 0012 airfoil analysis, at trailing edge

Figure 25: Final mesh for the NACA 0012 airfoil analysis, at trailing edge
Figure 26: Density distribution along two vertical cuts of the airfoil domain (\(x = -0.255\) and \(x = 0.755\))

Figure 27: Pressure distribution along two vertical cuts of the airfoil domain (\(x = -0.255\) and \(x = 0.755\))
Figure 28: u-velocity component distribution along two vertical cuts of the airfoil domain ($z = -0.255$ and $z = 0.755$)

Figure 29: v-velocity component distribution along two vertical cuts of the airfoil domain ($z = -0.255$ and $z = 0.755$)
Figure 30: Comparison of surface pressure coefficient for the non-adapted, adapted and Ghosh Choudhuri computation.

Figure 31: Comparison of surface pressure coefficient at the leading edge of the airfoil.