Finite Element Methods for Incompressible Viscous Flow*

Genki YAGAWA** and Yuzuru EGUCHI**

The present paper reviews the current research activities of the finite element methods for the incompressible viscous fluid flow. The status of the finite element flow analysis is discussed in comparison with the finite difference methods through the surveillance of the latest literature. The focus is placed upon the treatments of the incompressibility constraint and the non-linear convection term, both of which pose inherent difficulties in applying the finite element methods to the flow problems. Some fresh computational concepts which are intended to improve the efficiency and robustness are also scrutinized.

Key Words : Fluid Mechanics, Finite Element Method, Incompressible Viscous Flow, Computational Efficiency, Computational Robustness, Current Topics.

1. Introduction

The advent of sophisticated electronic computers has made it possible to predict the flow behavior based upon suitable mathematical description and numerical implementation. Such study of numerical investigations of fluid mechanics is known named as Computational Fluid Dynamics (CFD). Currently, CFD is being applied to a wide area of engineering fields, as tabulated in Table 1. The most vigorous activity of the CFD application is found in the field of aeronautical engineering, in the design of new flying vehicles such as STOL (short-takeoff-landing aircraft), space shuttles and hypersonic jetplanes. Although wind tunnels have traditionally been used to test the function of newly designed aircrafts, the investment for the construction of the advanced wind tunnel is very large, in the order of half a billion dollars.

Moreover, the experimental duration and running costs such as electricity and man-power are too excessive to design the unestablished aircraft. On the other hand, the price of the most advanced supercomputer is around ten million dollars, and it costs us much less time and money to do the same experiments numerically. The above situation, which is true of other engineering applications, explains the necessity of CFD research.

As is well-known, the computational methods for fluid flow are roughly categorized into two typical numerical schemes: the Finite Difference Method (FDM) and the Finite Element Method (FEM). The essential difference of both is the way of spatial discretization. In FDM, the differential operators of partial differential equations (PDE's) are directly replaced by the finite-differencing, and the solution is obtained at each grid. The scheme for the incompressible fluid flow is described well in the "bible" written by Roache⁽¹⁾. On the other hand, in FEM, the weak forms of PDE's are discretized in an indirect manner, using a basis function for solution expansion. Conse-

* Received 31st March, 1987.

** Department of Nuclear Engineering, University of Tokyo, 7-3-1 Hongo, Bunkyo-ku, Tokyo, 113, Japan

Table 1 Engineering fields and applications of the CFD

Engineering fields	Applications
Aeronautics	Space shuttle, Hypersonic plane, STOL designs
Biomechanics	Artificial heart, Blood vein prostheses
Biotechnology	Beer production, Bioreactor design
Civil engng.	Tsunami wave, Skyscraper, Harbour designs
Electronics	Semiconductor production, IC cooling
Mechanics	Turbojet design, Car engine design
Meteorology	Weather prediction, Typhoon course prediction
Nuclear engng.	Thermal hydraulics, MHD analyses

Table 2 Number of contributions presented at 4 th International Conference on Numerical Methods in Laminar and Turbulent flow (1985).

Numerical Meth.	FDM and FVM		FEM	
	2-D	3-D	2-D	3-D
Laminar Flow	40	10	30	8
Turbulent Flow	13	5	7	2

FDM : Finite Difference Method

FVM : Finite Volume Method

FEM : Finite Element Method

quently, one can seek the FEM solution in larger solution spaces where weaker differentiability than that of the original PDE is required, and the FEM solution can be found over the whole computational domain. The fundamental FEM concepts for flow analysis are given in some standard textbooks (e.g., see (2)). The advantageous feature of FDM is the ability to solve large problems with small computational costs, while FEM has the flexibility for approximation of complicated geometry. The question 'Which excels?' would not make any sense, since both are complementary to each other in the above sense. More detailed comparison of FDM and FEM is discussed by Shih and Chen⁽³⁾.

An up-to-date sense of the CFD research activity can be obtained, for example, by viewing the papers presented in the Fourth International Conference on Numerical Methods in Laminar and Turbulent Flow held at Swansea, U.K. in 1985⁽⁴⁾: The number of papers in each category is summarized in Table 2, according to the computational method, the number of dimension and the flow type. Among 155 papers presented there, 68 were based on the FDM and 47 the FEM. The reason for the prevalence of FDM may be because configurations appearing in fluid mechanics are generally not so complex as those found in structural mechanics, which enjoys the extensive use of the FEM. It is however, noted that the number of FEM studies are increasing steadily. The number of FEM turbulent flow analyses is also increasing, including

Fig. 1 Surface pressure computed by the FEM compressible flow analysis around a jetplane (after Jameson and Baker⁽⁵⁾)

sophisticated computations such as the Large Eddy Simulation (LES). Besides, one can see that the extension to three-dimensional problems is very active, although most of the three-dimensional computations are still limited to rather simple applications, such as flows in a pipe, duct or cavity.

