A 3D Unstructured Grid Method for Incompressible Viscous Flows

by Takanori Hino*, Member

Summary

An unstructured grid method for simulating three-dimensional incompressible viscous flows is presented. The governing equations to be solved numerically are the Navier–Stokes equations with artificial compressibility. The spatial discretization is based on a finite volume method for unstructured grid system. Second order accuracy in space is achieved using a flux-difference-splitting scheme with MUSCL approach for inviscid terms and a central difference scheme for viscous terms. Time integration is carried out by the backward Euler method. The linear system derived from the linearization in time is solved by the Gauss-Seidel iteration. For the analysis of high Reynolds number flows, the one equation turbulence model proposed by Spalart and Allmaras is used. The turbulence equation is solved in a similar way as the Navier-Stokes equations. Brief description of the numerical method is given together with some computational results for flows around a 2D wing section and a ship form.

1. Introduction

The rapid improvement of computer performance makes it possible to apply large scale computations to practical engineering problems. In fluid engineering, computational fluid dynamics (CFD) is now being used as a practical tool for predicting flows around complex configurations.

One of the difficulties associated with CFD analysis for complex geometries is grid generation. As configuration becomes more complex, grid generation requires larger amount of efforts. This is particularly true in case that traditional structured grids which are based on regular arrays of hexahedral cells are used. One way to overcome grid generation problems is unstructured grid methods which employ irregularly connected cells of various shapes. Unstructured grid methods offer greater flexibility for handling complex geometries compared with structured grid counterparts.

Aerospace engineering has been the field that leads CFD technology development. Therefore, unstructured grid methods were developed initially for solving compressible flow equations around complex geometries like a complete aircraft. Since then the methods have been applied to various fluid problems. Incompressible flow solvers have also been developed based on a pressure correction method or an artificial compressibility method.

In a hydrodynamics field, Hino developed unstructured grid methods for 2D incompressible inviscid flows with a free surface and for 2D turbulent free surface flows. The present study is concerned with the extension of the above methods to three dimensional flows. In the CFD applications for ship hydrodynamics, there are increasing needs for flow predictions around more complex configurations, because practical ships whose hull forms are complex enough are usually equipped with stern appendages, a rudder and a propulsor. Using a three-dimensional unstructured grid Navier–Stokes solver, such complex flow fields can be predicted without too much efforts for grid generation.

In the following, the governing equations are given and the spatial discretization is described followed by the explanation of the time integration scheme. Then the turbulence model used and the boundary conditions for numerical computations are briefly shown. Finally, numerical results are presented for flows around a 2D wing section and a ship hull form.

2. Numerical Procedure

2.1 Governing Equations

The governing equations are the three-dimensional Reynolds averaged Navier–Stokes equations for incompressible flows. With the introduction of artificial compressibility, they are written in a vector form as

\[
\frac{\partial q}{\partial t} + \frac{\partial (e - e^a)}{\partial x} + \frac{\partial (f - f^a)}{\partial y} + \frac{\partial (g - g^a)}{\partial z} = 0 \tag{1}
\]

where

\[
q = \begin{bmatrix}
p \\
u \\
v \\
w
\end{bmatrix} \tag{2}
\]
is the vector of flow variables which consist of $p$, pressure and $(u, v, w)$, the $(x, y, z)$-components of velocity. The inviscid fluxes $e, f$ and $g$ are defined as

$$
e = \begin{bmatrix} \beta u \\ u^2 + p \\ u \end{bmatrix}, \quad f = \begin{bmatrix} \beta v \\ v u \\ v^2 + p \\ v \end{bmatrix}, \quad g = \begin{bmatrix} \beta w \\ w u \\ w^2 + p \\ w \end{bmatrix}$$

where $\beta$ is a parameter for artificial compressibility. $e^v$, $f^v$ and $g^v$ are the viscous fluxes:

$$
e^v = \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \end{bmatrix}, \quad f^v = \begin{bmatrix} 0 \\ \tau_{yy} \\ \tau_{yz} \end{bmatrix}, \quad g^v = \begin{bmatrix} 0 \\ \tau_{zx} \\ \tau_{zz} \end{bmatrix}$$

