A Control Volume Finite Element Method for Potential Flow Analysis

Passakorn Vessakosol

Department of Mechanical Engineering, Faculty of Engineering
Prince of Songkla University, Hatyai, Songkla 90110, Thailand
E-mail: passakornv@hotmail.com

Manuscript received March 6, 2014
Revised March 25, 2014

ABSTRACT

A control volume finite element method for simulation of potential flow is presented. The formulation is developed for triangular mesh which has the linear interpolation of stream function over the element. The control volume finite element method (CVFEM) based on triangular mesh provides low computational cost as compared with the high order numerical method and high flexibility to find the stream function in complex shaped domain as compared with the analytical method and finite difference method. The accuracy of the present method is demonstratedby solving three test cases, i.e. flow through a straight channel, flow around a circular cylinder and flow through a backward facing ramp.

Keywords: *Triangular mesh; Control volume finite element method; potential flow; Numerical method.*

1. INTRODUCTION

The knowledge of velocity and pressure distributions through the passages of devices, pump, compressor, turbine or etc. is essential for fluid mechanics analysis.As we all know, the solutions from full Navier-Stokes equations almost represent the real fluid behavior. However, the calculation of the nonlinear differential equations as the full Navier-Stokes equations requires a lot of computer resources compared with the ideal fluid flow analysis such as potential flow. It should be preferred to start studying fluid dynamics with potential flow or ideal fluid flow. Here, the ideal fluid means the nonviscous, irrotational, incompressible fluid that flow steadily in two-dimensional region. An understanding of such ideal fluid flow usually provides the engineering students or the interest beginner of the computational fluid dynamics with a much broader approach to many real fluid flow situations.

In this paper, a control volume finite element method is developed to predict the streamline pattern in the potential flow. Generally, the control volume finite element method adapts the numerical integration on edges surrounding the control volume which normally used in the finite volume method and the interpolation by shape functions which employed in the finite element method. The governing equation of potential flow and the numerical formulation of control volume finite element method are described in section 2. The validations of proposed method are conducted in section 3 through comparisons of the solutions from present method with the analytical solution and the finite difference solution. Finally, the conclusion is given in section 4.

2. THEORY

2.1 Governing Equation

The potential flow problem predominantly relates with the irrotational flow. For irrotational flow, the velocity of a particle of fluid everywhere in flow field follows the condition of zero rotation:

$$\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0 \qquad (1)$$

where u and v are the velocity components of the flow field in x- and y-directions, respectively. They can be calculated from

$$u = \frac{\partial \psi}{\partial y} \ , \ v = -\frac{\partial \psi}{\partial x} \qquad (2)$$

where ψ is the stream function. The above definitions of velocity components satisfy the conservation of mass for two-dimensional incompressible steady flow as

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{3}$$

Substitution of Eq. (2) into Eq. (1) leads to differential equation in the form of Laplace's equation

$$\frac{\partial^2 \psi}{\partial x^2}+\frac{\partial^2 \psi}{\partial y^2}=0 \tag{4}$$

The boundary conditions for potential flow may be classified into two types. Firstly, the stream function along the wall can be imposed as

$$\psi=\psi_s(x,y) \tag{5}$$

Secondly, the derivatives of stream function along the inflow and outflow from the known u_s or v_s can be specified with the help of the known stream function, ψ_0, as

$$\psi=\psi_0+u_s\Delta y \text{ or } \psi=\psi_0-v_s\Delta x \tag{6}$$

where Δx and Δy are the distance between the boundary point and the reference point (x_0, y_0) in x and y coordinates, respectively.

After the stream function was calculated, the velocity components u and v at any point in the flow field can be taken from Eq. (2). Later, the pressure, p, can be computed from the Bernoulli's equation applied to inviscid flow on x-y plane as,

$$\frac{p}{\rho g}+\frac{V^2}{2g}=\text{constant} \tag{7}$$

where $V=\sqrt{u^2+v^2}$; ρ is the fluid density; g is the gravitational acceleration constant.

2.2 Control Volume Finite Element Method

The control volume finite element method is a numerical discretization procedure that based on the integration of the governing equation (4) over the control volume as shown in Fig. 1b:

$$\int_A\left(\frac{\partial^2 \psi}{\partial x^2}+\frac{\partial^2 \psi}{\partial y^2}\right)dA=0 \tag{8}$$

Gauss's theorem is employed in Eq. (8) to transform the area integral to the line integral enclosed the area. The result is

$$\int_s\left(\frac{\partial \psi}{\partial x}n_x+\frac{\partial \psi}{\partial y}n_y\right)ds=0 \tag{9}$$

where n_x and n_y are the direction cosines of the unit vector normal to the edge of area. The area is formed by edges from the neighborhood elements surrounding the node as shown in the Fig. 1. In control volume finite element method, the area can be obtained after the triangulation mesh generation which is a standard procedure of the finite element method as shown in Fig. 1(a). The sub-control volume edges are obtained from connecting the centroid of element with the midpoint of element edges. In Fig. 1(b), the sub-control volume edges from the neighborhood elements are integrated as the control volume edges after using the midpoint rule of numerical integration, the discretized equation for a nodal point is simplified to

$$\sum_{k=1}^{\mathrm{NE}}\left(\frac{\partial \psi}{\partial x}n_x+\frac{\partial \psi}{\partial y}n_y\right)_k=0 \tag{10}$$

where NE is the total number of edges surrounding the area.