Since 1969, the International Conference on Numerical Methods in Fluid Dynamics has been held a total of 10 times. Most of the presentations were concerned with the aeronautical computations of compressible fluids based upon the FDM. However, some applications of FEM to the compressible flow problems have been found in recent presentations. For example, Fig. 1 shows the FEM results of such compressible flow analyses around a jetplane computed by Jameson and Baker⁽⁵⁾. The appearance of the FEM in the compressible flow computations would vindicate the assumption that FDM has encountered difficulty in solving geometrically complex problems such as

three-dimensional flows around jetplanes.

As seen from the presentations in the latest international conferences, the current tendency of CFD is symbolized by the diversity of computational schemes and the upgrading of engineering applications. Therefore, there seems to be much potential for the more intensive application of FEM to the computation of fluid flow.

Although FEM has been applied to the solutions of various practical flow problems, there remain some difficulties to be overcome. The difficulties are mainly attributed to the treatments of the incompressibility constraint and the nonlinear convection term.

The assumption of incompressibility allows us to simplify the equation system. From the computational viewpoint, however, the incompressibility constraint is rather hard to deal with, since the change of the flow field, if any, immediately influences the flow field in the whole domain through the incompressible medium. Computationally speaking, it is impossible to advance the time integration in an explicit manner because of the singularity of the matrix of the time-dependent term. In other words, the incompressibility constraint and the associated pressure gradient term must be temporally integrated via an implicit method, and the matrix inversion is inevitable in the time-integration procedure.

The governing equations of flow are strongly nonlinear, and thus, sufficiently fine spatial subdivisions are required for numerically stable and physically meaningful solutions. The high- Re flow problems, especially, are of such a case. These inherent difficulties demand extremely large computational resources of storage and CPU time to obtain satisfactory results. Actually, most practical computations so far have been restricted to the two-dimensional spatial domain and low- Re problems. In order to extend the application of FEM to more realistic problems, i.e. fully three-dimensional, and physically and geometrically complicated problems, it is essential to explore supremely economic and robust computational schemes.

The remainder of the present paper reviews the FEM studies which are proposed for the improvement of robustness and efficiency to solve the flow problems. In the following section, we give some comments on the fundamental FEM formulations for incompressible viscous flow, focusing on the treatment of the incompressibility constraint. Some techniques proposed for high- Re flow analysis are reviewed in Section 3, and the up-to-date ad-hoc FE schemes are reviewed in Section 4.

2. Fundamental FEM Formulations

The most primitive formulation for the incompressible Navier-Stokes (NS) equations is constructed by the mixed formulation⁽⁶⁾ where the velocity components and the pressure are used as variables. From a mathematical viewpoint, the mixed formulation falls into a family of the Lagrange multiplier method, where the pressure plays the role of the Lagrange multiplier for the incompressibility constraint. The coupled equation system of the incompressibility equation and the NS equations can be solved simultaneously⁽⁷⁾ by such a direct solver as the LU decomposition or the Gauss elimination. Special care should be taken to apply an iterative solver to this linear system, since the diagonal zero entries and indefiniteness of the matrix exclude the utilization of familiar iterative methods such as SOR and the conjugate gradient method. Although Gresho, Lee and Sani⁽⁸⁾ mentioned that "there is currently no iterative solution method which can be guaranteed to work on the discretized, primitive-variable incompressible NS equations", the conjugate residual method is found to be effective to solve the linear system, as demonstrated by Robichaud and Tanguy⁽⁹⁾ and Eguchi, Yagawa and Fuchs⁽¹⁰⁾.

