Adaptive Finite Element Technique for Potential Flow Analysis

Sedthawatt Sucharitpwatskul
Graduate Student

Pramote Dechaumphai
Professor

Mechanical Engineering Department, Chulalongkorn University
Bangkok 10330, Thailand.

Abstract

A finite element method for analysis of potential flow is presented. The finite element formulation and the computational procedure are described. The corresponding computer program that can be executed on standard personal computers has been developed. The method is combined with an adaptive meshing technique to increase the solution accuracy, and at the same time, to minimize the computational time and computer memory required. The finite element equations derived and the computer program developed are then evaluated by examples of flow in a square tube and flow past a cylinder. The efficiency of the combined finite element method and the adaptive meshing technique is demonstrated by the example of flow in a reduced cross-sectional channel.

1. Introduction

The finite element method is one of the numerical techniques that has received popularity due to its capability for solving complex structural problems. Applications of the method have been extended to a number of other fields, such as heat transfer, electromagnetics, and fluid flows. The potential flow is probably the simplest fluid flow that could be analyzed effectively by the finite element method. However, large amount of unknowns and computer memory may be required for flow past complex geometry. In addition, accurate finite element solution also requires small elements clustered in the critical flow regions. One way to minimize the total number of unknowns, the amount of computer time and the data storage used, is to employ an adaptive meshing technique. The technique places small elements in the regions of larger change in the solution gradients to increase solution accuracy, and at the same time, uses larger elements in the other regions to reduce the computational time and computer memory.

The paper starts by explaining the finite element formulation and the corresponding solution procedure that lead to the development of a computer program. The basic idea behind the adaptive meshing technique is then described. Finally, the finite element equations derived and the computer program developed are then evaluated by the examples of flow in a square tube and flow past a cylinder. The efficiency of the combined finite element method and the adaptive remeshing technique is demonstrated by example of flow in a reduced cross-sectional channel.

2. Theoretical Formulation and Solution Procedure

2.1 Governing Differential Equation

The conservation of mass for the two-dimensional steady-state inviscid incompressible flow [1] is,

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

where u and v are the velocity components in the x and y directions, respectively. These velocity components can be defined in form of the stream function, ψ, as

$$u=\frac{\partial \psi}{\partial y} \text{ and } v=-\frac{\partial \psi}{\partial x} \tag{2}$$

such that the conservation of mass is satisfied. For inviscid irrotational flow,

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

Substituting Eq.(2) into Eq.(3) leads to the differential equation for the potential flow,

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

The above differential equation is to be solved together with appropriate boundary conditions. Boundary conditions for potential flow may consist of specifying the stream function,

$$\psi=\psi_1(x,y) \tag{5}$$

along the inflow and outflow boundary. Along the wall boundary, the flow velocity is in the tangential direction with the wall, i.e.,

$$\frac{\partial \psi}{\partial s} = 0 \qquad (6)$$

where s is the tangential direction along the wall.

After the stream function is computed, the velocity components u and v can be determined from Eq.(2). The pressure, P, can then be computed from the Bernoulli's equation [2] as,

$$P + \tfrac{1}{2} V^2 + \rho g h = \text{constant} \qquad (7)$$

where $V^2 = u^2 + v^2$; ρ is the fluid density; g is the gravitational acceleration constant; and h is the fluid head.

2.2 Finite element formulation

The three-node triangular finite element is used in this paper to derive the finite element equations. The element assumes linear interpolations for the stream function as,

$$\psi(x, y) = N_i \, \psi_i \qquad (8)$$

where i = 1,2,3; N_i are the element interpolation functions and ψ_i are the nodal stream functions.

To derive the finite element equations, the method of weighted residuals [3,4] is applied to the governing differential equation, Eq. (4), i.e.,

$$\int_A N_i \left(\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2}\right) dA = 0 \qquad (9)$$

where A is the element area. Gauss's theorem is then applied to Eq. (9) to generate the boundary integral terms. This leads to the finite equations in the form,

$$\int_A \left(\left\{\frac{\partial N}{\partial x}\right\} \left\lfloor \frac{\partial N}{\partial x} \right\rfloor + \left\{\frac{\partial N}{\partial y}\right\} \left\lfloor \frac{\partial N}{\partial y} \right\rfloor \right) dA \, \{\psi\} = \int_S \{N\}(-vl + um) \, ds \qquad (10)$$

where l and m are the direction cosines of the unit vector normal to the element boundary edge, s.