where

$$\tau_{aa} = \left( \frac{1}{R + \nu_t} \right) \left( \frac{\partial q_a}{\partial x_a} + \frac{\partial q_a}{\partial x_a} \right)$$

and if $a = x$ then $u_a = u$ and $x_a = x$ and if $a = y$ then $u_a = v$ and $x_a = y$ and so forth. In the above expressions all the variables are made dimensionless using the reference density $\rho_0$, velocity $U_0$ and length $L_0$. $R$ is the Reynolds number defined as $U_0L_0/\nu$ where $\nu$ is the kinematic viscosity. $\nu_t$ is the nondimensional kinematic eddy viscosity determined by an appropriate turbulence model.

### 2.2 Spatial Discretization

Spatial discretization is based on a finite volume method for an unstructured grid. In the present approach, a cell centered layout is adopted in which flow variables $q$ is defined at the centroid of each cell and a control volume is a cell itself. Cell shapes the present solver can cope with are tetrahedron, prism, pyramid or hexahedron and face shapes are either triangular or quadrilateral. These four types of cells give larger flexibility in handling complex geometries.

Volume integration of Eq. (1) over a cell yields

$$\int_{V_i} \left( \frac{\partial q}{\partial t} + \beta \frac{\partial e}{\partial x} + \frac{\partial f}{\partial y} + \frac{\partial g}{\partial z} \right) dV = 0 \quad (3)$$

The first term in the integral can be expressed as the time derivative of the product of the cell volume $V_i$ and the cell averaged value of flow variables $q$, since the grid is stationary in the current applications. The remaining terms are converted into surface integration over cell faces using the divergence theorem. This yields the semi-discrete form of the governing equation as follows.

$$\frac{\partial (V_i q_i)}{\partial t} + \sum_j F_{i+1/2} - \sum_j R_{i+1/2} = 0 \quad (4)$$

where $i$ is a cell index and $j$ is the index of neighboring cells of the cell $i$. $(i+j)/2$ denotes the face between cells $i$ and $j$ as shown in Fig. 1.

$F$ and $R$ are the inviscid and viscous fluxes defined as

$$F = eS_x + fS_y + gS_z, \quad R = e^vS_x + f^vS_y + g^vS_z$$

($S_x$, $S_y$, $S_z$) are the $(x, y, z)$-components of the area vector of a cell face in the direction from the cell $i$ to the cell $j$.

Components of the inviscid fluxes $F$ are

$$F(q) = \begin{bmatrix} \beta U \\ u(U+pS_x) \\ v(S_y) \\ w(S_z) \end{bmatrix}$$

where $U = uS_x + vS_y + wS_z$. The inviscid fluxes are evaluated by the upwind scheme based on the flux-difference splitting of Roe\(^\ddagger\). This can be expressed as

$$F_{i+1/2} = \frac{1}{2} \left[ F(q^i) + F(q^j) - |A| (q^j - q^i) \right]$$

where $q^i$ and $q^j$ are the flow variables on the right side and the left side of a cell face, respectively. $|A|$ is defined in the following way. First, let $A$ be the Jacobian of the inviscid flux $F$ at a cell face:

$$A = \frac{\partial F(q^i + q^j)}{\partial q}$$

The eigenvalues of $A$ are $U$, $U$, $U+c$, $U-c$ where $c$ is the pseudo-speed-of-sound defined as:

$$c = \sqrt{U^2 + \beta (S_x^2 + S_y^2 + S_z^2)}$$

By using the right-eigenvector $R$, $A$ can be expressed as $A = RAR^{-1}$ where $A = \text{diag}(U, U, U+c, U-c)$. Finally, $|A|$ is given by $|A| = |R|A|R^{-1}$ with $|R| = \text{diag}(|U|, |U|, |U+c|, |U-c|)$.

To maintain the second order accuracy in space, $q^i$ and $q^j$ are extrapolated using the Taylor expansion as follows:

$$q^j = q_i + \nabla q_i \cdot (x_{i+1/2} - x_i)$$

$$q^j = q_i + \nabla q_i \cdot (x_{i+1/2} - x_i)$$

where $x_{i+1/2}$ is the coordinate of the center of face $(i + j)/2$ and $x_i$ and $x_j$ are the coordinates of the centroid of cells $i$ and $j$, respectively. $\nabla q_i$ is the gradient of $q$ at the cell centroid $i$ and this can be computed by applying the divergence theorem to the cell $i$ with the values of $q$ at the cell $i$ and its neighbors $j$.