Otherwise, the NS equations and the incompressibility equation are solved separately in time integration process^{(11)~(13)}, based on the idea of the MAC algorithm⁽¹⁴⁾. In this solution scheme the incompressibility constraint can be satisfied by solving a Poisson equation with respect to pressure. The storage requirement can be reduced drastically since the NS equations can be temporally advanced explicitly with the mass matrix lumped. Moreover, the usage of the iterative method for the solution of the Poisson equation allows the minimum requirement of storage capacity. The disadvantageous feature of this algorithm is that the mass lumping could reduce the accuracy of the computed results, as reported by Gresho et al.⁽¹⁵⁾

The penalty method is a variation of the mixed formulation⁽¹⁶⁾⁽¹⁷⁾. The penalty method has been favorably used by the FE community in last decade to solve incompressible flow problems. When the penalty method was introduced into the FE flow analysis, the novelty and simplicity attracted the attention of the FE practitioners. Over the course of study, however, it has been found that the penalty method is inefficient in terms of storage. Although the pressure can be eliminated from the equation system, the resulting equations of velocity components couple with each other. Moreover, the property of the FE matrix is nearly ill-conditioned because of the large penalty

number and the singularity of the penalty matrix. Such ill properties of the matrix exclude the application of the iterative solvers. Consequently, one must solve rather bulky matrix equations only with a direct solver; then the practical applications are naturally limited to small-scale and low- Re problems.

The augmented Lagrangian formulation⁽¹⁸⁾ is apparently the mixture of the mixed method and the penalty method mentioned above. The Uzawa algorithm⁽¹⁹⁾ is often utilized to seek the solution which locates at the saddle point of the augmented Lagrangian functional. It is mathematically proved that the Uzawa algorithm converges for any positive penalty number, while the pure penalty method converges only for sufficiently large penalty numbers. Fortin and Fortin⁽²⁰⁾ have proposed the generalized Uzawa algorithm for the solution of the incompressible NS equations. The favorable features of the augmented Lagrangian with the Uzawa algorithm are that the incompressibility condition is strictly satisfied, and that the convergence of the Uzawa algorithm is guaranteed for any positive penalty number. The latter facet allows us to use the iterative solver with small penalty numbers, contrary to the pure penalty method. Eguchi and Fuchs⁽²¹⁾ proposed an iterative solution procedure using the biconjugate gradient solver⁽²²⁾ in the context of the generalized Uzawa algorithm. Such an iterative approach permits drastic reduction of the storage requirement.

Attention should be paid to the selection of the interpolation functions of the mixed formulation. The use of the same interpolation for the velocity and for the pressure leads to disappointing results⁽²³⁾. Usually, the order of the interpolation function for the velocity is chosen lower than that of the pressure to obtain physically reasonable solutions. For example, the multi-linear interpolation function for the velocity and piecewise-constant function for the pressure are commonly used in the mixed formulation. However, this element occasionally yields spatial oscillations under particular boundary conditions. Mathematically speaking, the interpolation functions of the velocity and the pressure should satisfy the Ladyzenskaya-Babuska-Brezzi (L-B-B) condition⁽²⁴⁾ for unique and stable solution. Recent papers have pointed out that this condition can be satisfied by adding internal velocity nodes to the conventional elements in case of discontinuous pressure⁽²⁵⁾, although the simplicity of the conventional lower order element compensates for the drawback in the practical applications. In the penalty formulation, the velocity and pressure should also be approximated by the interpolation functions which are consistent with the L-B-B condition. It should be noted here that the reduced integration

technique for the penalty term leads to an inconsistent imposition of the incompressibility constraint, since the incompressibility cannot be retained in the discretized system as explained by Engelman et al⁽²⁶⁾. The other basic formulation is based upon the use of somewhat artificial variables: streamfunction ψ and vorticity ω . A transport equation with respect to the vorticity and a Poisson equation of streamfunction are the basic equations to be solved. The use of the same order interpolation both for vorticity and for streamfunction, is permissible, contrary to the mixed formulation. The streamfunction-vorticity formulation seems to have difficulty in solving high- Re problems accompanied by sharp velocity gradients, since the governing equations contain the third-order derivatives of velocity, while the equations of the mixed formulation are of the second-order regarding velocity. The imposition of boundary conditions is not straightforward, due to the use of artificial variables. For example, if the domain is multi-connected such as the flow around obstacles, the value of the streamfunction on the obstacles cannot be known *a priori*, and a pre-computation is needed to impose the boundary condition of the streamfunction. Also, the extension of the streamfunction-vorticity formulation to the three-dimensional analyses is almost impossible, since the streamfunction is defined only in two-dimensional space. Considering these facts, the primitive variable formulation seems more attractive and natural.

