SOLUTION OF INCOMPRESSIBLE VISCOUS FLUID FLOW WITH HEAT TRANSFER USING ADINA-F

KLAUS-JÜRGEN BATHE† and JIAN DONG‡
† Massachusetts Institute of Technology, Cambridge, MA 02139, U.S.A.
‡ ADINA R&D, Inc., 71 Elton Avenue, Watertown, MA 02172, U.S.A.

Abstract—The finite element program ADINA-F for analysis of 2-D and 3-D incompressible fluid flow with heat transfer in laminar flow conditions is described. The governing continuum mechanics equations are summarized, the finite element formulation is given and some details on the implementations with the resulting analysis capabilities are presented. Finally, sample solution results that demonstrate some of the program capabilities are given.

I. INTRODUCTION

During recent years a very large research effort has been concentrated on the development of computational methods for the analysis of fluid flows [1, 2]. Although mostly finite difference techniques are used, finite element procedures have found increasing applications. The use of finite element techniques results in generality to represent complex geometries and can be effective in treating difficult boundary conditions. Surely, the success of finite element procedures in the analysis of structural and heat transfer problems points to excellent possibilities for the solution of fluid flow problems as well.

The objective in this paper is to present the current capabilities of the computer program ADINA-F which we have developed as part of the ADINA computer programs. ADINA-F is applicable to the solution of 2-D and 3-D viscous incompressible fluid flow with heat transfer assuming laminar flow conditions. In the next section we briefly summarize the continuum mechanics equations considered and then we give the corresponding finite element equations to be solved. In Section 4 we discuss the incremental/iterative solution schemes used in ADINA-F and in Section 5 we then present the results of some example solutions. Finally, we give in the conclusions some thoughts on further developments that we would like to undertake for the program.

2. CONTINUUM MECHANICS EQUATIONS

Using a Cartesian reference frame (xi, i = 1, 2, 3) the governing fluid flow equations solved are—using index notation and the usual summation convention [3]—

momentum:

\[ \rho \left( \frac{\partial \mathbf{u}_i}{\partial t} + \mathbf{u}_j \mathbf{u}_i, j \right) = \tau_{ij,j} + \mathbf{f}^p_i \quad (1) \]

constitutive:

\[ \tau_{ij} = -p \delta_{ij} + 2\mu \epsilon_{ij} \quad (2) \]

continuity:

\[ \nabla \cdot \mathbf{u}_i = 0 \quad (3) \]

heat transfer:

\[ \rho c_p \left( \frac{\partial \theta}{\partial t} + \mathbf{u}_i \theta, i \right) = (k \theta, j, j) + q^p \quad (4) \]

Here we have

- \( v_i \) = velocity of fluid flow in direction x_i
- \( \rho \) = mass density
- \( \tau_{ij} \) = components of stress tensor
- \( f^p_i \) = components of body force vector
- \( p \) = pressure
- \( \delta_{ij} \) = Kronecker delta
- \( \mu \) = fluid viscosity
- \( \epsilon_{ij} \) = components of velocity strain tensor
- \( c_p \) = specific heat at constant pressure
- \( \theta \) = temperature
- \( k \) = thermal conductivity
- \( q^p \) = rate of heat generated per unit volume.

The boundary conditions corresponding to eqns (1)-(4) are:

- prescribed fluid velocities, \( \mathbf{v}_i \), on the surface \( S_1 \),

\[ \mathbf{v}_i = \mathbf{v}_i \mid_{S_1} \quad (5) \]

- prescribed tractions, \( \mathbf{f}^p_i \), on the surface \( S_2 \),

\[ \tau_{ij} n_j = \mathbf{f}^p_i \mid_{S_2} \quad (6) \]
where the $n_j$ are the components of the unit normal to the fluid surface and the $f_{ij}$ are the components of the traction vector;

prescribed temperatures, $\delta$, on the surface $S_3$,

$$\delta = \frac{\partial \delta}{\partial n} |_{S_3}$$  \hspace{1cm} (7)

— prescribed heat flow into the surface $S_4$,

$$k \frac{\partial \theta}{\partial n} |_{S_4} = q^s$$  \hspace{1cm} (8)

where $q^s$ is the heat flux input to the body. We note that for eqns (5)-(8) $S = S_1 \cup S_2$, $\delta = S_1 \cap S_2$ and $S = S_3 \cup S_4$, $\theta = S_3 \cap S_4$, where $S$ is the total surface area of the fluid body.