Viscous fluxes components are written as

$$R(q) = \begin{bmatrix} 0 \\ S_x \xi_x + S_y \xi_y + S_z \xi_z \\ S_x \xi_x + S_y \xi_y + S_z \xi_z \\ S_x \xi_x + S_y \xi_y + S_z \xi_z \end{bmatrix}$$

where

$$\xi = \frac{1}{R + \nu_t} \left( \frac{\partial q}{\partial x} + \frac{\partial q}{\partial x} \right)$$

and if $a = x$ then $u_a = u$ and $x_a = x$ and if $a = y$ then $u_a = v$ and $x_a = y$ and so forth. In the above expressions all the variables are made dimensionless using the reference density $\rho_0$, velocity $U_0$ and length $L_0$. $R$ is the Reynolds number defined as $U_0L_0/\nu$ where $\nu$ is the kinematic viscosity. $\nu_t$ is the nondimensional kinematic eddy viscosity determined by an appropriate turbulence model.

### 2.2 Spatial Discretization

Spatial discretization is based on a finite volume method for an unstructured grid. In the present approach, a cell centered layout is adopted in which flow variables $q$ is defined at the centroid of each cell and a control volume is a cell itself. Cell shapes the present solver can cope with are tetrahedron, prism, pyramid or hexahedron and face shapes are either triangular or quadrilateral. These four types of cells give larger flexibility in handling complex geometries.
The computation of $R(i + j)/2$ requires velocity gradient on a cell face. These are computed again by applying the divergence theorem to another control volume surrounding a cell face as shown in Fig. 2. The values of $q$ at the centroids $i$ and $j$ and at the nodes $k, k+1$ ... surrounding the face $(i+j)/2$ are used for the surface integration. For example, $\delta q/\delta x$ is computed as

$$\frac{\delta q}{\delta x} = \frac{1}{V^*} \sum_{\text{faces}} n S_{r}$$

$$= \frac{1}{V^*} \left( \frac{h_i + h_{i+1}}{3} S_{r,i,k} + \frac{h_j + h_{j+1}}{3} S_{r,j,k+1} + \ldots \right)$$

Fig. 2 Control volume for the evaluation of velocity gradient.

where $V^*$ is the volume of the current control volume and $S_{r,x,y,z}$ is the $x$ component of the outward area vector of the face formed by the nodes $a, b, c$. The velocity values at the nodes $k, k+1$ etc. are computed from the values at the centroids by simple averaging.

### 2.3 Time Integration

The backward Euler scheme is used for the time integration in which the governing equation is written as

$$V_i \frac{\partial q_j}{\partial t} + \sum \frac{F_{i+1,j,z}^n - F_{i,j,z}^{n+1}}{\Delta t} = 0$$

where

$$\Delta q = q^{n+1} - q^n$$

and the superscripts denote the time step. $\Delta t$ is the time increment for local time stepping in which $\Delta t$ is determined cell by cell in such a way that the CFL number is globally constant. The linearization of the inviscid flux $F^{n+1}$ with respect to time is made as follows

$$F^{n+1} = F^n + \frac{\partial F}{\partial q} \cdot \Delta q$$

When the Jacobian $\partial F/\partial q$ is evaluated, the flux $F$ is computed with the first order accuracy by setting $q^i = q_c, q^f = q_l$.