3. Overview of FE Schemes for High- Re Flow

As the Reynolds number is increased, the Galerkin FEM spatial discretizations give rise to oscillations in the computed solutions at reasonable mesh size. A remedy to suppress such meaningless oscillations is the application of the upwind FE instead of the Galerkin method. The upwind discretization generally introduces artificial diffusion, which one must consider so as not to deteriorate the solution by false diffusion. For example, the upwind technique of one-point quadrature for the convective terms⁽²⁷⁾ introduces undesirable false cross-wind diffusion, as severely criticized by Leone and Gresho⁽²⁸⁾. The most reliable upwind technique can be implemented by applying the streamline-upwind/Petrov-Galerkin concept (SUPG), which was originally proposed by Hughes and Brooks^{(29)–(31)}. In the SUPG method, the weighting function is defined by an unsymmetric shape function leaning toward the upwind direction, while the weighting function of the Galerkin method is chosen so as to coincide with the basis functions in the solution expansion. The SUPG method introduces the artificial viscosity only in the streamline direction, which works not as a pollutant

against the solutions, but as a suppressant against the non-physical oscillations. However, the SUPG method sometimes yields the some spatial oscillations in the vicinity of sharp boundary layers. Mizukami and Hughes⁽³²⁾ proposed a new Petrov-Galerkin method for convection-dominated flow, which satisfies the discrete maximum principle and gives very accurate solutions.

If the Reynolds number Re is increased further, the NS equations exhibit strong nonlinearity, and the flow tends to be unstable, physically as well as numerically. The use of the upwind technique on coarse mesh would lose the physical meaning of the computed results, even if the numerical computation works stably. In order to solve such flow numerically, computational mesh must be fine enough to resolve the fine structure of small eddies. Therefore, large storage and CPU resources are demanded to accurately compute the high- Re problems. The key to successfully solving high- Re flow by FEM is to reduce the computational storage requirement, since FEM usually needs a large memory, even for rather small problems.

Kawahara and Hirano⁽³³⁾ applied the two-step explicit methods to the high- Re problems using an artificial compressibility method. The solutions seem reasonable in appearance, but this scheme would introduce too much artificial viscosity, which works as the damper against the nonlinear instability. This inference is supported by the fact that a feigned separation appears behind a cylinder in computing the flow around a cylinder without physical viscosity. The basic theory of fluid mechanics tells us that such an inviscid fluid flow must be potential flow without any separation. The non-physical separation can be explained by the artificial viscosity which is unnecessarily introduced in the time integration procedure.

Gresho et al.⁽³⁴⁾ proposed a cost-effective scheme to solve the convection-dominated flow. The scheme is based on the MAC-like algorithm combined with mass lumping, reduced quadrature, balancing tensor diffusivity and subcycling. These numerical strategies allow considerable storage reduction with a permissible loss of accuracy. The numerical example includes the two-dimensional cavity flow of Reynolds number 10 000⁽³⁵⁾ and three-dimensional natural convection of Rayleigh number 50 000⁽³⁶⁾. In a similar manner, Eguchi and Yagawa⁽³⁷⁾ solved the three-dimensional driven cavity problem of Reynolds numbers from 1 000 to 10 000 with $41 \times 41 \times 41$ velocity nodes.

Lasbleiz et al.⁽³⁸⁾ proposed an efficient computational scheme using the Ronat algorithm⁽³⁹⁾, which is a splitting-up technique based on a weak

form of the characteristics method. A Stokes-type problem in the Ronat algorithm is solved by the conjugate gradient Uzawa algorithm⁽⁴⁰⁾ to decouple the velocity and the pressure. As an industrial application, the flow in a nuclear reactor is solved with 20 108 velocity nodes and 12 936 tetrahedron elements.

The high- Re flow can be turbulent flow from a macroscopic viewpoint. The turbulent flow analysis is also one of the current key issues in FE society. The conventional turbulent models such as k - ϵ -like two-equation models are predominantly employed for turbulence modeling. The numerical algorithms used in FEM are more or less similar to those of the FDM in dealing with the turbulent model equations and the time-averaged incompressible NS equations⁽⁴¹⁾. For example, Okuda, Eguchi and Yagawa⁽⁴²⁾ have developed a turbulent flow analysis code based on the k - ϵ model to solve large-scale three-dimensional problems of rather complicated geometry. The most difficult point of the k - ϵ FE analysis would be the treatment of wall boundary conditions, since the universal velocity profile can no longer be presumed for the non-straight wall boundary generated by the FE mesh. The direct simulation of the incompressible NS equations⁽⁴³⁾ or the Large Eddy Simulation (LES)⁽⁴⁴⁾ could preclude such inconsistencies. However, the mesh spacing must be extremely small for the spatial resolution of small turbulent eddies, and thus, the computational speed and capacity of the currently available computers do not allow us to realize such tough computations for engineering applications. For the above-mentioned reasons, the application of the FEM to the turbulent flow analysis is thus far restricted to geometrically very simple problems. In order to extend the FE turbulent flow analysis to more practical problem, a supremely efficient NS solver and suitable turbulence modeling are desperately needed.