The heat flow input in eqn (8) comprises: the effect of actually applied distributed heat flow; the effect of convection heat transfer, in which case

$$q^s = h(\theta_e - \theta^s)$$  \hspace{1cm} (9)

where $h$ is the convection coefficient, and $\theta_e$ and $\theta^s$ are the environmental and body surface temperatures; and the effect of radiation heat transfer, in which case

$$q^s = \kappa(\theta_e - \theta^s)$$  \hspace{1cm} (10)

$$\kappa = h_e[(\theta_e^2 + (\theta^s)^2)(\theta_e + \theta^s)]$$  \hspace{1cm} (11)

and $h_e$ is the radiation coefficient and $\theta_e$ is the temperature of the radiating body [4]. The value of $h_e$ is determined from the Stefan-Boltzmann constant, the emissivity of the radiating and absorbing materials and the geometric view factors.

The above equations are the standard Navier-Stokes equations governing the motion of a viscous, incompressible fluid in laminar flow with heat transfer. Inherent nonlinearities are due to the convective terms in eqns (1) and (4) and the radiation boundary condition in eqn (8). In our solution we allow further for the following nonlinearities:

— The viscosity coefficient can depend on the temperature and on the velocity strain.

— The specific heat $c_p$, conductivity $k$ and convection coefficient $h$ can depend on the temperature.

3. FINITE ELEMENT GOVERNING EQUATIONS

The finite element solution of the continuum mechanics equations governing the fluid flow is obtained by establishing a weak form of the equations using the Galerkin procedure [3]. The momentum equations are weighted with the velocities, the continuity equation is weighted with pressure and the heat transfer equation is weighted with the temperature. Using integration by parts gives the variational equations to be discretized by finite element interpolations:

momentum:

$$\int_{V} \rho (\dot{u}_i + v_j u_j) \delta v_i \, dV + \int_{V} \tau_{ij} \delta e_{ij} \, dV$$

$$= \int_{S_3} f^p \delta v_i \, dV + \int_{S_3} f^s \delta u_i \, dS$$  \hspace{1cm} (12)

where $\delta$ means 'variation in' (or a virtual quantity);

continuity:

$$\int_{V} \delta p \, v_i \, dV = 0;$$  \hspace{1cm} (13)

heat transfer:

$$\int_{V} \rho c_p \delta \theta + v_i u_j \delta \theta \, dV + \int_{V} \kappa \delta \theta + \delta^s \, dV$$

$$= \int_{S_4} q^p \delta \theta \, dV + \int_{S_4} \delta \theta \, q^s \, dS + \int_{S_4} \delta \theta \kappa(\theta_e - \theta^s) \, dS$$

$$+ \int_{S_4} \delta \theta \kappa(\theta_e - \theta^s) \, dS$$  \hspace{1cm} (14)

where $S_4$, $S_e$ and $S_s$ are part of $S_4$ and correspond, respectively, to the surfaces subjected to heat flow input, convection and radiation conditions.

We note that the essential boundary conditions in eqns (5) and (7) need be satisfied by the finite element interpolation assumptions, whereas the natural boundary conditions in eqns (6) and (8) are incorporated in the terms on the right-hand sides of eqns (12) and (14).

Consider next a single finite element for solution of eqns (12)-(14). The finite element equations for an assemblage of elements are obtained by the usual direct addition of element matrices [3].

For the finite element discretization we use for 2-D (planar and axisymmetric) solutions the 9/4 node element shown in Fig. 1:

— the nine nodes are used to interpolate the geometry, the velocities and the temperature.

$$x_2 = \sum_{k=1}^{9} h_k x_k, \quad x_3 = \sum_{k=1}^{9} h_k x_k$$  \hspace{1cm} (15)

$$v_2 = \sum_{k=1}^{9} h_k v_k, \quad v_3 = \sum_{k=1}^{9} h_k v_k$$  \hspace{1cm} (16)

$$\theta = \sum_{k=1}^{9} h_k \theta_k$$  \hspace{1cm} (17)
Solution of incompressible viscous fluid flow

Correspondingly, we use in 3-D analysis the 27/8 node element, see Fig. 1, for a triquadratic interpolation of the geometry, velocities and temperature and a trilinear interpolation of the pressure.

