IOP Conf. Series: Earth and Environmental Science **19** (2014) 012001 doi:10.1088/1755-1315/19/1/012001

Comparison Between Roe's Scheme and Cell-centered scheme For Transonic Flow Pass Through a Turbine Blades Section

Abobaker Mohammed Alakashi[1, a],Bambang Basuno[2, b]
[1, 2] Faculty of Mechanical and Manufacturing Engineering, University Tun Hussein Onn Malaysia (UTHM), Parit Raja, Batu Pahat, 86400 Johor, Malaysia

[a] Abobaker.alakashi@gmail.com, [b]Bambangb@uthm.edu.my

Abstract. The finite volume method (FVM) is a discretization technique for partial differential equations, especially those that arise from physical conservation laws. FVM uses a volume integral formulation of the problem with a finite partitioning set of volumes to discretize the equations[1]. The present work employs the Cell- centred and Roe's scheme to investigate the inviscid compressible flow pass through a two dimensional blade on two types of blade shapes: Blazek and SE1050 blade models. The governing equation of fluid motion of the flow problem in hand is the Euler Equation. The behavior of this equation will depend on the local Mach number if the governing equation stated in the form as the governing equation of steady flow problem. If the local Mach number is less than one, the governing equation will behave as elliptic type of differential equation while if the Mach number is greater than one it will behave as hyperbolic type of differential equation. Elimination of the presence of those two types governing equation for the case of transonic flow problem can be achieved by representing the Euler equation in unsteady form. So the equation becomes hyperbolic with respect to time. There are various Finite Volume Methods that can used for solving hyperbolic type of equation, such as Cell-centered scheme [2], Roe Upwind Scheme [3] and TVD Scheme[1]. Those computer codes apply in the case of inviscid two dimensional compressible flow pass through blades of turbine. The present work focuses on two computer codes, first based on Cell Centre scheme and the second one based on Roe's scheme.

1. Introduction

In the absence of viscous effects, the governing equation of fluid motion pass through an arbitrary body can be represented by Euler equations. For two dimensional compressible flow problems, this governing equation of motion in the form of non linear differential system equation. As result no analytic solution can be found for such kind of flow problems and a numerical approach is required. The governing equation of fluid motion can be presented in integral form which allow one to apply spatial descretization through the use of finite volume approach. There are various Finite Volume Methods can used for solving hyperbolic type of equation, such as Cell-centered scheme [2], Roe Upwind Scheme [3] and TVD Scheme[1]. Basically those three type finite volume methods are able to resolve some of the difficulties are often faced by the other methods which may required a fine and structured grid such as in the finite difference based method [2].

As the method of solving the flow problem through the governing equation of fluid motion in integral form make the finite volume method has a large number of options for the 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[4]. In addition to this, through directly discretizing to the integral form of the conservation laws, one can ensure that the basic quantities mass, momentum and energy will also remain conserved at the discrete level [3]. The main advantage of the finite volume method is that the spatial discretisation is carried out directly in the physical space. Thus, there are no problems with any transformation between coordinate systems [1,2].

2. Governing Equation of fluid motion in The integral conservation laws

As in usually adopted in flow analysis , let the flow variables are denoted by p, ρ, u, v, E and H which they are representing the pressure, density, Cartesian velocity components, total energy and total enthalpy respectively. For a perfect gas one has .

$$E = \frac{p}{(\gamma-1)\rho} + \frac{1}{2}(u^2 + v^2) \quad \text{and} \quad H = E + \frac{p}{\rho} \tag{1}$$

Where γ is the ratio of specific heat. The Euler equations for two dimensional inviscid flows can be written in integral form for a region Ω with boundary $d\Omega$ as:

$$\frac{\partial}{\partial t}\iint_{\Omega} \vec{U}dxdy + \oint_{d\Omega} (\vec{F}dy - \vec{G}dx) = 0 \tag{2}$$

Where x and y are Cartesian coordinates and

$$\vec{U} = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{pmatrix}, \quad \vec{F} = \begin{pmatrix} \rho \\ \rho u^2 \\ \rho uv \\ \rho uH \end{pmatrix}, \quad \vec{G}\begin{pmatrix} \rho \\ \rho uv \\ \rho v^2 + p \\ \rho vH \end{pmatrix}$$

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 as in the sketch, and a system of ordinary differential equations is obtained by applying equation 2 to each cell separately. The resulting equations can then be solved by several alternative time stepping schemes. In general if applied the conservation laws on the Cell ABCD as figure (1)