Thus, the inviscid flux is expressed as

$$F_{i,j,z}^{n+1} = F_{i,j,z}^n + \frac{1}{2} \left| A \cdot (\Delta q_i + A \cdot \Delta q_j - |A| \cdot (\Delta q_i - \Delta q_j)) \right|$$

where

$$A = \frac{\partial F(q)}{\partial q}$$

In the similar manner the viscous flux is linearized in time as

$$R^{n+1} = R^n + \frac{\partial R}{\partial q} \cdot \Delta q$$

For the evaluation of the Jacobian $\partial R/\partial q$, $R$ is approximated by neglecting the contribution from the values at the nodes, i.e., $q_k, q_{k+1}$. Thus, $R$ becomes dependent only on the values on the cell centroid, $q_i$ and $q_j$ and $\partial R/\partial q \cdot \Delta q$ becomes

$$\frac{\partial R}{\partial q} \cdot \Delta q = B \cdot (\Delta q_i - \Delta q_j)$$

where

$$B = \left( \frac{1}{R^* + \nu_r} \frac{1}{3 V^*} \right) \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & |S|^2 & S_x S_y & S_x S_z \\ 0 & S_y S_z & |S|^2 & S_x S_z \\ 0 & S_z S_x & S_x S_y & |S|^2 + S_z^2 \end{bmatrix}$$

with

$$|S|^2 = S_x^2 + S_y^2 + S_z^2$$

Eq. (5) now becomes

$$V_i \frac{\partial q_j}{\partial t} + \sum \frac{\partial F_{i,j,z}^{n+1} - \partial F_{i,j,z}^n}{\Delta t} + \sum \frac{1}{2} \left( A \cdot \Delta q_i + A \cdot \Delta q_j - |A| \cdot (\Delta q_i - \Delta q_j) \right) + \sum B \cdot (\Delta q_i - \Delta q_j) = 0 \tag{6}$$

The delta terms are rearranged into the form as follows.

$$\left\{ \frac{V_i}{\partial t} I + \sum \left( \frac{A}{2} + \frac{A}{2} \cdot B \right) \right\} \cdot \Delta q_i$$

$$+ \sum \left( \frac{A}{2} \cdot B \cdot (\Delta q_i - \Delta q_j) \right)$$

$$+ \sum \frac{F_{i,j,z}^{n+1} - \sum R_{i,j,z}^{n+1}}{\Delta t} = 0 \tag{7}$$

The equation above is a linear equation with respect to $\Delta q$. In order to solve this equation, the symmetric Gauss-Seidel (SGS) iteration is adopted in the present scheme. To achieve fast convergence, the cells are ordered from upstream to downstream. The Gauss-Seidel sweep is carried out from the upstream cell to the downstream first, then the second sweep follows the reverse order. The following sweeps change the direction alternately. Typically 20 SGS sweeps are performed at each time step.

### 2.4 Turbulence model

A turbulence model is essential for simulating high Reynolds number flows of practical interests. The simplest among various turbulence models are algebraic models or zero-equation models in which eddy viscosity can be computed with algebraic formula using velocity profile information. Implementation of this kind of models on unstructured grids encounters difficulties because velocity profile is not easily defined without inherent grid structure.

One-and two-equation models are more suitable for
unstructured grid methods although they require solution of one or two field equations which are an advection-diffusion equation with source terms in addition to flow equations. The one equation model proposed by Spalart and Allmaras is used in the present scheme. This model has been applied to viscous flows around a ship and it has turned out that the model can predict viscous flow fields a little more reasonably than an algebraic model.

The numerical method employed for the solution of turbulent equation is similar to the one for flow equations except that the advection terms are evaluated by the first-order upwind scheme.

2.5 Boundary Conditions

The cell faces on the boundaries are classified as the boundary faces. The boundary conditions are implemented by giving the appropriate fluxes on the boundary faces. The boundary types and the conditions to be imposed are shown in Table 1. In case of the Dirichlet conditions, the fluxes can be computed using given values, while in case of the Neumann conditions, the flow variables on the face is set equal to those at the cell that contains the current face. In the time integration process, all the boundary conditions are treated implicitly.

3. Results

3.1 NACA 0012 Wing Section

The first application is for turbulent flow computations around a NACA 0012 wing section. Although this flow field is two-dimensional, three-dimensional computations are carried out to examine the fundamental performance of the present solver.

Reynolds number based on a chord length is $R = 3 \times 10^6$. Angle of attack is zero and only a half side is considered. Four unstructured grids are generated from the $112 \times 1 \times 64$ (streamwise, span and normal directions) H-H grid. The minimum spacing in the direction normal to the wall is $5.8 \times 10^{-6}$.