These interpolations provide stable elements as expressed by the inf-sup condition of Brezzi and Babuška [5].

The finite element governing equations are now established as usual,

\[ M_{v2} \dot{v} + (K_{vv} + K_{wp}) v + K_{vp} p = R_B + R_S \]  
\[ \dot{p} = 0 \]  
\[ C \theta + K_{\theta \theta} \theta + K_{\theta v} v = Q_B + Q_S + Q_\tau + Q_r \cdot \]

Here, for example, in 2-D planar flow analysis, for a single element,

\[
\begin{bmatrix}
M_{v2} & zeros \\
zeros & M_{v3}
\end{bmatrix}
\begin{bmatrix}
v_2 \\
v_3
\end{bmatrix}
+ \begin{bmatrix}
K_{vv2} & 0 & 0 & K_{vp2} \\
0 & +K_{vv3} & 0 & 0 \\
K_{vp2} & K_{vp3} & 0 & 0 \\
K_{vp3} & 0 & 0 & K_{v3}
\end{bmatrix}
\begin{bmatrix}
v_2 \\
v_3 \\
p \\
\theta
\end{bmatrix}
= \begin{bmatrix}
R_{B2} + R_{S2} \\
R_{B3} + R_{S3} \\
Q_B + Q_S + Q_\tau + Q_r
\end{bmatrix}
\]
\[
R_S = \int_{S_2} \mathbf{H}^T \mathbf{Y}^S \, dS
\] (29)

\[
C = \rho \int_V c_p \mathbf{H}^T \mathbf{H} \, dV
\] (30)

\[
K_h = \int_V k(\mathbf{H}^T_{x_2} \mathbf{H}_{x_2} + \mathbf{H}^T_{x_3} \mathbf{H}_{x_3}) \, dV
\] (31)

\[
K_{\nu z} = \rho \int_V c_p \mathbf{H}^T_{x_2} \mathbf{H}_{x_2} \, dV
\] (32)

\[
K_{\nu y} = \rho \int_V c_p \mathbf{H}^T_{x_3} \mathbf{H}_{x_3} \, dV
\] (33)

\[
Q_h = \int_V \mathbf{H}^T \mathbf{q} \, dV
\] (34)

\[
Q_S = \int_{S_1} \mathbf{H}^T \mathbf{q}^S \, dS
\] (35)

\[
Q_c = \int_{S_1} \mathbf{H}^T \mathbf{q}^c \, dS
\] (36)

\[
Q_l = \int_{S_1} \mathbf{H}^T \mathbf{q}^l \, dS
\] (37)

Here we should note that \( S_1 \) contains the components

\[
J_p = -p + \mu \frac{\partial u_p}{\partial n} \quad \text{and} \quad J_t = \mu \frac{\partial u_t}{\partial n},
\]

with \( u_p \) and \( u_t \) being the normal and tangential boundary velocities. To arrive at eqns (24) and (29) we used the continuity condition, eqn (3), on the momentum equation, eqn (1) or eqn (12).

In eqns (23)-(37) we use

\[
\mathbf{H} = [h_1 h_2 \cdots h_9]
\] (38)

\[
\mathbf{H} = [h_1 h_2 h_3 h_4]
\] (39)

and the matrix \( \mathbf{H}^S \) interpolates velocities on the element surfaces. The ordering of the nodal point state variables is as follows:

\[
\begin{bmatrix}
  u_1^1 \\
  u_2^2 \\
  \vdots \\
  u_3^3 \\
  v_3^3 \\
  v_2^2 \\
  \vdots \\
  v_1^1
\end{bmatrix} =
\begin{bmatrix}
  \theta_1 \\
  \theta_2 \\
  \vdots \\
  \theta_3
\end{bmatrix} =
\begin{bmatrix}
  p_1 \\
  p_2 \\
  p_3 \\
  p_4
\end{bmatrix}.\) (40)

The finite element matrices for axisymmetric flow and 3-D flow are straightforward extensions of the above relations [3].

It should be noted that eqns (19)-(21) are simply the finite element representations of eqns (12)-(14).

These equations are highly nonlinear in general and must be solved by some incremental/iterative scheme as discussed in the next section.