4. Current Topics

Some exotic approaches have been investigated actively. The present review highlights the adaptive mesh technique, spectral method, combination of FEM and FDM and multigrid method among others. In the following, we shall briefly go through the facets of these ad-hoc schemes.

The adaptive mesh technique⁽⁴⁵⁾ is a sort of mesh optimization to minimize the discretization error. This approach would be very helpful in solving troublesome problems such as shock wave, shear layer and phase change. Some applications are found in structural problems, but the FE flow analysis is not involved in the application, as yet, partially because the nonlinearity of the NS equations makes it difficult to estimate discretization error using mathematical

manipulations. Löhner et al.⁽⁴⁶⁾ developed a computer code for transonic and supersonic flow with mesh adaptation. Figure 2 (a) shows the automatically generated finite element mesh for the problem of a planar shock impinging on a complex structure. Figure 2 (b) shows the computational result of density which clearly illustrates the shock fronts interacting with the structure. As seen from this example, the mesh adaptation is indispensable for the prediction of the shock front location.

The spectral method⁽⁴⁷⁾ has emerged as a major tool in computational meteorology. The approximate solution is expanded by, for example, the pseudospectral Chebyshev function over the whole domain. For smooth flows, it saves us a great amount of computer memory and time, since a small number of nodes are sufficient to obtain the same accuracy

obtained by the other low order FEM's. For example, Ku et al.⁽⁴⁸⁾ solved the three-dimensional driven cavity flow of Reynolds number 1000 with a Chebyshev pseudospectral matrix method.

The FEM-FDM zonal approach is proposed by Nakahashi⁽⁴⁹⁾ to compute the multi-body flow field. The FD grids are placed around the bodies, and the FD grids are connected with FE meshes as shown in Fig. 3. This approach takes advantage of FDM's efficiency for computation and FEM's flexibility for geometry approximation. Since the computational algorithms of FEM and FDM are being assimilated with each other such mixture could be one of the ideal numerical schemes.

The multigrid method⁽⁵⁰⁾ was originally introduced for the solution of the elliptic equations in the FD context by Russian researchers, and the con-

(a) An example of adaptive finite element mesh for a shock problem (after Löhner et al.⁽⁴⁶⁾).

(b) Computed density distribution (after Löhner et al.⁽⁴⁶⁾).

Fig. 2

Fig. 3 FDM-FEM zonal grid for a turbine cascade flow ; (a) FDM-grid, (b) FEM-grid, (c) patched FDM-FEM grid (after Nakahashi⁽⁴⁹⁾).

Fig. 4 An example of a series of multi-grids; (a) coarse grid, (b) fine grid (after Perez et al.⁽⁵³⁾)

cept was completed by Brandt⁽⁵¹⁾. It is based on the idea that errors of an iterative solution on a fine grid can be effectively smoothed out on a coarse grid. The solution can be obtained through a rather small computational task by correcting the iterative solution on different coarse and fine grids several times. Practical aspects of FE implementation are given in reference⁽⁵²⁾. Perez et al.⁽⁵³⁾ propose the adaptive multi-grid FEM where the initial local coarse grid is refined only locally after *a posteriori* error estimations. Such series of adaptive multi-grids are shown in Figs. 4 (a) and (b).

5. Concluding Remarks

As seen in the above, the FEM for flow analysis has been actively investigated, and many advanced schemes are proposed for the improvement of efficiency and robustness in solving the flow problems. However, the FEM is not necessarily used as the practical tool for the design, mainly due to the expensive computational costs. Especially, high-*Re* flow and turbulent flow are too burdensome for FEM to compute reliably. It seems that there is no state of the art FE algorithm so far for solving such engineering problems, in spite of intensive research activities.

The future direction of FE research would be the construction of a supremely efficient and robust algorithm which is applicable to practical engineering problems. Furthermore, the extension to interdisciplinary problems will be studied more actively and widely, as currently applied to fluid-structure interaction, free surface flow, thermal convection, magneto-hydrodynamics, gasdynamics and non-Newtonian flow. What the authors hope is that fluid mechanics will enjoy the casual use of FEM in the near future as much as the structural mechanics does presently.