2.3 Computational Procedure

The finite element equations, Eq.(10), are linear and can be solved by standard routine after assembling the equations from all elements and applying appropriate boundary conditions. A corresponding finite element program that can be executed on standard personal computers has been developed. The main objective in the development of this computer program is such that it follows the formulation derived and is easy to understand. The program has been verified by solving a number of examples that have exact solutions before applying to solve more complex flow problems.

2.4 Adaptive meshing technique

The idea behind the adaptive meshing technique presented herein is to construct a new mesh based on the solution obtained from the previous mesh. The new mesh will consist of small elements in the regions with large change in solution gradients and large elements in the other regions where the change in solution gradients is small. As an example of a flow past a cylinder, small elements are needed near the cylinder surface to capture detailed flow field, whereas larger elements can be used in the region far away from the cylinder because the flow behavior is almost uniform. To determine proper element sizes at different locations in the flow field, the solid-mechanics concept of determining the principal stresses from a given state of stresses at a point is employed. Since small elements are needed in the regions of complex flow behavior, thus the velocity distribution can be used as an indicator in the determination of proper element sizes.

To determine proper element sizes, the second derivatives of the flow velocity with respect to the global coordinates x and y are first computed,

$$\begin{bmatrix} \dfrac{\partial^2 V}{\partial x^2} & \dfrac{\partial^2 V}{\partial x \partial y} \\ \dfrac{\partial^2 V}{\partial x \partial y} & \dfrac{\partial^2 V}{\partial y^2} \end{bmatrix} \qquad (12)$$

where V is the magnitude of the two velocity components u and v,

$$V = \sqrt{u^2 + v^2} \qquad (13)$$

The principal quantities in the principal directions X and Y where the cross derivatives vanish, are then determined,

$$\begin{bmatrix} \dfrac{\partial^2 V}{\partial X^2} & 0 \\ 0 & \dfrac{\partial^2 V}{\partial Y^2} \end{bmatrix} \qquad (14)$$

The magnitude of the larger principal quantity is then selected,

$$\lambda_{max} = \max\left(\left|\frac{\partial^2 V}{\partial X^2}\right|, \left|\frac{\partial^2 V}{\partial Y^2}\right| \right) \qquad (15)$$

This value is used to compute proper element size h at that locations from the conditions,

$$h^2 \lambda = \text{constant} = h_{min}^2 \lambda_{max} \qquad (16)$$

where h_{min} is the specified minimum element size, and λ_{max} is the maximum principal quantity for the entire model.

Based on the condition shown in Eq.(16), proper element sizes are generated according to the given minimum element size h_{min}. Specifying too small h_{min} may result in a model with an excessive number of elements. On the other hand, specifying too large h_{min} may result in an inadequate solution accuracy or excessive analysis and remeshing cycles. These factors must be considered prior to generate a new mesh.

3.4 Examples

Two examples, which are flow in a square tube and flow past a cylinder, are used to validate the finite element formulation derived and the computer program developed. The capability of the finite element formulation combining with the adaptive meshing technique is demonstrated by the last example of flow in a reduced cross-sectional channel.

3.1 Flow in a Square Tube

To validate the finite element equations derived, an example of potential flow in a square tube is used as shown in Fig. 1. The dimensions of the computational domain with the flow boundary condition are shown in the figure. The problem has exact solution [5] that can be used to compare with the finite element solution. The figure also shows the uniform finite element mesh with 400 triangular elements and 231 nodes. The predicted finite element solution in form of the stream function is shown in Fig. 2. The figure shows the finite element method can provide exact solution for the stream function distribution along the y-direction at any x-location.

3.2 Flow Past a Cylinder

The finite element formulation and the developed computer program have been used to analyze potential flow past more complex geometry. Figure 3 shows the computational domain and the boundary conditions of a flow past a cylinder. The flow with uniform velocity enters the flow domain from the left boundary. Due to symmetry of the solution, only the upper or the lower half of the flow domain is used in the analysis. The lower half in Fig. 3 also shows a relatively uniform finite element mesh used in the analysis with 1200 triangular elements and 656 nodes.