4. EQUATION SOLUTION STRATEGIES

Equations (19)-(21) are the finite element equations that express the conditions of momentum balance, continuity and heat flow balance at any time during the solution. In ADINA-F the user can specify the use of either the method of successive substitution or Newton-Raphson iteration for the incremental iterative solution of these equations.

To obtain the solution we employ time as a variable, where time is merely used to specify the load level in a steady-state analysis. For the ADINA-F solution, variable time step sizes can be specified as indicated in Fig. 2.

Assume that the solution has been obtained for time \( t \) and that next the solution is to be calculated for time \( t + \Delta t \), where \( \Delta t \) is the time step size. Since the initial conditions define the solution at time 0, the algorithm used to calculate the response at time \( t + \Delta t \) is the basic procedure to obtain, successively, the solution at any of the discrete times specified by the user.

In the discussion to follow we use the notation of [3], where, for example, a left superscript \( t \) on a variable denotes that variable at time \( t \).

For the time integration in ADINA-F we use the \( \alpha \)-method with \( \frac{1}{4} \leq \alpha < 1 \) [3]. Hence, we have in transient analysis

\[
in + \Delta t \theta = \frac{1}{\Delta t} (i + \Delta \theta - \theta)
\] (41)

\[
in + \Delta t \psi = (1 - \alpha) \psi + \alpha \Delta \psi
\] (42)

and

\[
in + \Delta t \psi = \frac{1}{\Delta t} (i + \Delta \theta - \theta)
\] (43)

\[
in + \Delta t \theta = (1 - \alpha) \theta + \alpha \Delta \theta
\] (44)
Solution of incompressible viscous fluid flow

\( \rho = (1 - \alpha)\rho + \alpha \rho \Delta t. \) (43)

Usually \( \alpha = \frac{1}{2} \) (trapezoidal rule) or \( \alpha = 1 \) (Euler backward method).

In the equations below we have \( \alpha \) to be specified. Also, in steady-state analysis the equations are used by setting \( \alpha = 1 \) and dropping the terms corresponding to the time derivatives.

In the method of successive substitution the governing equations are solved by evaluating the residual and using the coefficient matrices in eqns (19)-(21) to obtain the next iterated values in the velocities, pressures and temperatures.

The equations used are:

\[
\frac{1}{\Delta t} M_v r + \Delta \phi = \Delta \rho + \alpha \Delta \rho
\] (44)

and

\[
K_{rr} r + \Delta \phi = 0
\] (45)

Once the values \( r, \theta, \phi \) and in all cases the superscripts \( t \pm \Delta t \) and \( (i - 1) \) on the matrices denote that the matrices are evaluated using the last calculated state variables,

\[
r + \Delta \rho r' = (1 - \alpha)r' + \alpha \Delta \rho
\] (51)

\[
r + \Delta \rho p' = (1 - \alpha)p' + \alpha \Delta \rho
\] (52)

The initial conditions in the iterations are

\[
r + \Delta \rho (0) = r', \quad p + \Delta \rho (0) = p', \quad \theta + \Delta \rho (0) = \theta'.
\] (53)

In the Newton–Raphson iteration the gradient of the governing equations to be solved is introduced. If we write eqns (19)-(21) combined in the form

\[
F = R
\] (54)

the Newton–Raphson iteration is

\[
r + \Delta t K_{rr} \Delta \rho (i) = r + \Delta t R - r + \Delta t \Delta F(i - 1)
\] (55)

where

\[
\Delta U^0 = \left[ \Delta v^0 \Delta \rho^0 \Delta \phi^0 \right], \quad r + \Delta t \Delta U^0 = r + \Delta t R + \Delta U^0.
\] (56)

In comparison to the use of successive substitutions we solve now for the increments in the state variables. The matrix \( r + \Delta t K_{rr} \) contains the coefficient matrices of eqns (44)-(46) and the coefficient matrices \( r + \Delta t J_{rr} \) and \( r + \Delta t J_{\theta} \), which we show in the following equations:

\[
\frac{1}{\Delta t} M_v + \alpha r + \Delta \rho K_{rr} = \alpha \Delta \rho
\] (46)

\[
\frac{1}{\Delta t} \Delta v^0 = \frac{\partial F}{\partial U}
\] (57)

\[
\Delta v^0 = K_{pp} \Delta v^0 = - K_{pp} \Delta v^0
\] (58)
The use of such gradients may well help to decrease
the total cost of solution and will be addressed
in our future work.