References

- (1) Roache, P. J., Computational Fluid Dynamics, Albuquerque, (1972), Hermosa Press.
- (2) Connor, J. J. and Brebbia, C. A., Finite Element Techniques for Fluid Flow, (1976) Butterworth.
- (3) Shih, T. M. and Chen, Y. N., Comparison of Finite Difference Method and Finite Element Method, Shih, T.M. ed., Numerical Properties and Methodologies in Heat Transfer, (1982), p. 33, Hemisphere Publ.
- (4) Taylor, C., Olson, M. D., Gresho, P. M. and Habashi, W. G. ed., Proc. 4th Int. Conf. on Numer. Meth. in Laminar and Turbulent Flow, Swansea, (1985), Pineridge Press.
- (5) Jameson, A. and Baker, T. J., Euler Calculations for a Complete Aircraft, In: Zhuang, F. G. and Zhu, Y. L. eds., Lecture Notes in Physics, Vol. 264 Berlin (1986), p. 334, Springer-Verlag.
- (6) Hood, P. and Taylor, C., Navier-Stokes Equations Using Mixed Interpolation, In: Int. Symp. on Finite Element Method in Flow Problems, Swansea, U. K., (1974).
- (7) Taylor, C. and Hood, P., A Numerical Solution of the Navier-Stokes Equations Using FEM Technique, Computers and Fluids, No. 1 (1973), p. 73.
- (8) Gresho, P. M., Lee, R. L. and Sani, R., On the Time-dependent Solution of the Incompressible Navier-Stokes Equations in Two and Three Dimensions, In: Taylor, C. and Morgan, K. ed., Recent Advances in Numerical Methods in Fluids, Vol. 1 (1980), p. 27 Pineridge Press.
- (9) Robichaud, M. and Tanguy, P., Incomplete Factorization In 3-D Incompressible Fluid Flow Problems, In: Taylor, C., Olson, M. D., Gresho, P. M. and Habashi, W. G. ed., Proc. 4th Int. Conf. Numer. Meth. Laminar and Turbulent Flow, (1985), p.1783, Pineridge Press.
- (10) Eguchi, Y., Yagawa, G. and Fuchs, L., A Conjugate-Residual-FEM for Incompressible Viscous Flow Analysis', Comp. Mech, (to appear).