The finite element analysis was then performed and typical solutions are shown in Fig. 4. The upper half of the figure shows the predicted streamlines whereas the flow velocity contours are shown in the lower half of the figure. The finite element solution is also used to compute the distribution of the potential function along the cylinder surface. Figure 5 shows good agreement of the predicted finite element potential function and the exact solution [5].

3. Flow in a Reduced Cross-Sectional Channel

To demonstrate the capability of the adaptive meshing technique combining with the finite element method, the problem of a potential flow in a reduced cross-sectional channel is selected. A uniform velocity flow enters the left boundary of the channel as shown in Fig. 6. The velocity of the flow increases gradually as cross-sectional area of the channel decreases. Figure 6 also shows the finite element model with a relatively uniform mesh. The model consists of 1,113 triangular elements and 607 nodes. The predicted flow velocity vectors of the flow field are shown in Fig. 7. The figure shows a gradual increase of the flow velocity as expected, except at the upper corner of the lower surface. The change of the flow velocity in this region is quite large. It can be expected that the change of the flow velocity will be pronounced if the mesh in this region is refined.

The adaptive remeshing technique is applied to provide more accurate solution to this problem. The technique starts from using a relatively uniform mesh, such as shown in Fig. 6, to compute the corresponding finite element solution. The obtained finite element solution is then used to generate a new finite element mesh according to the algorithm explained in the adaptive meshing technique section [6,7]. The new mesh will consist of small elements in the regions of large change in the solution gradient. At the same time, larger elements are generated in the other regions where the change in the solution gradient is small. Thus, information of the flow field behavior is not needed prior to analyzing the problem.

Figure 8 shows the adaptive finite element mesh after performing the finite element analysis and adaptive remeshing three times. This third adaptive finite element mesh consists of only 266 triangular elements and 153 nodes. The total number of nodes has reduced to 25% of those by the uniformed mesh in Fig. 6. This is because larger elements are generated in the regions of relatively uniform flow field such as near the free stream region. At the same time, smaller elements are clustered in the regions of larger change in flow velocity such as near the upper corner of the lower surface. Figures 9(a) and 9(b) show details of the mesh and the predicted flow velocity vectors in this region, respectively.

4. Concluding Remark

An adaptive meshing technique is combined with the finite element method to improve the potential analysis solution accuracy. The finite element equations for the potential flow differential equation were first derived. The finite element formulation and computer program have been validated by a number of problems that have exact solutions prior to solving more complex flow problems. The adaptive meshing technique was incorporated to improve the flow analysis solution accuracy. The technique generates small elements automatically in the regions of large change in the solution gradient. in order to provide more accurate solution accuracy. Larger elements are generated in the other regions where the solution gradients are small in order to reduce the total number of unknowns and hence the computational time. The examples shown in this paper demonstrate the capability and the effectiveness of the combined adaptive meshing technique and the finite element method for the potential flow analysis.

5. Acknowledgement

The authors are pleased to acknowledge the Royal Thai Navy for supporting this research work.

References

[1] Anderson, J.D., Jr., Computational Fluid Dynamics, McGraw-Hill, New York, 1995.

[2] Vries, G. de and Norrie, D. H., "The Applications of the Finite Element Technique to Potential Flow Problems", Trans. ASME, Series E: Journal of Applied Mechanics, Vol. 38, 1971, pp. 798-802.

[3] Dechaumphai, P., Finite Element Method in Engineering, Second Ed., Chulalongkorn University Press, Bangkok, 1999.

[4] Zienkiewicz, O. C. and Taylor, R. L., The Finite Element Method, Fourth Ed., McGraw-Hill international, New York, 1991.

[5] Streeter, V. L., and Wylie, B. E., Fluid Mechanics, McGraw-Hill international, Singapore, 1987.

[6] Dechaumphai, P. "Adaptive Finite Element Technique for Heat Transfer Problems", Energy, Heat and Mass Transfer, Vol. 17, 1995, pp. 87-94.