5. SAMPLE SOLUTIONS

We present the following solution results to indicate
some of the capabilities of ADINA-F. No attempt was made to use in these solutions optimal
meshes or optimal solution approaches, nor to make
complete studies of each of the problems considered.

Flow in a converging channel

Figure 3 shows the problem we consider [6]. For
the ideal Hamel problem (with infinite plates), the
velocity profile is self-similar at all radii and each
fluid particle moves along radial lines towards the
source that exists at the plate juncture. In finite
element analysis, only a finite length of the wedge
geometry can be modeled. The boundary conditions
for this problem are that the velocity is applied on
the upstream end of the wedge and zero tractions are
applied on the downstream end. For the solution
presented here, the inflow boundary is taken to be at
the radius \( r = 4 \), and the outflow boundary at the
radius \( r = 0.25 \). Since the flow between the plates is
symmetric, only one half of the domain is modeled
with a symmetry boundary condition applied at the
center line of the wedge. No slip velocity conditions
are prescribed at the plate wall.

The Reynolds number for this problem is defined
by

\[
Re = \frac{av_0}{\nu} = \frac{\rho v_0}{\mu}
\]

where \( v \) is the kinematic viscosity, \( \alpha \) is the wedge
half-angle, \( r \) the radial coordinate and \( v_0 \) the velocity
along the centerline at \( r \).

For \( Re = 61 \) for the finite element solution.

Flow past a cylinder

We consider a cylinder placed in a uniform flow
field, see Fig. 4. The flow is irrotational far from the
cylinder, but the fluid develops a boundary layer
flow near the cylinder due to viscous effects. Whether
the boundary layer detaches from the cylinder
depends on the Reynolds number [7]

\[
Re = \frac{\rho D}{\mu}
\]

where \( D \) is the diameter of the cylinder.

The finite element model and boundary conditions
used are shown in Fig. 4. Due to symmetry condi-
tions, only one half of the domain is considered. We

\[
\left( \frac{1}{\Delta t} + \alpha + \Delta t \right) C^{(i-1)} + \alpha + \Delta t \right) A^{(i)}
\]

and

\[
J_{\tau v_2} = \int \rho H^T H_2 v_2 H \, dV \tag{60}
\]

\[
J_{\tau v_3} = \int \rho H^T H_3 v_3 H \, dV \tag{61}
\]

\[
J_{\tau v_3} = \int \rho H^T H_3 v_3 H \, dV \tag{62}
\]

\[
J_{\tau v_2} = \mu \int \rho H^T H_2 v_2 H \, dV \tag{63}
\]

\[
J_4 = \mu \int c_\beta (H^T H \nu_1 H_2) \, dV \tag{64}
\]
Fig. 3. Analysis of Hamel problem. (a) Problem considered. (b) Mesh used. (c) Velocity field, Re = 61. (d) Centerline velocity. (e) Wall minus centerline pressure.
KLAUS-JÜRGEN BATHO and JIAN DONG

show the finite element solutions for Re = 20 and 120. The Re = 120 solution displays counter-rotating flow behind the cylinder.

Figure 4 also shows pressure contours for the two flow conditions considered. These indicate that the mesh used is rather coarse for the case Re = 120.

Flow in nozzle passage

In this problem solution we simulate the flow between the blades of a nozzle. We assume that this nozzle is one of many arranged in a circumferential manner [8]. Figure 5 shows the domain considered.

To ensure the periodicity in the finite element analysis we employ constraint equations to set the velocities of flow equal along the lines A-A' and B-B'.

The Reynolds number used in this problem is

\[ \text{Re} = \frac{\rho v_0 L}{\mu} \]

where \( L \) is the 'length' of the blade.

We analyzed the problem for Re = 100 and Re = 400. Figure 5 shows the predicted velocity fields and pressure contours.

