Comparison between Structured and Unstructured Grid Generation on Two Dimensional Flows Based on Finite Volume Method (FVM)

Abobaker Mohammed Alakashi, and Dr. Ir. Bambang Basuno

Abstract—Finite volume methods (FVM) had been recognized as one of numerical has proven highly successful in solving problem of fluid mechanics, meteorology, and many other engineering areas. The implementation of the finite volume required spatial discretization of domain of the problem in hand. In this respect one may adopt structured or unstructured grid approach. However their implementation will generate different algorithm although they are used the same Finite Volume scheme. The present work develop two computer codes both used a cell centered Finite volume scheme with different of term of spatial discretization. The first computer code used a structured grid and the second one used unstructured grid. Both computer codes applied for solving two internal flow problems and one external flow problem of flow past through airfoil NACA 0012. Each of flow problems are solved for three different flow condition and their result presented in term of pressure and Mach number distribution along their geometry surfaces. Their comparison result indicates the cell centered Finite volume scheme is a robust scheme, since their capability to produce same result are achievable without strongly influenced by manner how the mesh flow domain created.

Keywords—Structured grid, unstructured grid, Euler solver, Cell-centred scheme

I. INTRODUCTION

The mesh generation plays an important role in the numerical analysis procedure and over the past two decades, efficient methods of grid generation, together with the power of modern digital computers, have been the key to the development of numerical finite-volume (as well as finite-difference and finite-element) solutions of linear and nonlinear partial differential equations in regions with boundaries of complex shape [1].

In the absence of viscous effects, the governing equation of fluid motion pass through an arbitrary body can be represented by Euler equation. This equation is in the form of a nonlinear differential system equation which their analytic solutions are difficult to be found. As a result a numerical approach is required. There are various forms to represent the governing equation of fluid motion. The governing equation of fluid motion in integral form allows one to apply a spatial discretization by use of finite volume approach.

In the relationship between the control volume and the grid cells, Finite Volume method can be classified into three groups. They are namely (1) Cell-centred scheme, (2) Cell-vertex scheme with overlapping control volumes and (3), Cell-vertex scheme with dual control volumes [2, 3]. In the Cell-centred Finite volume approach, the control volume is taken identically as its grid cell and the flow variables are described as the flow variables at the centroid of grid cells [4]. This approach give a better approach compared to the finite difference method which may require a fine grid for the same flow problem [4, 5].

Basically the Cell-centred Finite volume method provides a large number of options in defining the control volumes around which the conservation laws are expressed. Modifying the shape and location of the control volumes associated with a given mesh point, as well as varying the rules and accuracy for the evaluation of the fluxes through the control surfaces, gives considerable flexibility to the Finite volume method [6,7]. In addition to this, discretizing to the integral form of the conservation laws directly ensure that the basic quantities mass, momentum and energy will remain conserved at the discrete level [8, 9]. Beside that the finite volume method allows the spatial discretisation is carried out directly in the physical space, transformations between coordinate systems are no needed any more [9, 10].

The Cell centered Finite volume scheme can be combined with structured or unstructured grid approach in discretizing of the flow domain of the problem in hand. However the two spatial discretizing approaches in their implementation as part of Cell centered finite volume method will generate two different algorithms in view of programming to computer code. As result two computer code need to be developed to handle in solving flow problem based a Cell centered Finite volume scheme in combining structured and unstructured grid generation. The two developed computer applied to the case of two internal flow problems related to the internal flow problem past through a bump and the other one for the case of flow past through cascade. While in the case of external flow problem, these two computer code applied to the flow past through airfoil NACA 0012. Each case of flow problem as mentioned are treated with three different flow conditions. Their comparison results in term of Mach number and pressure distribution along the body surface indicate in a good agreement. These results conclude the cell centered Finite volume scheme represent a robust scheme since this methods is not strongly dependent to the manner how discretizing flow domain are carried out.
II. METHODOLOGY