The value of stream function in the triangular element is computed from

$$\psi=\sum_{i=1}^{3}N_i\psi_i \tag{11}$$

where N_i are the element interpolation functions and ψ_i are the nodal stream functions. The definition of N_i [1] at any coordinate position x, y within the element are

$$N_i=\frac{1}{2A_e}\left(a_i+b_ix+c_iy\right) \tag{12}$$

where A_e is the area of triangular element; a_i, b_i and c_i are the coefficients which can be computed from the coordinate positions x, y at three vertices of the element. Thus, the x- and y-derivatives are then determined from

$$\frac{\partial\psi}{\partial x}=\sum_{i=1}^{3}\frac{\partial N_i}{\partial x}\psi_i\ ,\ \frac{\partial\psi}{\partial y}=\sum_{i=1}^{3}\frac{\partial N_i}{\partial y}\psi_i \tag{13}$$

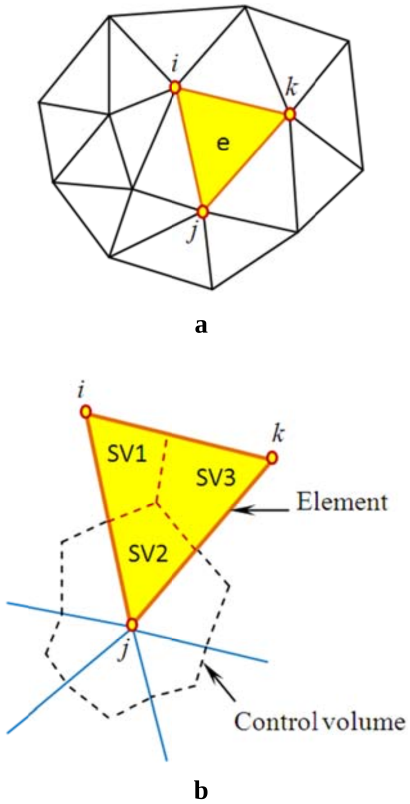

Fig. 1 *(a) Subdivision of calculation domain into triangular elements; (b) Sub-control volumes of three vertices in a triangular element and a complete area of node j.*

3. EXAMPLES

Three examples are chosen to validate the control volume finite element formulation and a computer program is especially developed for working with triangular mesh. The test cases for this section include flow through a straight channel, flow around a circular cylinder and flow through a backward facing ramp.

3.1 Flow Through a Straight Channel

The first case for validating the numerical result from the proposed method relates with the potential flow through a straight channel. The dimensions of computational domain and the boundary conditions are illustrated in Fig. 2. The uniform velocity U enters the flow domain from the left and exit at the right of domain. The numerical solution for this case can be compared with exact solution [2]. The numerical solution is obtained by using 100 triangular elements and 66 nodes. Fig. 3 shows the good agreement between exact solution and numerical solution.

3.2 Flow Around a Circular Cylinder

The flow domain for this case is rather more complex than that in the first case. Fig. 4 shows the triangular mesh and boundary conditions for a flow around a circular cylinder. The uniform velocity enters the flow domain from the left side. According to the symmetry of the problem, only upper left of the flow domain is used for modeling the flow field. The triangular mesh consists of 32 elements and 25 nodes. The stream functions at the boundary points are computed from the exact solution [2] as follows

$$\psi = Ur\sin\theta - \frac{\mu\sin\theta}{r},$$

$$r=\sqrt{\left(x-x_c\right)^2+\left(y-y_c\right)^2}\ , \tag{14}$$

$$\theta=\pi+\tan^{-1}\left(\frac{y-y_c}{x-x_c}\right)$$

where $\left(x_c, y_c\right)$ is the coordinates at the center of cylinder, U is the uniform velocity enters the flow domain and μ is the strength of doublet involved in the solution. The parameters used in the simulation are $U=1$ and $\mu=1$. The predicted streamlines from the CVFEM is very close to the streamlines from analytical method [2] as shown in Fig. 5.