---

**Fig. 4. (a)-(c).**
Natural convection in a square cavity

This problem, described in Fig. 6, has been considered extensively by researchers, for numerical solutions [9]. The fluid in the square cavity is subjected to gravity and the vertical walls are at different temperatures causing a natural convection. The right wall is at temperature $\theta_r$ and the left wall is at temperature $\theta_l$. For the solution we use the Boussinesq approximation which assumes that the density of the fluid changes with its temperature (for the load calculation) but the flow of the fluid is still governed by the incompressible flow equations. All the equations are coupled because the momentum equations are driven by the buoyancy term, which is a function of temperature. The body force term introduced into the $z$-momentum equation is

$$\rho \beta \theta (\theta - \theta_0)$$

---

Fig. 4. Analysis of flow around a cylinder. (a) Problem considered (including the symmetry conditions). (b) Mesh used. (c) Velocity field, Re = 20. (d) Velocity field, Re = 120. (e) Pressure contours, Re = 20. (f) Pressure contours, Re = 120.
Fig. 5. (a)-(c).
Fig. 5. Analysis of flow in a nozzle passage. (a) Problem considered. (b) Mesh used. (c) Velocity field, $Re = 100$. (d) Velocity field, $Re = 400$. (e) Pressure contours, $Re = 100$. (f) Pressure contours, $Re = 400$. 
where
\[ \beta = \text{coefficient of thermal expansion} \]
\[ \theta_0 = \text{reference temperature} = (\theta_h + \theta_c)/2 \]
\[ \rho_0 = \text{density at reference temperature} \]

The problem is characterized by a Rayleigh number
\[ \text{Ra} = \frac{\rho \beta g \Delta \theta L^3}{\mu \kappa} \]
and the Prandtl number
\[ \text{Pr} = \frac{\mu}{\kappa} \]

where
\[ L = \text{depth of the cavity} \]
\[ \kappa = \text{thermal diffusivity} = \frac{k}{\rho c_p} \]
\[ \Delta \theta = \theta_h - \theta_c \]

A 16 x 16 element mesh (with equal size elements) was selected to reasonably resolve both the thermal and velocity boundary layers that develop near the walls. The problem was solved for the Rayleigh numbers of \(10^3, 10^4, 10^5\) and \(10^6\) each time with \(Pr = 1.0\), using the steady-state equations.

Figure 6 shows the velocity fields and temperature and pressure contours predicted in the finite element solution. Also shown is a comparison of the ADINA-F solution with the solution given by Taylor and Ijam [9].

**Flow in a closed channel**

We consider the laminar steady-state flow at a uniform temperature in a closed channel of sides \(2h\), see Fig. 7. Due to symmetry conditions only one-quarter of the channel fluid is represented by 3-D finite elements, with one element layer in the (flow) \(y\)-direction and three element layers each in the \(x\)- and \(z\)-directions. All elements are of equal size.

Figure 7 shows the calculated velocity profile and compares the profile with the solution for flow through a channel of infinite width in the \(x\)-direction.

**6. CONCLUDING REMARKS**

The objective in this paper was briefly to present the capabilities of the newly developed computer program ADINA-F. The program can be used to analyze viscous laminar 2-D and 3-D flow conditions with heat transfer assuming incompressible flow conditions.

Although a number of important capabilities are already available in this first version of ADINA-F,
Fig. 6. (b)-(c).
Fig. 6. Analysis of natural convection in a square cavity. (a) Problem considered. (b) Velocity fields, \(Ra = 10^3\)\(-10^6\), \(Pr = 1\). 
(c) Temperature contours, \(Ra = 10^3\)\(-10^6\), \(Pr = 1\). (d) Pressure contours, \(Ra = 10^3\)\(-10^6\), \(Pr = 1\). (e) Vertical velocity along horizontal centerline of cavity, case \(Pr = 1000\). (f) Temperature along horizontal centerline of cavity, case \(Pr = 1000\).
we plan to undertake additional developments in the following major areas:

— The program should contain more finite elements, including mixed interpolated elements [10] and spectral finite elements [11].
— Improvements in the time integration schemes (explicit/implicit) would be attractive.
— A library of material models for non-Newtonian flow simulation should be implemented.
— The program should have automatic procedures which select time step sizes and iteration techniques largely without user intervention.

In addition, ADINA-F should be fully coupled to the pre- and post-processors ADINA-IN and ADINA-PLOT, which in turn should be enhanced for quantities specific to fluid flow analysis (such as the evaluation and plotting of stream lines).

These are some developments that we would like to pursue in the near future. However, the field of fluid flow analysis is very rich and undoubtedly many further ideas to render ADINA-F more practical in actual analyses will arise. All these developments should, of course, be undertaken with a view towards the new hardware/computing equipment that will become available.

REFERENCES