A. Governing Equation Of Fluid Motion

The governing equation of compressible inviscid two dimensional flows can be written in the integral form for a given region $\Omega$ and boundary $d\Omega$ as

$$\frac{\partial}{\partial t} \int_{\Omega} \bar{Q} dx dy + \int_{\partial\Omega} (\bar{E} dy - \bar{H} dx) = 0$$

Where:

$$\bar{Q} = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{pmatrix}, \quad \bar{E} = \begin{pmatrix} \rho \\ \rho u^2 + p \\ \rho uv \\ \rho uH \end{pmatrix} \quad \text{and} \quad \bar{H} = \begin{pmatrix} \rho \\ \rho v^2 + p \\ \rho uv \\ \rho vH \end{pmatrix}$$

In above equation, the independent variables are $x$, $y$, and $t$, where $x$ and $y$ represent spatial independent and $t$ is temporal independent. Other variables are defined as normally use in CFD analysis, namely the density $\rho$, pressure $p$, component velocity in $x$-direction $u$, the component velocity in the $y$-direction $v$, the total energy $E$ and the total enthalpy $H$ [11].

For a perfect gas, there is a unique a relationship between $E$ and $H$ given as:

$$E = \frac{p}{\gamma - 1} \rho + \frac{1}{2} (u^2 + v^2),$$

and

$$H = E + \frac{p}{\rho}$$

The discretization procedure follows the method of lines in decoupling the approximation of the spatial and temporal terms. The computational domain is divided into quadrilateral cells. At each cell, the conservation laws is applied, such as to the cell ABCD as shown in figure (1). As result a system of ordinary differential equations are obtained and several alternative time stepping schemes can be adopted.

Considering the grid cell ABCD of Figure (1), the side AB has a surface vector $\vec{S}_{AB}$ and a normal vector $\vec{i}_{AB}$. They are defined as:

$$\vec{S}_{AB} = \Delta y_{AB} \vec{e}_y - \Delta x_{AB} \vec{e}_x$$

$$\vec{i}_{AB} = (y_B - y_A) \vec{e}_y - (x_B - x_A) \vec{e}_x$$

The grid cell area of ABCD can be obtained from:

$$\Omega_{ABCD} = \frac{1}{2} \left| \vec{X}_{AC} \times \vec{X}_{BD} \right|$$

Where $\vec{X}_{AB} = \vec{X}_B - \vec{X}_A$

Or in term of coordinate point $(X, Y)$ as:

$$\Omega_{ABCD} = \frac{1}{2} \left| (X_c - X_A)(Y_B - Y_D) - (X_D - X_B)(Y_c - Y_A) \right|$$

The flux vector cross the side surface can be estimated by various approaches. The flux $E$ crossing the side surface AB denoted as $\bar{E}_{AB}$ can be obtain by use of one following approaches.

a. Average of fluxes

$$E_{AB} = \bar{E} \left( \frac{Q_{ij} - Q_{i+1,j}}{2} \right)$$

Where: $E_{ij} = E(Q_{ij})$

b. Flux of the average flow variable:

$$E_{AB} = \bar{E} \left( \frac{Q_{ij} + Q_{i+1,j}}{2} \right)$$

c. Average of fluxes in A&B

$$E_{AB} = \frac{1}{2} (E_A + E_B)$$

In the last approach, approach c, the flux $E_A$ can be determined by firstly defining the flow variable $Q$ at A as given below:

$$Q_A = \frac{1}{4} (Q_{ij} + Q_{i+1,j} + Q_{i+1,j-1} + Q_{ij+1}) \quad (2)$$

Then the average of the fluxes $E_A$ becomes:

$$E_A = \frac{1}{4} (E_{ij} + E_{i+1,j} + E_{i+1,j-1} + E_{ij+1}) \quad (3)$$

In similar manner applies to the flux vectors H. As result the implementation of the finite volume to the Eq. (1) makes that an ordinary differential equation with respect to time as given below[12,13].