The first grid consists of hexahedral cells. A cell shape and connectivities of this grid are equivalent to the original structured grid. The second grid is obtained by dividing each hexahedron into two prisms in such a way that triangular faces are located on the foil surface. The third one is a hybrid grid in which the hexahedral cells are placed inside the boundary layer and the wake and the remaining cells are split into two prisms. The last grid generated is another hybrid grid. This grid is obtained by putting the prismatic cells in the boundary layer and the wake and the cells on top of them are divided into five tetrahedra. The number of cells are 7,168 for a hexahedral grid, 14,336 for a prismatic grid. The hexahedral-prismatic hybrid grid contains 4,480 hexahedra and 5,376 prisms. The prismatic-tetrahedral grid contains 8,960 prisms and 13,440 tetrahedra. Note that the number of nodes for each grid is the same and 36,725. Figs. 3 shows the partial views of the hexahedral grid and the hexahedral-prismatic grid. Appearance of the hexahedral grid is identical to the structured grid, while the hybrid grid shows the hexahedral cells (the quadrilateral faces) near the solid wall and the symmetric plane and the prismatic cells (the triangular faces) in the outside.

Fig. 4 shows the pressure distribution around a wing computed with the hybrid hexahedral-prismatic grid. The results with the other unstructured grids give essentially the same plot.

Fig. 5 depicts the comparison of computed pressure coefficients on a NACA 0012 wing section. Angle of attack = 0 degree, Reynolds number $R = 3 \times 10^6$. 

Table 1 Boundary types and boundary conditions.

<table>
<thead>
<tr>
<th>Boundary Type</th>
<th>$\phi_y/\phi_n=0$</th>
<th>$w=0$</th>
<th>$v=0$</th>
<th>$u=0$</th>
<th>$\psi=0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solid Wall</td>
<td>$\phi_y/\phi_n=0$</td>
<td>$w=0$</td>
<td>$v=0$</td>
<td>$u=0$</td>
<td>$\psi=0$</td>
</tr>
<tr>
<td>Inflow</td>
<td>$\phi_y/\phi_n=0$</td>
<td>$w=0$</td>
<td>$v=0$</td>
<td>$u=0$</td>
<td>$\psi=0$</td>
</tr>
<tr>
<td>Outflow</td>
<td>$\phi_y/\phi_n=0$</td>
<td>$w=0$</td>
<td>$v=0$</td>
<td>$u=0$</td>
<td>$\psi=0$</td>
</tr>
<tr>
<td>Side</td>
<td>$\phi_y/\phi_n=0$</td>
<td>$w=0$</td>
<td>$v=0$</td>
<td>$u=0$</td>
<td>$\psi=0$</td>
</tr>
<tr>
<td>Y-symmetry</td>
<td>$\phi_y/\phi_n=0$</td>
<td>$w=0$</td>
<td>$v=0$</td>
<td>$u=0$</td>
<td>$\psi=0$</td>
</tr>
<tr>
<td>Z-symmetry</td>
<td>$\phi_y/\phi_n=0$</td>
<td>$w=0$</td>
<td>$v=0$</td>
<td>$u=0$</td>
<td>$\psi=0$</td>
</tr>
</tbody>
</table>
coefficients on a wing surface. The structured grid data shown is computed by the NS solver FRESH\textsuperscript{12} which is a finite volume multigrid method for incompressible Navier-Stokes equations. All the unstructured grid results are in reasonably good accordance with the structured grid result. This suggests that the present solver can be applied to grids consist of the hybrid combination of various cell shapes without loss of accuracy.

3.2 SR 196 A Ship Form

The second case is a truly three dimensional flow field around a ship hull form. A ship model is SR 196 A which is a typical VLCC hull with a bulbous bow. The Reynolds number based on the length between perpendiculars \( L_{pp} \) is \( 1.9 \times 10^6 \) which corresponds to a model scale.

The grid used is based on the \( 80 \times 24 \times 40 \) (streamwise, span and normal direction) structured grid of H-O topology. The minimum spacing on a hull in the normal direction is \( 4.8 \times 10^{-6} \) \( L_{pp} \).