[7] Huebner, K. H. and Thornton, E. A., The Finite Element Method for Engineers, Third Ed., John Wiley & Sons, New York, 1995.

[8] Mohr, G. A., Finite Element for Solids, Fluids, and Optimization, Oxford University Press, Oxford, 1992.

[9] Ninomiya, H. and Onishi, K., Flow Analysis Using a PC, CRC Press, London, 1991.

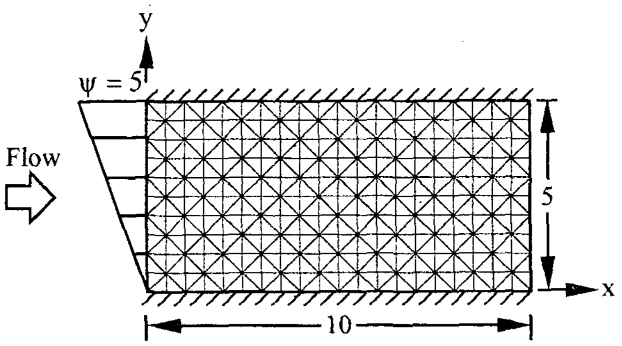


Fig. 1 – Finite element model and boundary conditions for flow in a square tube.

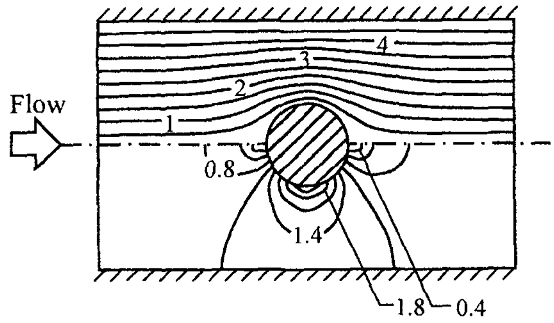


Fig. 4 – Predicted stream function and velocity contour for flow past a cylinder.

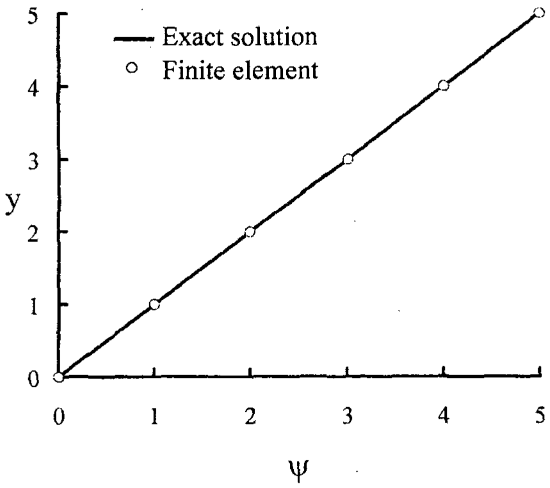


Fig. 2 – Comparative stream function between the exact solution and the finite element solution.

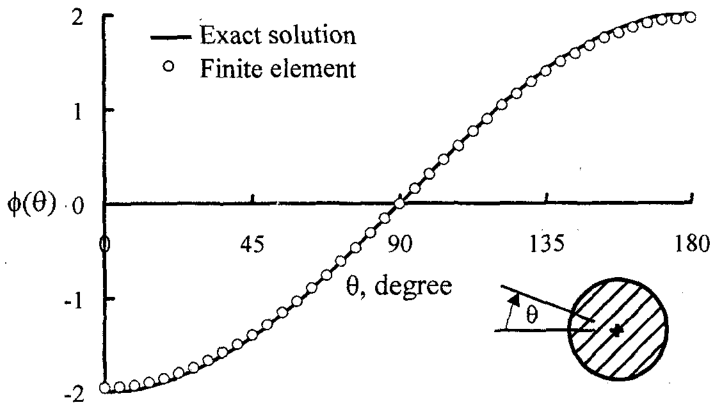


Fig. 5 – Comparative potential function between the exact solution and the finite element solution.