$$\frac{\partial}{\partial \tau} \frac{\partial}{\partial \tau} [Q_{ij} \Delta x \Delta y] + \left( E_{i+1,j} - E_{i,j} \right) \Delta y + \left( Q_{ij+1} - Q_{ij} \right) \Delta x = H_{ij} \Delta x \Delta y \quad (4)$$

Or

$$\frac{\partial Q_{ij}}{\partial \tau} + \frac{(E_{i+1,j} - E_{i,j})}{2} \Delta y + \frac{(Q_{ij+1} - Q_{ij})}{2} \Delta x = H_{ij} \quad (5)$$

Through above equation, the flow problems are solved. However to solve above equation one has to make a mesh flow domain appropriately. This can be done by firstly choosing the grid topology.
B. Grid Generation

Many kinds of the grid generation techniques can be used for defining the mesh of computational space. The grid generation based elliptic partial differential equation is the most popular one [14]. It allows the user to prescribe the angle between a grid line and boundary, controlling the grid spacing and their expansion ratio near surfaces. Elliptic grid generation also guarantees a smooth grid in the entire flow domain. Thus, high quality, boundary orthogonal grids can be generated [15].

C. Elliptic Grid Generation

This type of grid generation is motivated by the maximum principle for elliptic partial differential equations. Where the inverse grid transformation, \( \xi(x, y), \eta(x, y) \) can be obtained as the solution of

\[
\begin{align*}
\xi_{xx} + \xi_{yy} &= 0 \\
\eta_{xx} + \eta_{yy} &= 0
\end{align*}
\]

When \( 0 \leq \xi \leq 1 \) and \( 0 \leq \eta \leq 1 \) are monotone on the boundaries and following the maximum principle, it will make and \( \eta \) will stay between these values. Furthermore, there will be no local extreme in the interior, and thus grid lines cannot fold. As result one may solve the Eq. 6 in other form as defined by the following equation.

\[
\begin{align*}
(x^2_i + y^2_i)\xi_{xx} - 2(x_i x + y_i y)\xi_{x\eta} + (x^2_i + y^2_i)\eta_{x\eta} &= 0 \\
(x^2_i + y^2_i)\eta_{xx} - 2(x_i x + y_i y)\eta_{x\eta} + (x^2_i + y^2_i)\eta_{x\eta} &= 0
\end{align*}
\]

Above equation can be discretized by using a second order finite difference approach. The term \( x\xi, x\eta \) and \( x\xi \) becomes:

\[
\begin{align*}
x\xi &= (x_{i+1,j} - x_{i-1,j})/2 \\
x\eta &= (x_{i,j+1} - x_{i,j-1})/2 \\
x\xi \xi &= x_{i+1,j} - 2x_{i,j} + x_{i-1,j} \text{ etc}
\end{align*}
\]

The index \( i \) and \( j \) are \( 1 \leq i \leq n_i \) and \( 1 \leq j \leq n_j \) a uniform subdivision of the \((\xi, \eta)\) coordinates, ini which:

\[
\xi = (i - 1)/(n_i - 1), \text{ and} \quad \eta = (j - 1)/(n_j - 1)
\]

The number of grid points is specified as \( n_i \times n_j \)

To introduce more control over the grid, so called control functions are introduced into (6) [14, 17]. The elliptic equation then becomes

\[
\begin{align*}
\xi_{xx} + \xi_{yy} &= P(\xi, \eta) \\
\eta_{xx} + \eta_{yy} &= Q(\xi, \eta)
\end{align*}
\]

Where, \( P, Q \) are known functions to control the concentration of the inner grid points.

By interchanging the independent and the dependent variables, Eq. (7) becomes:

\[
\begin{align*}
\alpha x\xi\xi - 2\beta x\xi\eta + \gamma x\eta\eta + f^2 (P x\xi + Q x\eta) &= 0, \\
\alpha y\eta\eta - 2\beta y\xi\eta + \gamma y\eta\eta + f^2 (P y\xi + Q y\eta) &= 0,
\end{align*}
\]

\[
\alpha = x^2 + \gamma^2, \quad \beta = x\xi + y\eta, \quad \gamma = x^2 + \gamma^2
\]

\[
J = x\xi\eta + x\eta\xi
\]

Figure 2 show the effect of the value of function \( P \) and \( Q \) to the manner how the grid points will be distributed in the flow domain.