- (11) Donea, J., Giuliani, S., Laval, H. and Quartapelle, L., Finite Element Solution of The Unsteady Navier-Stokes Equations by a Fractional Step Method, *Comp. Meth. Appl. Mech. Eng.*, No. 30 (1982), p. 53.
- (12) Mizukami, A. and Tsuchiya, M., A Finite Element Method for the Three-Dimensional Non-Steady Navier-Stokes Equations, *Int. J. Numer. Meth. Fluids*, No. 4 (1984), p. 349.
- (13) Bristeau, M. O., Glowinski, R., Mantel, B., Periaux, J. and Perrier, P., Numerical Methods for Incompressible and Compressible Navier-Stokes Problems, In: Gallagher, R. H., et al. eds., *Finite Elements in Fluids*, Vol. 6 (1985), p. 1, Chichester John Wiley and Sons.
- (14) Chorin, A. J., Numerical Solution of the Navier-Stokes Equations, *Math. Compt.*, No. 22 (1968), p. 745.
- (15) Gresho, P. M., Lee, R. L. and Sani, R., Advection-dominated Flows with Emphasis on the Consequence of Mass Lumping, In: Gallager, R. H., Zienkiewicz, O. C., Oden, J. T., Morandi Cecchi, M. and Taylor, C., ed., *Finite Elements in Fluids*, Vol. 3 (1978), p. 335, John Wiley & Sons.
- (16) Malkus, D. S. and Hughes, T. J. R., Mixed Finite Element Methods-Reduced and Selective Integration Technique: A Unification of Concept, *Compt. Meth. Appl. Mech. Eng.*, No. 15 (1978), p. 63.
- (17) Bercovier, M. and Engelman, M., A Finite element for the Numerical Solution of Viscous Incompressible Flows, *J. Compt. Phys.*, No. 30 (1979), p. 181.
- (18) Glowinski, R. and Le Tallec, P., Numerical Solution of Equilibrium Problems in Incompressible Finite Elasticity by Augmented Lagrangian Techniques, *SIAM J., Appl. Math.*, No. 42 (1982), p. 400.
- (19) Temam, R., *Navier-Stokes Equations*, (1977), North-Holland.
- (20) Fortin, Y. and Fortin, A., A Generalized Uzawa's Algorithm for the Solution of the Navier-Stokes Equations, *Commun. Appl. Numer. Meth.*, No. 1 (1985), p. 205.
- (21) Eguchi, Y. and Fuchs, L., Conjugate-Gradient Methods Applied to the Finite-Element Approximation of the Navier-Stokes Equations, In: Yagawa, G. and Atluri, S. N., ed., *Proc. Int. Conf. Comp. Mech.*, Tokyo, (1986), p. VII-109, Springer-Verlag.
- (22) Saad, Y., The Lanczos Biorthogonalization Algorithm and Other Oblique Projection Methods for Solving Large Unsymmetric Systems, *SIAM J. Numer. Anal.*, No. 19 (1982), p. 485.
- (23) San I, R. L., Gresho, P. M., Lee, R. L. and Griffiths, D. F., The Cause and Cure (?) of the Spurious Pressures Generated by Certain FEM Solutions of the Incompressible Navier-Stokes Equations: Pt. 1, *Int. J. Numer. Meth. in Fluids*, Vol. 1 (1981), p. 17.
- (24) Brezzi, F., On the Existence, Uniqueness and Approximation of Saddle-Point Problems Arising from Lagrangian Multiplier, *R. A. I. R. O.*, No. 8 (1974), p. 129.
- (25) Fortin, M. and Fortin, A., Experiments with Several Elements for Viscous Incompressible Flows, *Int. J. Numer. Meth. in Fluids*, Vol. 5 (1985), p. 911.
- (26) Engelman, M. S., Sani, R.L., Gresho, P. M. and Bercovier, M., Consistent vs. Reduced integration Penalty Methods for Incompressible Media Using Several Old and New Elements, *Int. J. Numer. Meth. in Fluids*, No. 2 (1982), p. 25.
- (27) Hughes, T. J. R., Lui, W.K. and Brooks, A., Finite Element Analysis of Incompressible Viscous Flows by the Penalty Function Formulation', *J. Compt. Phys.*, No. 30 (1979), p. 1.
- (28) Leone, J. M. Jr., and Gresho, P. M., Finite Element Simulation of Steady, Two-Dimensional, Viscous Incompressible Flow Over a Step, *J. Compt. Phys.*, No. 41 (1981), p. 167.
- (29) Hughes, T. J. R. and Brooks, A. N., A Theoretical Framework for Petrov-Galerkin Methods with Discontinuous Weighting Functions: Application to the Streamline Upwind Procedure, In: Gallagher, R. H., Norrie, D. H., Oden, J. T. and Zienkiewicz, O. C. ed., *Finite Elements in Fluids Vol. 4* (1982), p. 46, John Wiley & Sons.
- (30) Hughes, T. J. R. and Brooks, A., A Multidimensional Upwind Scheme with No Crosswind Diffusion, In: Hughes, T. J. R. ed., *Finite Element Methods for Convection Dominated Flows*, AMD, Vol. 34 (1982).
- (31) Brooks, A. and Hughes, T.J. R., Streamline Upwind/Petrov-Galerkin Formulations for Convection Dominated Flows with Particular Emphasis on the incompressible Navier-Stokes Equations, *Comp. Meths. Appl. Mech. Eng.*, No. 32 (1982), p. 199.
- (32) Mizukami, A. and Hughes, T. J. R., A Petrov-Galerkin Finite Element Method for Convection-Dominated Flows: An Accurate Upwinding Technique for Satisfying the Maximum Principle, *Comp. Meths. Appl. Mech. Eng.*, No. 50 (1985), p. 181.
- (33) Kawahara, M. and Hirano, H., A Finite Element Method for High Reynolds Number Viscous Fluid Flow Using Two Step Explicit Scheme, *Int. J. Numer. Meth. in Fluids*, No. 3 (1983), p. 137.
- (34) Gresho, P. M., Chan, S. T., Lee, R. L. and Upson, C. D., A modified Finite Element Method for Solving the Time-Dependent, Incompressible Navier-Stokes Equations. Pt. 1: Theory, *Int. J. Numer. Meths. in Fluids*, No. 4 (1984), p. 557.
- (35) Gresho, P. H., Chan, S. T., Lee, R. L. and Upson, C. D., A modified Finite Element Method for Solving the Time-Dependent, Incompressible Navier-Stokes Equations. Pt. 2: Applications, *Int. J. Numer. Meths. in Fluids*, No. 4 (1984), p. 619.
- (36) Upson, C. D., Gresho, P. M., Sani, R. L., Chan, R. T. and Lee, R. L., A Thermal Convection Simula-