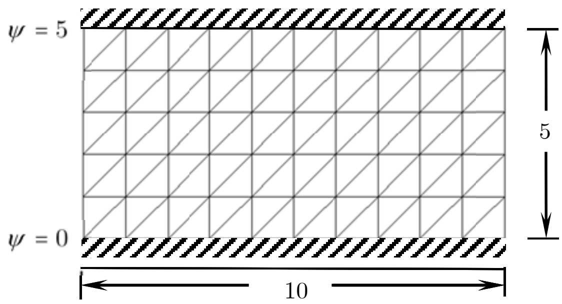

Fig. 2 *Triangular mesh and boundary conditions for flow through a straight channel.*

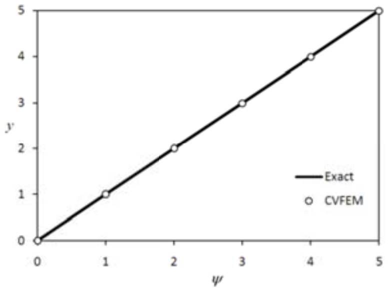

Fig. 3 *Comparison between the exact solution and the control volume finite element solution (CVFEM)*

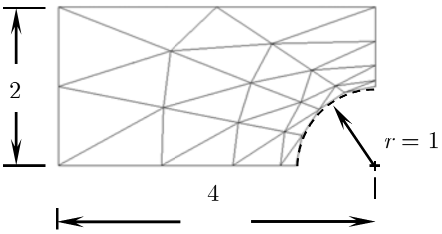

Fig. 4 *Model of flow around a circular cylinder*

3.3 Flow Through a Backward Facing Ramp

The last case is the flow through a backward facing ramp which has another solution from finite difference method [3] to compare with the solution of present method. The dimensions and shape of flow domain with the fluid properties used in the simulation are kept at the same values as those in Ref. [3]. Fig. 6 illustrates the geometry and boundary conditions used in the simulation. A uniform velocity enters the flow domain at the left boundary and a uniform velocity exits the flow domain at the right boundary. The rest boundary is impermeable wall which there is no flow across it. The model consists of 225 triangular elements and 136 nodes. Good agreement between the reference solution and the present solution is shown in Table 1.

Table 1 Comparison between the stream function solutions from finite difference method (FDM) and the control volume finite element method at selected points.

x	y	FDM[3]	CVFEM
1.2	1.6	6.58	6.58
1.6	0.8	1.77	1.77
1.8	1.4	5.93	5.93
2.4	1.2	5.50	5.50
2.4	1.6	7.71	7.71
2.6	0.4	1.77	1.77

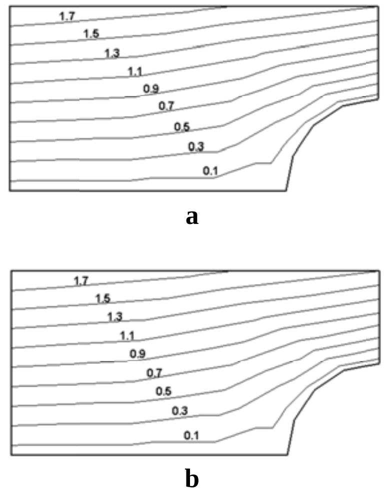

Fig. 5 *Streamlines around a circular cylinder from (a) analytical method (b) CVFEM*

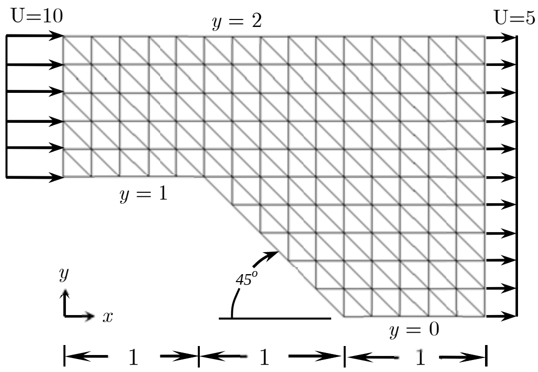

Fig. 6 *Mesh of flow through a backward facing ramp*

4. CONCLUSION

A Control volume finite element method is proposed in this paper for analyzing the potential flow problem. The introduction to the concept of the control volume finite element was given. Three test cases were used to validate the numerical solution from the present method include flow through a straight channel, flow around a circular cylinder and flow through a backward facing ramp. Comparison was made between the solution of present method and available solution from literature. In conclusion, the present method can give the solutions close to the reference solutions and provides the flexibility to model the complex flow domain as well as the finite element method with triangular mesh.

REFERENCES

[1] Reddy, J.N., *Finite Element Method*, McGraw-Hill, 1993.
[2] Streeter, V.L., and Wylie, B.E., *Fluid Mechanics*, McGraw-Hill, 1987.
[3] White, F.M., *Fluid Mechanics*, Fourth edition, McGraw-Hill, 1999.

Nomenclature

a	coefficient of shape function
A_e	element area
b	coefficient of shape function
c	coefficient of shape function
N	shape function
NE	total number of edges for a control volume
u	velocity component in x-direction
v	velocity component in y-direction
V	magnitude of velocity
x	x-coordinate
y	y-coordinate
ψ	stream function
ρ	fluid density

Subscripts

0	reference value
s	specified value

Passakorn Vessakosol received the Bachelor, Master and Doctoral degrees in mechanical engineering from King Mongkut's Institute of Technology Ladkrabang (KMITL), Bangkok, Thailand. He has been working as a Lecturer of Department of Mechanical Engineering, Prince of Songkla University (PSU), Hatyai, Thailand since 2011. His current interest is the development of numerical methods for the nonlinear engineering problems in Solid Mechanics, Heat Transfer, Fluid Mechanics and Multiphysics.