![Figure 2 Effects of the control functions P controls the skewness and Q the spacing](image)

In order one be able to apply a mesh generation based on elliptic partial differential equation one required to define the grid topology. There are three types of grid topologies; they are namely C-, H- grid and O- topology [8, 14]. The present work uses C-topology for the case of external flow past through airfoil and H-topology for the case of flow past through Bump channel [15] and the blade turbines [18].

D. C-Grid Topology

In view of C-grid topology, the elliptic grid generations will guarantees a smooth grid in the entire domain. Thus, high quality boundary orthogonal grids can be generated [6][19].

C-topology is enclosed by one family of grid lines, if there is a wake region, the method will form it. C-grid topology as depicted in the Figure 3. The lines \( \eta = \text{const.} \) start at the far field \( (\xi = 0) \), follow the wake, pass the trailing edge (node b), wrap in clockwise direction round the body, and finally start with the far field once again \( (\xi = 1) \). The other grid lines \( (\xi = \text{const.}) \) driven in normal direction from the body and the wake. Segment a-b is part of the grid line \( \xi = 0 \) which represents a coordinate cut.

Which lead that the segment a-b in the physical space is created into two segments, namely \( a \leq \xi \leq b \) and \( b' \leq \xi \leq a' \).

Where, the nodes on the upper part and the lower part \((b' - a')\), \((a - b)\) of the cut respectively are memorized in two parts in the computer memory. [8, 20, 9]
In the context of C-grid topology, the mesh flow domain is obtained by solving the set of partial differential equation in the form:

\[
\begin{align}
\alpha_{11} \left( \frac{\partial^2 x}{\partial \xi^2} + P \frac{\partial x}{\partial \xi} \right) - 2\alpha_{12} \frac{\partial^2 x}{\partial \xi \partial \eta} + \alpha_{22} \left( \frac{\partial^2 x}{\partial \eta^2} + Q \frac{\partial x}{\partial \eta} \right) &= 0 \\
\alpha_{11} \left( \frac{\partial^2 y}{\partial \xi^2} + P \frac{\partial y}{\partial \xi} \right) - 2\alpha_{12} \frac{\partial^2 y}{\partial \xi \partial \eta} + \alpha_{22} \left( \frac{\partial^2 y}{\partial \eta^2} + Q \frac{\partial y}{\partial \eta} \right) &= 0
\end{align}
\]

(8a)

Where \( P \) and \( Q \) represented the control functions, the metric coefficients \( \alpha \) are given as:

\[
\alpha_{11} = \left( \frac{\partial x}{\partial \eta} \right)^2 + \left( \frac{\partial y}{\partial \eta} \right)^2, \quad \alpha_{12} = \frac{\partial x}{\partial \xi} \frac{\partial x}{\partial \eta} + \frac{\partial y}{\partial \xi} \frac{\partial y}{\partial \eta}, \quad \alpha_{22} = \left( \frac{\partial \xi}{\partial \eta} \right)^2 + \left( \frac{\partial \eta}{\partial \eta} \right)^2
\]

E. H-Grid Topology

The H-grid topology is quite often employed in turbomachinery flow problem. The H topology is displayed in the Figure 4. Considering that figure, one can observe, the surface of the aerodynamic body is described here by two different grid lines, i.e., \( \eta = 0 \) and \( \eta = 1 \). On contrary to the C-grid, one family of grid lines (\( \eta = \text{const.} \) closely follows the streamlines (inlet located at \( \xi = 0 \), outlet at \( \xi = 1 \)).