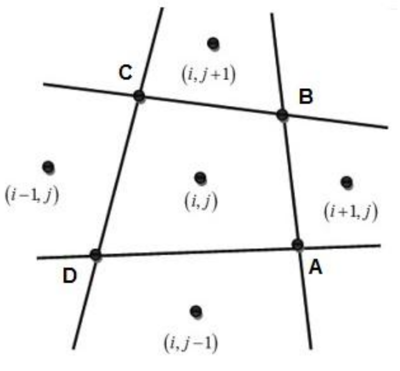

Figure 1. Cells in the finite-volume approach

One need to define the parameter geometry of the grid cell. The grid cell ABCD as shown in above figure has a side AB surface vector $\bar{S}_{AB}$ and its normal direction $\hat{j}_{AB}$ defined as :

$$\bar{S}_{AB} = \Delta y_{AB}\hat{i} - \Delta x_{AB}\hat{j} \quad \text{and} \quad \hat{j}_{AB} = (y_B - y_A)_i - (X_B - X_A)_j$$

While the area of grid cell ABCD [1,2]. Can be obtained as:

$$\Omega_{ABCD} = \frac{1}{2}\left|\vec{X}_{AC} - \vec{X}_{BD}\right| \;; \qquad \text{Where} \quad \vec{X}_{AB} = \vec{X}_B - \vec{X}_A$$

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

$$\Omega_{ABCD} = \frac{1}{2}[(X_C - X_A)(y_D - y_B) - (X_D - X_B)(y_C - y_A)] \tag{3}$$

In manner to determine the flux vector can be done in various approaches. The flux F crossing the side surface AB denoted as $\vec{F}_{AB}$ can be done with one of following approach.

- Average of fluxes $\qquad F_{AB} = F\left(\frac{U_{ij} - U_{i+1,j}}{2}\right)$
- Flux of the average flow variable $\qquad F_{AB} = F(\frac{U_{ij} + U_{i+1,j}}{2})$
- Average of fluxes in A&B $\qquad F_{AB} = \frac{1}{2}(F_A + F_B)$

In the last approach, approach c, the flux F_A determine firstly by defining that the flow variable U at A determine as :

$$U_A = \frac{1}{4}\left(U_{ij} + U_{i+1,j} + U_{i+1,j-1} + U_{i,j+1}\right) \tag{4.a}$$

Then the average the fluxes F_A becomes

$$F_A = \frac{1}{4}\left(F_{ij} + F_{i+1,j} + F_{i+1,j-1} + F_{i,j+1}\right) \tag{4.b}$$

With the ingredient in manner how to define the flux vector F as well as G as above, the numerical approach for solving the flow problem by using a finite volume method becomes solving the problem of ordinary differential equation with respect to time in the form of equation as given below;

$$\frac{\partial}{\partial t}U_{ij}\Delta X\Delta y + \left(F_{i+\frac{1}{2}j} - F_{i-\frac{1}{2}}\right)\Delta y + \left(g_{i,j+\frac{1}{2}} - g_{i,j\frac{1}{2}}\right)\Delta X = Q_{ij}\Delta X\Delta y \tag{5}$$

In part of the Roe's Scheme approach, The finite-volume form of the Euler equations can be written as;

$$\frac{dU}{dt} = \frac{1}{A}\Sigma_{faces}(F\Delta y - G\Delta x) \tag{6}$$

Where *A* is the area of that cell Δx and Δy are the changes of *x* and y along face

Defining the face length and normal and tangential velocities as;

$$\Delta s = \sqrt{(\Delta x)^2 + (\Delta y)^2}, \quad u_n = \frac{(u\Delta y - v\Delta x)}{\Delta s} \quad \text{and} \quad u_t = \frac{(u\Delta x - v\Delta y)}{\Delta s}$$

The flux through a face may be written as:

$$(F\Delta y - G\Delta x) = \begin{pmatrix} \rho u_n \\ \rho u_n u + p\frac{\Delta y}{\Delta s} \\ \rho u_n v - p\frac{\Delta x}{\Delta s} \\ \rho u_n H \end{pmatrix}\Delta s \cong \emptyset\,\Delta s \tag{7}$$