- tion in Three Dimensions by a Modified Finite Element Method', In: Shih, T. M. ed., Numerical Properties and Methodologies in Heat Transf., (1982), p. 245, Hemisphere Publ.
- (37) Eguchi, Y. and Yagawa, G., A Large-Scale Three - Dimensional Finite Element Analysis of Incompressible Viscous Fluid Flow, *Comp. Mech.*, (to appear).
- (38) Lasbleiz, P., Chabard, J. P., Gregoire, J. P., Daubert, O., Goussebaile, J. and Hemmerich, P. H., Efficient Computation of 3 D Industrial Turbulent Flow by Finite Element Method', In: Proc. 6th Int. Symp. Finite Element Methods in Flow Problems, Antibes, France, (1986), p. 189.
- (39) Benque, J. P. and Ronat, J., Quelques Difficultes des Modeles Numeriques en Hydraulique, 5 th Int. Symp. on Computing Methods in Applied Science and Engineering, Versailles, France, (1981).
- (40) Goussebaile, J., Labadie, G. and Lasbleiz, P., Resolution du Probleme de Stokes en Mecanique des Fluide par la Methode d'Uzawa Preconditionnee, 16eme Colloque d'Analyse Numerique, Guidel, France, (1983).
- (41) Hutton, A. G., Current Progress in the Simulation of Turbulent Incompressible Flow by the Finite Element Method, In: Taylor, C., Olson, M. D., Gresho, P. M. and Habashi, W. G. ed., Proc. Numer. Meth. in Laminar and Turbulent flow, Swansea, U. K., (1985), p. 289, Pineridge Press.
- (42) Okuda, H., Eguchi, Y. and Yagawa, G., Finite Element Turbulent Flow Analysis of Three-Dimensional Large-Scale Problem Using the $k-\epsilon$ Model, *Trans. Jpn. Soc. Mech. Eng.*, (in Japanese), (to appear).
- (43) Kuwahara, K., Direct Simulation of Transonic Flow Around an Airfoil, *J. Japan Soc. Aeronautical and Space Science*, (in Japanese), No. 34 (1986), p. 363.
- (44) Deardorff, J. W., A Numerical Study of Three-Dimensional Turbulent Channel Flow at Large Reynolds Number, *J. Fluid Mech.*, Vol. 41 (1970), p. 453.
- (45) Diaz, A. R., Kikuchi, N., Papalambros, P. and Taylor, J. E., 'Design of an Optimal Grid for Finite Element Methods, *J. Struct. Mech.*, No. 11 (1983), p. 215.
- (46) Löhner, R., Patnalk, G., Boris, J. P., Oran, E. S. and Book, D. L., Application of the Method of Flux-corrected Transport to Generalized Meshes, In: Zhuang, F. G. and Zhu, Y. L. ed., *Lecture Notes in Physics*, Vol. 264, (1986), p. 428, Springer-Verlag.
- (47) Gottlieb, D., Spectral Methods for Compressible Flow Problems, in: Soubbaramayer, F. and Boujot, J. P. eds., *Lecture Notes in Physics* Vol. 218 (1985), p. 48, Springer-Verlag.
- (48) Ku, H. C., Hirsh, R. S. and Taylor, T. D., A Pseudospectral Method for Solution of the Three - Dimensional Incompressible Navier - Stokes Equations, In: Zhuang, F. G. and Zhu, Y. L. ed., *Lecture Notes in Physics*, Vol. 264 (1986), p. 391, Springer-Verlag.
- (49) Nakahashi, K., FDM-FEM Zonal Approach for Computations of Compressible Viscous Flows, In: Zhuang, F. G. and Zhu, Y. L. ed., *Lecture Notes in Physics*, Vol. 264 (1986), p. 494, Springer-Verlag.
- (50) Hackbusch, W., On the Convergence of a Multigrid Iteration Applied to Finite Element Equations', Rept. 77-8, Math. Inst. Univ. Köln., (1977).
- (51) Brandt, A., Multi-level Adaptive Solutions to Boundary-Value Problems, *Math. Comp.*, No. 31 (1977), p. 333.
- (52) Brandt, A., 'Multi-Level Adaptive Finite Element Methods: I Variational Problems, NASA ICASE Research Rept., Vol. 79, No. 8 (1979).
- (53) Perez, E., Periaux, J., Rosenblum, J. P., Stoufflet, B., Dervieux, A. and Lallemand, M. H., Adaptive Full-Multigrid Finite Element Methods for Solving the Two-Dimensional Euler Equations, In: Zhuang, F. G. and Zhu, Y. L. ed., *Lecture Notes in Physics*, Vol. 264 (1986), p. 523, Springer-Verlag.