Two unstructured grids are generated from the above grid. The first grid is a hexahedral grid which is equivalent to the structured grid. The second grid is a prismatic grid in which all the hexahedral cells of the structured grid are divided into two prisms. The partial views of two grids are shown in Figs. 6. Distributions of triangular faces on a body and a symmetric plane can be observed in the prismatic grid. The hexahedral grid contains 76,800 cells and 236,480 faces while the prismatic one contains 153,600 cells and 392,000 faces. The prismatic grid has twice as many cells and 1.66 times as many faces as the hexahedral grid.

All computations run using local time stepping with initial \( CFL \) of 10 then \( CFL \) is increased linearly up to 100 in the first 50 steps. The residual is reduced approximately five orders of magnitude with 500 steps.

In Figs. 7, computed pressure distributions on a ship hull and symmetric planes are shown. Two unstructured grid solutions are shown together with the structured grid solution which is again obtained using the NS solver FRESH with the Spalart-Allmaras turbulence model\textsuperscript{12}. Three solutions are virtually the same, although the order of accuracy in the spatial discretization is different; third order for the structured solver and the second order for the unstructured solver.

Fig. 8 shows the resistance coefficients for each computation. Differences of the total resistance coefficients of all three computations are less than 1% of the average value. It turns out that overall accuracy of the unstructured solver is as good as the structured grid method.

Fig. 9 is the comparison of wake distribution at a propeller plane. One of the missions of CFD in ship hydrodynamics is the accurate prediction of a propeller inflow. Unfortunately, this mission has not yet been properly accomplished due to the limitation of the current turbulence modeling. The hexahedral grid case agrees quite well with the structured grid result, because the grid geometry is identical in both cases. As

---

Fig. 6 Partial views of two unstructured grids around an SR 196 A hull. Hexahedral Grid (Top) and Prismatic Grid (Bottom).

Fig. 7 Computed pressure distributions on a hull surface and symmetric planes. Hexahedral Grid (Top), Prismatic Grid (Middle) and Structured Grid (Bottom), SR 196 A Hull, \( R=1.9 \times 10^6 \).

Fig. 8 Computed Resistance coefficients. SR 196 A Hull, \( R=1.9 \times 10^6 \).
was shown in Ref.11), the Spalart-Allmaras turbulence model produces reasonably better flow field than the algebraic model in terms of the hook shape of the wake distribution. However there remain some discrepancies in the detail of flows. The two results with the hexahedral and the structured grids show similar tendency. The hook shape of the wake contours is not reproduced well and the longitudinal vortex is still not strong enough.

The prismatic grid result, however, shows slightly different wake distribution. The hook shape seems to be a little pronounced compared with the hexahedral cases. Note that the numbers of cells and faces in the prismatic grid is larger than in the hexahedral grid. Therefore, the spatial resolution of the prismatic grid is somewhat finer than the other two. This suggests that if a sufficient resolution is provided, a flow field can be more accurately predicted. A solution adaptive grid strategy based on unstructured grid methods appears to be the most effective in this situation.

4. Concluding Remarks

An unstructured grid method for simulating three-dimensional incompressible viscous flows was developed. The numerical method solves the Navier-Stokes equations with artificial compressibility using a finite volume method on an unstructured grid system. Time integration is carried out by the backward Euler method. The derived linear system is solved by the symmetric Gauss-Seidel iteration. The Spalart-Allmaras equation turbulence model is used for high Reynolds number flow simulations.

The method was applied to the flow simulations around a two dimensional wing section with various cell shape combinations. The results show good agreement to each other and the fundamental performance of the method was verified.

Three dimensional computations were made for a tanker hull. The grids used are the hexahedral grid and the prismatic grid generated out of the structured grid. The results with both grids show reasonable agreement with the structured grid computation.

Future works are summarized as follows: In order to apply the present flow solver to complex flow configurations, unstructured grid generation procedures should be developed. Also, the method should be able to cope with free surface flows, since the most flow problems in ship hydrodynamics are associated with a free surface. Solution adaptive grid method is the way that takes full advantage of unstructured grids and this approach should be investigated.

Acknowledgment

The author would like to thank Mr. N. Hirata for his valuable discussions and for providing the numerical results for a wing section using the structured grid. Also, discussions and suggestions of the CFD group at Ship Research Institute are gratefully acknowledged.

References