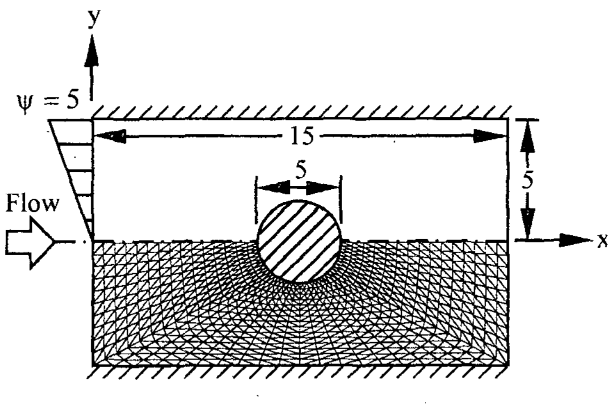


Fig. 3 – Finite element method and boundary conditions for flow past a cylinder.

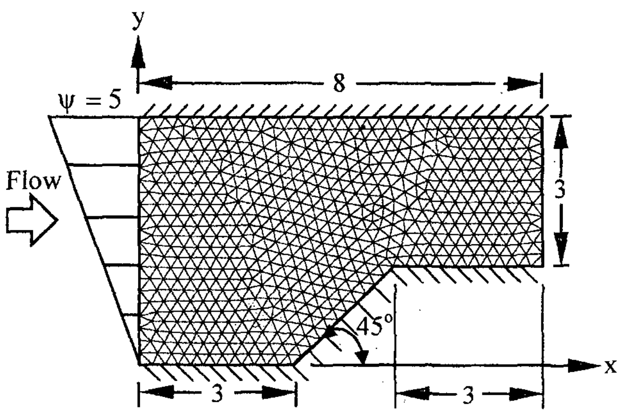


Fig. 6 – Flow in a reduced cross-sectional channel.

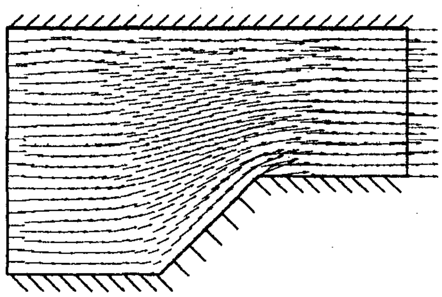


Fig. 7 – Predicted flow velocity distribution in a reduced cross-sectional channel.

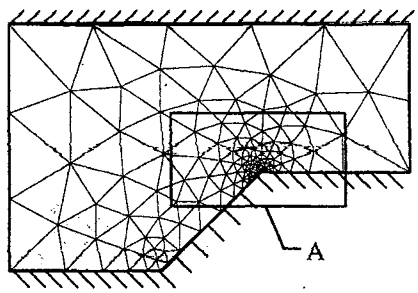
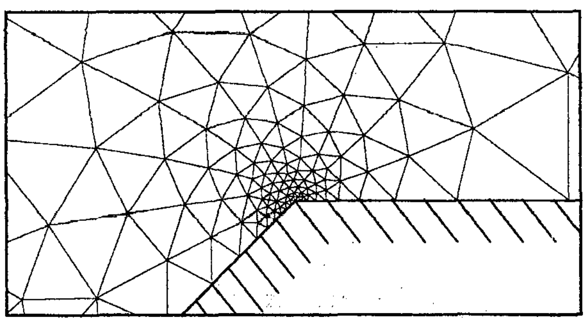
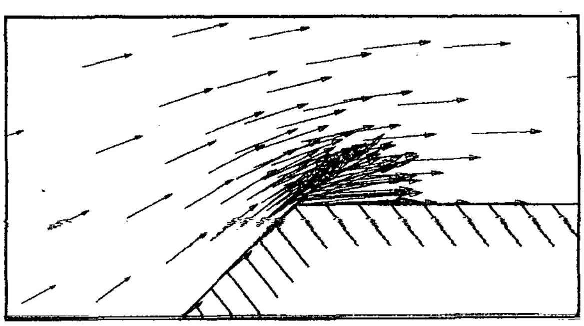


Fig. 8 – Adaptive finite element mesh for flow in a reduced cross-sectional channel.



(a) Detailed mesh of insert A in Fig. 8.



(b) Detailed velocity vectors of insert A in Fig. 8.

Fig. 9 – Detailed mesh and velocity vectors near the upper corner of the lower surface for flow in a reduced cross-sectional channel.