At the first sight, there is no obvious coordinate cut. However, in turbomachinery the segments \( a-b \) and \( e-f \) are periodic (rotationally periodic in 3D) to each other. The same is true for the segments \( c-d \) and \( g-h \). This type of boundary condition is treated in Section 6.a. Figure 6(b) show a non-orthogonal H-grid between turbine blades. [3, 10]
Fig 6.a: Cross section for structured grid generation of Blazek turbine blades

Fig 6.b: Cross section for unstructured grid generation of Blazek turbine blades

Fig 7.a: Cross section for structured grid generation of Naca 0012

Fig 7.b: Cross section for Unstructured grid generation of Naca 0012

III. RESULT AND DISCUSSION

The present work uses two approaches in preparing the mesh flow domains required by the governing equation of fluid motion solver. Figure 5.a and Figure 5.b are shown the mesh model for structured and unstructured grid of flow past the bump. Figure 6.a and Figure 6.b show the mesh flow domain for the case of flow past through blade turbine. While for the case of the flow past through airfoil NACA 0012, are shown in the Figure 7.a and Figure 7.b. Using such kinds mesh flow domain accompanied with their appropriate boundary condition, the governing equation of fluid motion is solved. Their result in term of Mach number and pressure distributions are presented.

A. Bump Channel

The boundary conditions for the Bump channel at entry and exit station applied to this flow problem are given as:
- Stagnation pressure: 100000 Pa
- Static pressure: 99448.5 Pa
- The velocity component in x direction 300.4 m/sec
- The velocity component in y direction 0.0 m/sec
- Static temperature T = 300 K

Three different exit pressure conditions are prescribed as follows.
- The exit pressure at the back pressure (a) P_b = 50000 Pa, (b) P_b = 60000 Pa and (c) P_b = 70000 Pa, respectively.

The comparison result structured and unstructured grid in term of Mach number distribution as shown in the Figure 8, while in term of pressure distribution as shown in the Figure 9. Their result show in a good agreement and both predict nearly the same position when the shock wave appears in the flow field.
B. NACA 0012 Airfoil

The boundary conditions at the free streams are given as:
- Angle of attack [deg] = 1.250
- Static pressure [Pa] = 1.E+5
- Static temperature [K] = 288.0

The assessments carry out over three different values of Mach number. The calculations proceed with Mach number $M = 0.6$, $M = 0.8$ and $M = 1.0$. Their comparison result of these two developed computer code for Mach number and pressure distribution over the airfoil surface as depicted in the Figure 10 and Figure 11 respectively.

C. Turbine Blade

The boundary condition at entry and exit station to the case of flow past through a blades turbine are given:
- Stagnation pressure: $100000 \text{ Pa}$
- Static pressure: $99448.5 \text{ Pa}$
- The velocity component in $x$ direction $300.4 \text{ m/sec}$
- The velocity component in $y$ direction $0.0 \text{ m/sec}$
- Static temperature $T = 300 \text{ K}$
- The incoming flow speed $300 \text{ m / sec}$ at standard sea level

Above entry condition accompanied with three different exit pressure $P_e$. They are namely:

(a) $P_e = 47830 \text{ Pa}$,
(b) $P_e = 52830 \text{ Pa}$ and
(c) $P_e = 57830 \text{ Pa}$.

Figure 12 and Figure 13 are shown their comparison result in term Mach number and pressure distribution along the turbine surface respectively. Considering these two figures, it is clear that both two computer codes produce nearly the same result.
CONCLUSION

Considering above result as presented in the previous sub chapter concludes that structured grid and unstructured grid will produce nearly the same result. The structured grid may difficult to be implemented to case of the flow with a complex flow domain. While the unstructured grid offer flexibility in creating a meshing flow domain for a complex flow domain. In addition that the flow solution does not strongly influence by the manner of meshing flow domain, the Euler solver had been used here can be considered as a robust Euler Solver. This Euler Solver can be used for solving a flow problem with a more complex flow domain such as flow past through multi component airfoils or flow past through rotor and stator of turbine.

REFERENCES