The flux through a face is a function of the values at the face midpoint, given by the reconstruction in the cells to the " left" and "right" of the face. Using Roe's approximate Riemann solver, this flux function is;

$$\emptyset(U_L, U_R) = \frac{1}{2}[\emptyset(U_L) + \emptyset(U_R)] - \frac{1}{2}\sum_{k=1}^{4}|\hat{a}_k|^* \Delta V_k \hat{R}_k \tag{8}$$

$$\hat{a} = \begin{pmatrix} \hat{u}_n - \hat{c} \\ \hat{u}_n \\ \hat{u}_n \\ \hat{u}_n + \hat{c} \end{pmatrix} \quad ; \qquad \Delta V = \begin{pmatrix} \frac{\Delta p - \hat{\rho}\hat{c}\,\Delta u_n}{2\hat{c}^2} \\ \frac{\hat{\rho}\Delta u_t}{\hat{c}} \\ \Delta\rho - \frac{\Delta p}{\hat{c}} \\ \frac{\Delta p - \hat{\rho}\hat{c}\,\Delta u_n}{2\hat{c}^2} \end{pmatrix} \qquad \text{and}$$

$$\hat{R} = \begin{pmatrix} 1 & 0 & 1 & 1 \\ \hat{u} - \hat{c}\frac{\Delta y}{\Delta s} & \hat{c}\frac{\Delta x}{\Delta s} & \hat{u} & \hat{u} + \hat{c}\frac{\Delta y}{\Delta s} \\ \hat{v} + \hat{c}\frac{\Delta x}{\Delta s} & \hat{c}\frac{\Delta y}{\Delta s} & \hat{v} & \hat{v} - \hat{c}\frac{\Delta x}{\Delta s} \\ H - \hat{u}_N\hat{c} & \hat{u}_t\hat{c} & \frac{\hat{u}^2 + \hat{v}^2}{2} & H + \hat{u}_n\hat{c} \end{pmatrix}$$

Where;

$$\hat{\rho} = \sqrt{\rho_L\rho_R}\ \hat{u} = \frac{\sqrt{\rho_L}u_L + \sqrt{\rho_R}u_R}{\sqrt{\rho_L} + \sqrt{\rho_R}} \quad \hat{v} = \frac{\sqrt{\rho_L}v_L + \sqrt{\rho_R}v_R}{\sqrt{\rho_L} + \sqrt{\rho_R}} \quad \hat{H} = \frac{\sqrt{\rho_L}H_L + \sqrt{\rho_R}H_R}{\sqrt{\rho_L} + \sqrt{\rho_R}}$$

Where $\hat{c}, \hat{u}_n,$ and $\hat{u}_t$ are calculated directly from $\hat{\rho}, \hat{u}, \hat{v},$ and $\hat{H}$. The flux difference terms (the summation in Eq. (38)) provide the upwind character which stabilizes the scheme. To prevent expansion shocks, an entropy fix imposed [2.5]. A smoothed value, $|\hat{a}^{(k)}|^*$, is defined to replace $|\hat{a}^{(k)}|$

For the two acoustic waves (k = 1, k = 4). For those two waves,

$$|\hat{a}^{(k)}|^* = \begin{cases} |\hat{a}^{(k)}|, & |\hat{a}^{(k)}| \geq \frac{1}{2}\delta a^{(k)} \\ \frac{(\hat{a}^{(k)})^2}{\delta a^{(k)}} + \frac{1}{4}\delta a^{(k)}, & |\hat{a}^{(k)}| \leq \frac{1}{2}\delta a^{(k)} \end{cases} \tag{9}$$

$$\delta a^{(k)} = max\left(4\Delta a^{(k)}, 0\right), \qquad \Delta a^{(k)} = a_R^{(k)} - a_L^{(k)}$$

For each cell, the face fluxes, calculated as above, are summed to give the residual for the cell [2.6],

$$\text{Res(U)} = -\frac{1}{A}\sum_{\text{faces}} \emptyset \Delta s \tag{10}$$

3. Result and Discussion

The present work employs the Cell- centred and Roe's scheme to investigate the inviscid compressible flow pass through two dimensional blade on two type of blade shapes Blazek and SE1050 blades. The grid cells of the cases follow structured grid system The grid cells of the problem in hand as shown in Figure 2 and 3. Two computer codes of solver had been developed, the first one created base on Cell- centred scheme by using a structured grid while the second one based on Roe's

IOP Conf. Series: Earth and Environmental Science **19** (2014) 012001 doi:10.1088/1755-1315/19/1/012001

scheme by using a structured grid approach also . In the incoming flow speed is around 300 m / sec at standard sea level, the flow problem can be considered as problem of the transonic flow around a turbine blade cascade. The boundary condition applied to this flow problem is given as follows 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, The exit pressure Pe = 52830 Pa.

Figure 4,a,b..d. below describe the Comparison of Mach number distribution in domain of interest of cross section Blazek and SE1050 turbine blades by using computer code of Cell- centred and Roe's Schemes on lower and upper parts respectively.

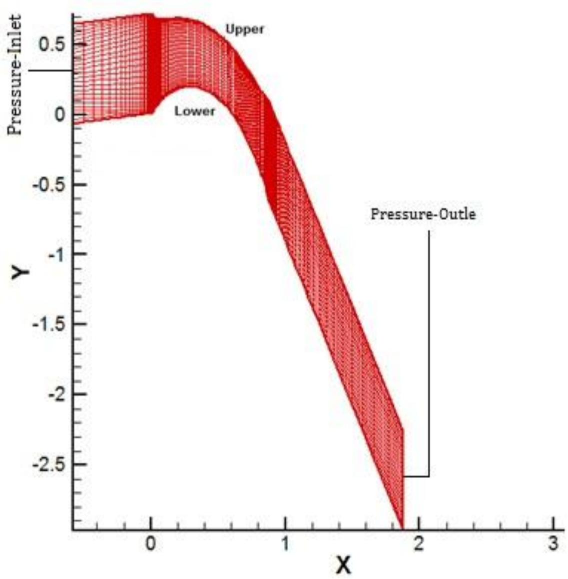

Figure 2. Cross section of Blazek turbine blades

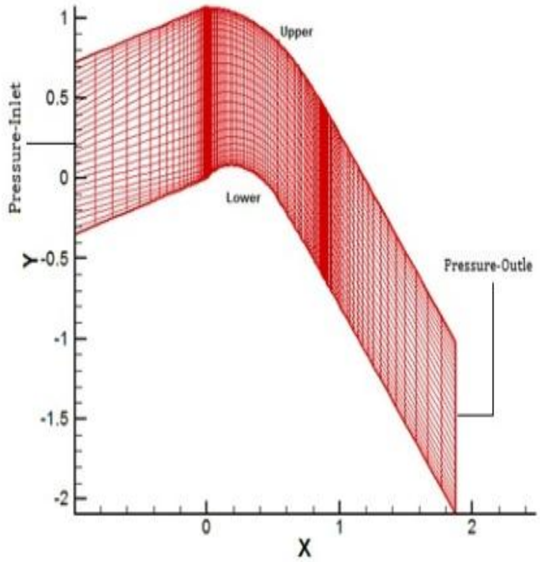

Figure 3. Cross section of SE1050 turbine blades

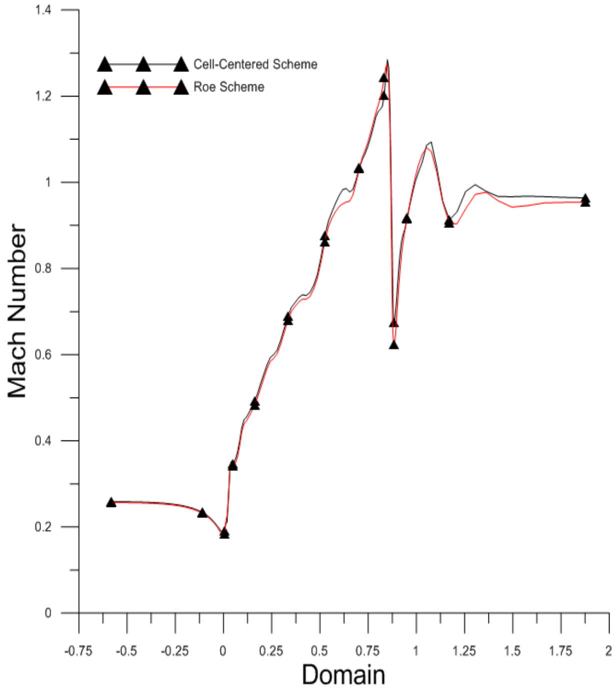

Figure 4.a. Mach number at lower part for Blazek blade.

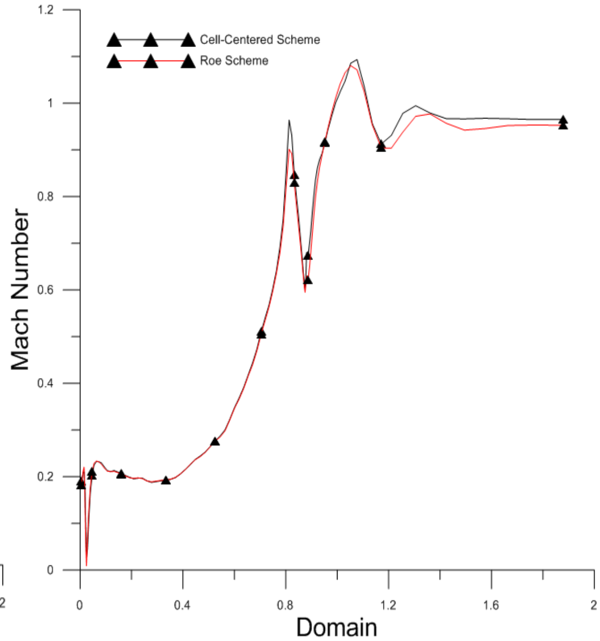

Figure 4.b. Mach number at upper part for Blazek blade.

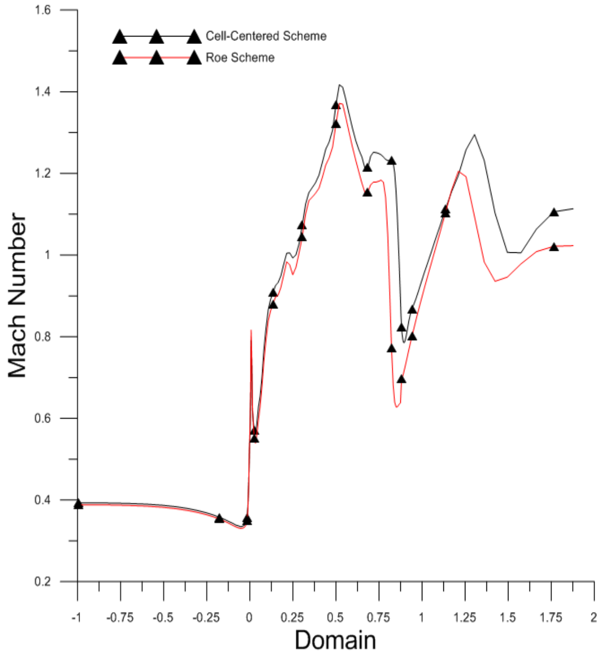

Figure 4.c. Mach number at lower part for SE1050 blade.

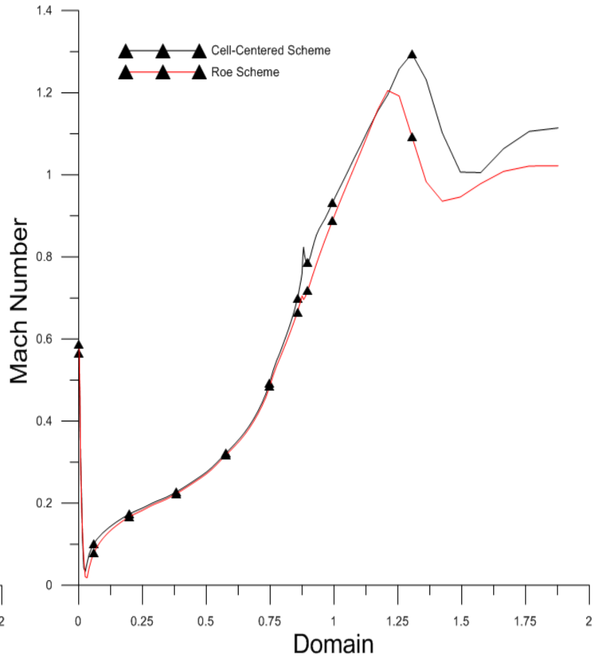

Figure 4.d. Mach number at Upper part for SE1050 blade.

Conclusion:

From the graph as shown above, the results in term of Mach number distribution over blade surfaces (upper and lower part) , for the case of Blazek blade, both numerical schemes give almost the same result. While for the case of SE1050 blade, a slightly different result occurred in the region near and after the blade trailing edge. However over the whole blade surface both numerical scheme provide their results in a good agreement each to other. From this point, it can be concluded the present Cell-centred Scheme and Roe's Scheme may represents the numerical scheme which can be extended in the use for further study on fluid behaviour over a complete turbine system which consist of "stator and rotor" sections. This combined system is on going in the present research.

References:

[1] K.. A . Hoffman and S.T .Chiang, Computational-Fluid-Dynamics Volume I, Engineering Education System.," USA, 2000.

[2] J. Blazek, Computational Fluid Dynamics Principles and Applications, 2nd Ed.," Switzerland, 2001.

[3] J. H. Ferziger, and M. Perić, Computational Methods for Fluid Dynamics.," Springer, 1996

[4] J. D. Anderson, Jr, Computational Fluid Dynamics, the basics with applications, McGraw-Hill, 1995.

[5] V. Hannemann, and others, Dissipation of upwind schemes at high wave numbers.," DLR - Institute of Aerodynamics and Flow Technology, Germany, 2012.

[6] G. Herve and Angelo Murrone, On the behavior of upwind schemes in the low Mach number limit: II. Godunov type schemes.," France, 2003